Design Selection and Numerical Simulation Analysis of Diesel Engine Intercoolers

Wang Xuyan

School of Energy and Power Engineering, North China University of Technology, Taiyuan, Shanxi, China

Abstract: Addressing the coupled trade-off between heat transfer efficiency and flow resistance in heavy-duty diesel engine intercoolers, this study focuses on engines meeting China VI emission regulations. Employing a progressive design approach—"structure selection, layout optimization, simulation validation"—research was conducted using Converge for detailed numerical simulation. First, the comprehensive performance of two intercooler configurations—Type A (coolant inside tubes, gas outside tubes) and Type B (gas inside tubes, coolant outside tubes)—was compared. Based on thermal performance (higher turbulent heat transfer coefficient for Type B) and engineering adaptability (flexible layout, easy maintenance), Type B was selected as the baseline design. Subsequently, a 3D model was established using SolidWorks. Numerical simulations employing the RNG k-ε turbulence model and conjugate heat transfer (CHT) model validated the independence of the 1.8 million mesh density. Simulation results indicate that the core section temperature drop reaches 91K, approaching the target of 90 ℃. However, heat accumulation occurs in the inlet section, with low-speed vortex zones present in the intake chamber and curved outlet piping sections, accompanied by uneven internal airflow distribution. Finally, improvement measures such as optimizing the inlet guide structure and adjusting the heat exchange tube layout are proposed to enhance heat transfer efficiency and flow uniformity, reduce friction loss, and meet the energy-saving and consumption-reduction requirements of diesel engines.

Keywords: Diesel engine intercooler; Structural selection; Numerical simulation; Heat transfer characteristics; Flow resistance

1. Introduction

the current technical dilemma in intercoolers lies in the trade-off between heat transfer efficiency and flow resistance—enhancing heat exchange typically requires increasing heat transfer area or adding flow disturbance structures, which inevitably increases flow resistance and raises engine intake losses[2]. Conversely, simplifying the structure to reduce resistance leads to insufficient heat exchange, failing to meet cooling demands. This contradiction is particularly pronounced in heavy-duty diesel engines[3] . Existing research primarily focuses on optimizing individual structural parameters. Zhao Changpu et al. [1] experimentally validated intercooler cooling performance under different EGR rates but did not conduct quantitative comparisons of multiple core design configurations. [6]. Hu Xingjun et al. investigated the impact of cooling tube structures on air-cooled intercooler performance[7], but their conclusions pertain to air-cooling scenarios and cannot be directly applied to water-cooled intercooler design.

This paper focuses on heavy-duty diesel engines meeting China VI emission regulations. Adopting a progressive design approach of "structure selection-layout optimization-simulation verification, " it employs Converge for detailed numerical simulation to systematically compare the comprehensive performance of different structural configurations. This leads to a proposed solution balancing high heat transfer efficiency with low resistance characteristics.

2. Intercooler Structural Design and Simulation Methodology

2.1 Core Configuration Selection

Two intercooler configurations were designed: Type A (Figure 1), where cooling water flows inside the condensing tubes connected to inlet/outlet ports, while high-temperature compressed gas flows externally in the shell-side space; and Type B (Figure 2), where the gas flow configuration is reversed. The second type (hereafter referred to as Type B) employs the opposite configuration. Compressed gas is conveyed through the interior of the condenser tubes, with the tube inlets and outlets connected to the compressor gas piping of the intercooler. Cooling water flows within the jacket or channels surrounding the condenser tubes, forming a cooling medium flow field around them.

From a thermal performance perspective, the B-type configuration offers significant advantages: when compressed gas flows through the tubes, optimizing tube diameter and velocity creates sufficient turbulence (Reynolds number Re>10000), boosting the heat transfer coefficient by 3-5 times compared to laminar flow. In contrast, the A-type configuration features cooling water flowing through the tubes at lower velocities (typically Re<5000), resulting in weaker heat exchange efficiency. Additionally, the outer cooling water side of the B-type configuration can incorporate flow-disturbing structures such as baffles to further enhance external heat transfer.

In terms of engineering adaptability, the gas flow channel in the Type B configuration is located inside the pipe, allowing the piping to be bent and arranged according to the engine compartment space for greater adaptability. During maintenance, the external cooling water channel is less prone to dust accumulation, and cleaning does not require disassembly of the gas piping, offering higher convenience. the Type A configuration, however, faces challenges with dust accumulation on the gas side, which is difficult to clean and may lead to degraded heat transfer performance over extended use. Considering both thermal performance and engineering adaptability, the B-type configuration was selected as the baseline for subsequent optimization.

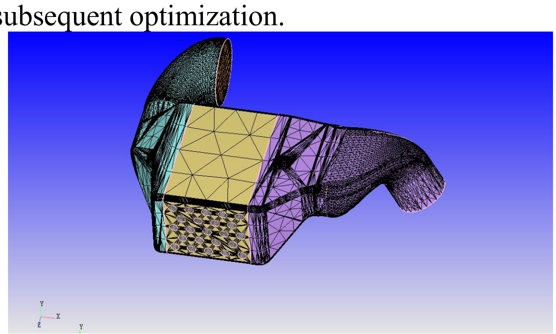


Figure 1. Intercooler A

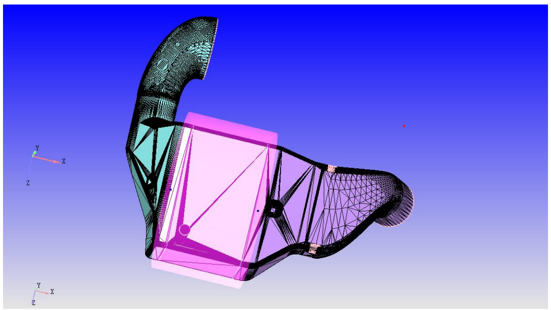


Figure 2. Intercooler B

2.2 Simulation Model Development

A three-dimensional solid model of the intercooler was created using SolidWorks, encompassing the inlet chamber, core assembly (heat exchange tubes and tube sheet), and outlet chamber. During modeling, structures such as bolts and flanges were simplified to reduce the number of mesh elements. the model was imported into Converge for mesh generation, as detailed in Figure 4. the RNG k-ε turbulence model was selected for the physical model. the conjugate heat transfer (CHT) model was employed: (air density set to 1.225 kg/m³, specific heat capacity to 1005 J/(kg·K); cooling water density set to 997 kg/m³, specific heat capacity to 4186 J/(kg·K)). Solution settings: Employed the semi-implicit pressure-coupled equation scheme (SIMPLEC). Pressure terms were discretized using second-order upwind schemes, while energy and momentum equations used first-order upwind discretization. Convergence criteria were set to a residual curve decrease of three orders of magnitude and an inlet-outlet mass flow difference ⩽ 0.5% to ensure stable computational convergence. Mesh independence verification: Three mesh density schemes (1.2 million, 1.8 million, 2.4 million) were constructed to compare heat transfer coefficients and pressure drops for the rotary triangular arrangement. Results show that the heat transfer coefficient deviation between 1.8 million and 2.4 million meshes is only 1.2%, with a pressure drop deviation of 0.8%. This indicates that the 1.8 million mesh density satisfies independence requirements, and subsequent simulations will adopt this mesh density.

3 Simulation Results and Analysis

3.1 Temperature Field and Flow Characteristics Analysis

Figure 5 illustrates the temperature distribution in the intercooler: the inlet side (red on the right) is at 450K (177°C). After heat exchange through the condenser tubes (vertical pipes), the core region (yellow-green) cools to 320K (47°C), achieving a total temperature difference of 130K. Specifically, the core section with dense condenser tubes achieves a temperature drop of 91K, closely matching the planned 90°C target and confirming the core heat transfer efficiency of the condenser tubes. However, the inlet section experiences heat accumulation due to ineffective flow guidance toward the condenser tubes. This requires optimizing the inlet flow path (e. g., adding baffles) to enhance initial heat transfer, reduce thermal stagnation, and improve cooling uniformity.

Figure 6 shows the temperature profile of the intake chamber. As illustrated, the entire chamber appears red. Based on the diagram on the right, it is evident that the temperature distribution ranges from 450K (177°C) to 424K (150°C), with a temperature difference of 26K. This trend aligns with the temperature distribution shown in Figure 5, providing mutual corroboration.

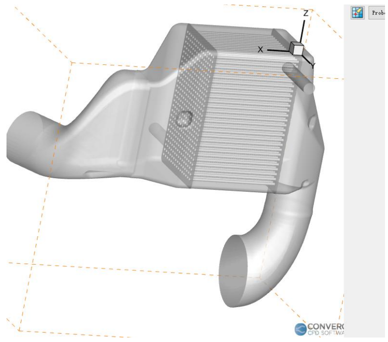


Figure 3. Intercooler Integral Model

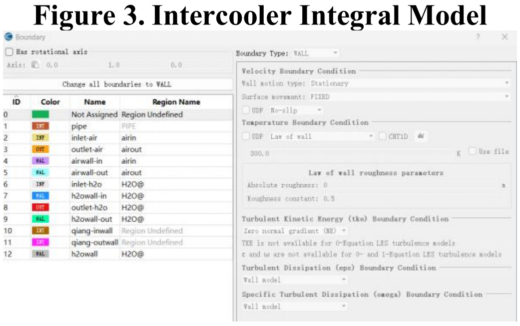


Figure 4. Mesh Generation

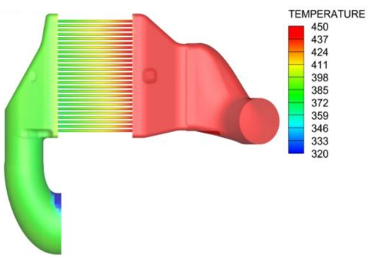


Figure 5. Temperature Contour

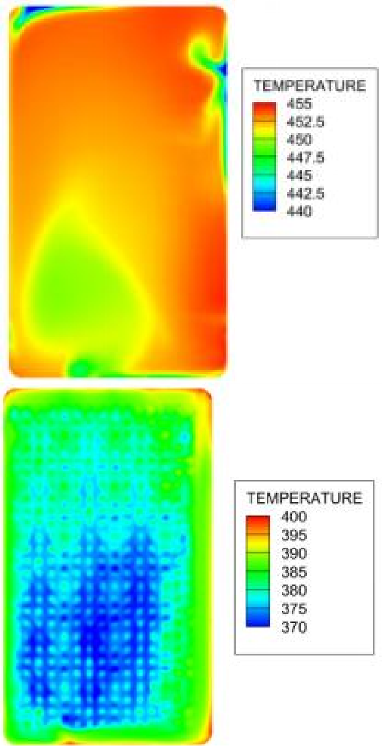


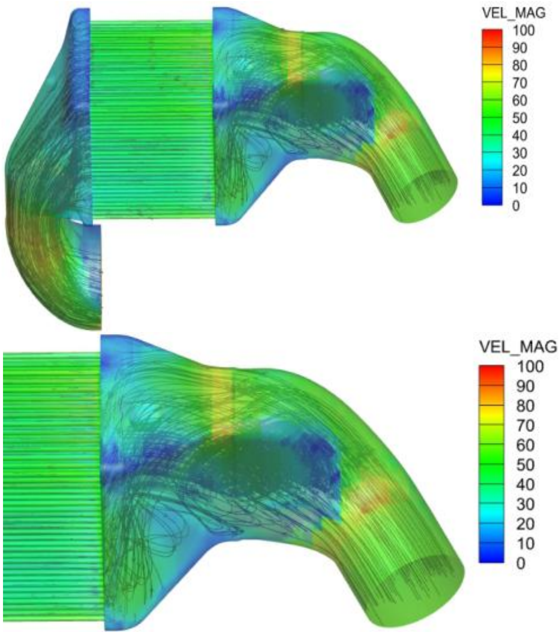
Figure 6. Inlet & Outlet Temperature Profile

3.2 Velocity Field Analysis

the overall velocity contour map reveals a distinct velocity distribution difference between the intake chamber and core regions after the airflow enters the intercooler from the duct. When high-speed airflow enters the intake chamber, influenced by changes in flow path geometry (such as curved structures and cross-sectional expansions), complex flow patterns form within the chamber due to the combined effects of inertia and flow path constraints. Within the core region (represented by the green, regular flow path), the airflow velocity distribution is relatively uniform, demonstrating the core's airflow rectification effect. In contrast, the curved sections of the intake chamber and outlet duct exhibit pronounced low-velocity vortex zones (blue and cyan areas) due to changes in flow direction, reflecting energy dissipation and flow distortion within irregular passages.

Focusing on the velocity contour map of the intake chamber in Figure 8 allows detailed analysis of flow dynamics within the chamber: Air entering through the inlet is constrained by the chamber's curved geometry and spatial topology, inducing multi-zone vortex structures throughout the chamber. In the outer regions of the bends, fluid inertia causes the flow to impact the walls and form reverse currents. These reverse currents collide with the subsequent main flow, creating concentrated zones of low-speed vortices (indicated by blue and cyan distributions). Simultaneously, a portion of the flow proceeds directly toward the central channel of the core, resulting in a decay of flow velocity in the peripheral channels of the core (as shown by the blue-green regions on both sides of the core). This non-uniform velocity distribution visually illustrates the mechanism by which the intake chamber's flow path geometry influences airflow dynamics—a synergistic interaction between fluid inertia and flow path constraints, shaped by the combined effects of the curved path and spatial structure.

Based on the velocity field analysis in Figures 7 and 8, the curved passages and cross-sectional geometry of the intercooler inlet chamber are the core elements driving velocity field distortion and vortex formation. As air flows through the inlet chamber, constrained by these structures, the interaction between inertia and flow path boundaries generates multi-region vortices and velocity differences. These velocity field characteristics clearly reveal the intake chamber structure's regulatory mechanism over airflow dynamics, providing quantitative basis for structural optimization (e. g., adding flow-guiding components or adjusting bend curvature). By weakening vortex intensity and balancing core flow velocity distribution, airflow uniformity within the intercooler flow path can be enhanced, laying a solid foundation for synergistic optimization of thermal and flow performance.



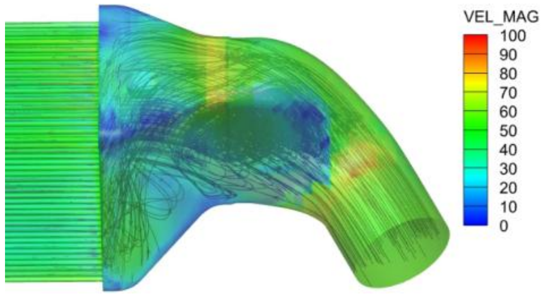


Figure 7. Velocity Streamline Contour

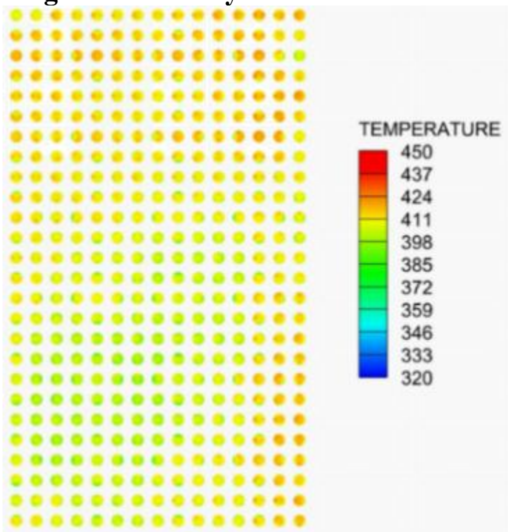


Figure 8. Outlet Condenser Tube Temperature Contour Plot

4 Conclusions and Optimization Recommendations

Simulation results indicate that the internal friction loss of the intercooler exhibits a pattern of "high inlet pressure, high core resistance, and low outlet pressure. " the inlet region forms a high-pressure zone due to abrupt geometric changes in the flow path. the central core section experiences a significant drop in pressure gradient due to the influence of cooling tubes. the outlet region exhibits low resistance owing to its regular flow path geometry. Regarding the temperature field, the temperature drop in the core section reaches 91K, closely approaching the planned 90°C target. However, heat accumulation occurs in the inlet section, necessitating optimization of the inlet flow path. Additionally, the gas mass flow rate rises sharply in the initial stage before gradually stabilizing. Velocity field analysis reveals low-velocity vortex zones in the inlet plenum and curved sections of the outlet piping. Temperature field uniformity analysis indicates uneven vertical airflow distribution within the intercooler, which impacts cooling efficiency.

Based on these findings, further improvements to the intercooler's heat transfer efficiency and flow uniformity can be achieved by optimizing the inlet guide structure, adjusting the heat exchange tube arrangement, and adding guide components. These measures will reduce friction losses along the flow path, meeting the requirements for energy conservation and consumption reduction in diesel engines.

References

[1] Zhao, C. P., Zhu, Y. Y., Li, Y. L., et al. Design and Testing of Venturi Tube for EGR System of Turbocharged Intercooled Diesel Engine [J]. Transactions of the Chinese Society for Agricultural Engineering, 2018, 29(15):49-56.

[2] Hu Pan, An Conghui, Qin Wanli, et al. Integrated Design and Validation of Water-Cooled Intercooler for Range Extender [J]. Small Internal Combustion Engines and Vehicle Technology, 2020, 49(6):50-53.

[3] Zhang Zhongyuan, Zhang Fengli, Guo Junjun, et al. Finite Element Static and Dynamic Analysis of Air Cooler Bracket Based on Lightweight Design [J]. Mechanical Design and Manufacturing, 2021, 38(01):120-123.

[4] Sheikholeslami M, Farshad S A. Nanoparticle Transportation with Turbulent Regime through a Solar Collector with Helical Tapes [J]. Advanced Powder Technology, 2022, 33(3):1062-1075.

[5] Mashhour A, Mohammad R S. Non-isothermal hydrodynamic characteristics of a nanofluid in a fin-attached rotating tube bundle [J]. Mathematics, 2021, 9(10):1153.

[6] Liu, S., & Zhang, Y. (2020). Research On the Integrated Intercooler Intake System of Turbocharged Diesel Engine. International Journal of Automotive Technology, 21(2), 339-3 Air-Cooled Intercoolers [J]. Journal of Jilin University (Engineering Science), 2021, 51(6):1933-1942.49.

[7] Hu Xingjun, Zhang Jinglong, Luo Yufei, et al. Influence of Cooling Tube Structure and Air Inflow Direction on the Performance of Air-Cooled Intercoolers [J]. Journal of Jilin University (Engineering Science), 2021, 51(6):1933-1942.

