

Thermal-Flow Stabilization Design of a Coolant Heater for Electric Vehicles

-CFD Based Comparative Analysis of Flow Channel Configurations-

Daeseong Han ¹⁾ Junghan Kim ¹⁾ Daenyeon Kim ¹⁾ Ilhwan Yi ²⁾ Jaehwan Jeong ³⁾

1) Gyeongbuk Institute of IT Convergence Industry Technology

1, Gongdan 9-ro 12-gil, Jinryang-eup, Gyeongsan-si, Gyeongsangbuk-do 38463, Republic of Korea (E-mail: dshan@gitc.or.kr)

2) Kumoh National Institute of Technology

61, Daehak-ro, Gumi-si, Gyeongsangbuk-do 39177, Republic of Korea

3) DHG Co., Ltd.

190, Gukgasandan-daero 39-gil, Guji-myeon, Dalseong-gun, Daegu 43011, Republic of Korea

KEY WORDS: Electric Vehicle, Coolant Heater, Thermal Management, CFD, Heat Transfer, Flow Channel Design

The thermal management system of electric vehicles requires rapid and stable coolant heating to improve battery warm-up and cabin-heating performance under low-temperature conditions. Because the coolant directly absorbs heat generated by the heating plate, the internal flow-channel configuration significantly affects coolant circulation, heat transfer performance, and thermal stability. Poor coolant distribution may generate stagnant regions around the heating surface, potentially resulting in localized overheating and an increased risk of coolant boiling. Therefore, optimizing the internal flow channel is essential for improving the thermal performance of high-voltage coolant heaters.

This study compares three internal flow-channel configurations using computational fluid dynamics (CFD) to investigate coolant flow behavior and thermal characteristics and to identify an effective flow-channel design.

CFD simulations were performed using ANSYS Fluent under identical operating conditions of 8 kW heating power, 10 L/min coolant flow rate, and an inlet coolant temperature of 273.15 K. Three configurations were investigated: Design #1 without internal fins, Design #2 with housing-mounted fins, and Design #3 with heating-plate-mounted fins.

The SST k- ω turbulence model was employed to accurately predict turbulent coolant flow and near-wall heat-transfer characteristics. To compare the temperature distributions under identical visualization conditions, the heating plate and coolant temperature contours were displayed within 273–1273 K and 273–290 K, respectively. Thermal-flow performance was evaluated using temperature contours, streamline distributions, and maximum coolant velocity.

Figure 1 compares the coolant temperature distributions of the three flow-channel configurations. Design #1 exhibited a maximum coolant velocity of 1.916 m/s. Because no internal flow-guiding structure was applied, coolant circulation was concentrated along one side of the chamber, resulting in a large stagnant region and a non-uniform coolant temperature distribution. Consequently, heat transfer around the heating plate was limited, increasing the possibility of local heat accumulation. Design #2 showed the highest maximum coolant velocity of 2.125 m/s. The housing-mounted fins redistributed the coolant flow and promoted turbulence, significantly reducing the stagnant region observed in Design #1 and improving the coolant temperature distribution. However, localized recirculation regions remained behind the fins due to flow obstruction. Design #3 exhibited a maximum coolant velocity of 2.073 m/s. Although its maximum velocity was slightly lower than that of Design #2, the heating-plate-mounted fins increased the effective heat-transfer area while simultaneously generating turbulence near the heating surface. Consequently, Design #3 exhibited the most uniform coolant temperature distribution and effectively reduced local thermal concentration around the heating plate.

Figure 2 presents the coolant streamline distributions. Design #1 exhibited asymmetric coolant circulation with a large stagnant region, whereas Design #2 improved coolant mixing by reducing flow stagnation. Design #3 provided the most stable coolant circulation, promoting coolant mixing near the heating plate and enhancing heat transfer throughout the chamber.

Overall, the CFD results indicate that the thermal-flow performance of the coolant heater is governed by the combined effects of coolant mixing, near-wall turbulence, and effective heat-transfer area. Among the three configurations, Design #3 demonstrated the best thermal-flow performance by improving coolant circulation around the heating plate while reducing local thermal concentration.

This work was supported by MOTIR through KEIT (Project No. RS-2026-25545455).

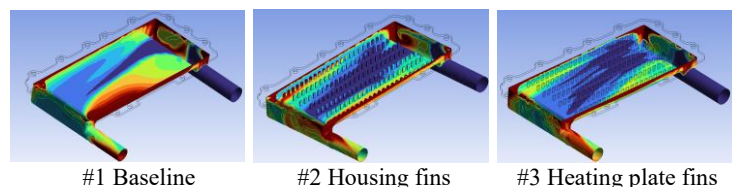


Fig. 1 Comparison of coolant temperature distributions

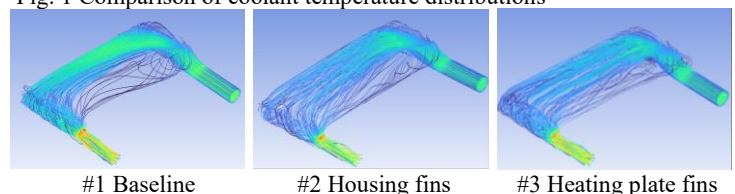


Fig. 2 Comparison of coolant streamline distributions