International Journal of Heat and Mass Transfer 73 (2014) 511–520

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Numerical assessment of liquid cooling system for power electronics in fuel cell electric vehicles

Heesung Park *

Research and Development Division, Hyundai Motor Company, 104, Mabuk-dong, Yongin-si 446912, South Korea

ARTICLE INFO

Article history:
Received 9 July 2013
Received in revised form 15 February 2014
Accepted 16 February 2014
Available online 14 March 2014

Keywords:
Power electronics
Liquid cooling system
CFD
Fuel cell electric vehicle

ABSTRACT

Electrical power from the fuel cells is converted and controlled by power electronics that are composed of control units, converters and switching devices. During the power management, the inevitable power losses induce heat generation in the power electronics. In this, effective design for the cooling system is essential in terms of safety, reliability, and durability. A liquid cooling system for the power electronics is applied to chill the electrical components below the thermal specifications. Nonetheless, the layout of cooling components is usually designed after the completion of the chassis and power electronics in the automotive applications, thus, only a little freedom is allowed to change the layout. Thus, it is significant and urgent to investigate the cooling performance before finalizing the layout design. In this paper, one dimensional and computerized fluid dynamics code is employed to simulate the performance of the cooling system at the early stage of conceptual design. Three different layouts of cooling systems are chosen to compare the ensuing systematic cooling performances. The liquid flow rates, pressure drops, and maximum temperatures are computed by the numerical simulations of the cooling system which comprises the cold plates, liquid pump, radiator, and plumbing network. It is demonstrated that for a fuel cell electric vehicle of 100 kW, the dual cooling loops with a specified array control the maximum temperatures below thermal specification by inducing the higher liquid flow rate of rate of 33.4 L/min through radiator than 20.0 L/min in a single loop. The proposed systematic numerical simulation provides significant information to determine the layout of the power electronics coupled with the cooling performance at the early stage of conceptual design.

1. Introduction

In the automotive industries, environmental friendly and efficient fuel consumption technologies have been demanded for social benefits. Biofuels and natural gases could be anticipated to reduce hydrocarbon gas emission and fossil fuel consumption but are only temporary fuels at best [1]. Although hybrid electric vehicles or plug-in hybrid electric vehicles have been emerged in the consumer market, the technical development is moving towards fuel cell electric vehicles that can satisfy the social demand of zero hydrocarbon emission without requiring external electric power supplement. Fuel cell electric vehicles are powered up to 100 kW with high voltage by electrochemical reactions of hydrogen and oxygen in a fuel cell stack. Technically, the generated electric power is converted to lower the voltage and supplied to electric motor or other power consuming parts. At a typical conversion efficiency of 90% to 95% in the power electronics [2], the dissipated heat up to 10 kW should be rejected by a cooling system. The performance and fuel economy of the electric vehicles can be successfully improved by the effective design and control of the power electronics [3]. In addition, since the critical components of the power electronics become smaller and the long-term reliability and effective conversion depend on the cooling performance [4,5], various thermal characteristics and performance targets must be met along with reducing the weight, volume, and cost of the power electronics [5–7]. Therefore, the optimum design for cooling system of the power electronics is required in commercialization of energy effective fuel cell electric vehicle.

Conventional air cooling devices [8–11] have been developed to effectively chill the low heat dissipated electronics, whereas advanced liquid cooling techniques have been studied to dissipate high heat flux from densely integrated circuit chips [12] and power electronics [13–15]. Since the liquid coolant has an advantage over air in terms of heat capacity and thermal conductivity, an extensive amount of research has been conducted to enhance the cooling performance and efficiency by using single-phase liquid, [12,13,16–19], phase changed liquid [14,20–24], liquid jet [25–27], liquid spray [5],

---

* Tel.: +82 10 2690 3461.
E-mail addresses: heesung.park@hyundai.com, heros93@gmail.com

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.02.038
0017-9310/© 2014 Elsevier Ltd. All rights reserved.

thermo-electric cooler [15], and phase change material [28]. Although the advanced cooling techniques show good cooling performances to remove high heat flux, however, they induce increased pressure drops due to high surface to volume ratio [12,29]. In this regard, it is important to evaluate systematic cooling performance. Pang et al. [30] investigated the cooling system of internal combustion engine by coupling one-dimensional and three-dimensional simulations. They showed the thermal performance of the radiator played critical role in the cooling system performance. Sharma et al. [31] proposed to determine the optimal operating conditions of liquid cooling for electronics chips. By conducting systematic analysis, they have shown that the reuse of recovered heat from the electronic chips and thermal stability were simultaneously maximized. An automotive fuel cells' thermal management systems were studied by modeling fuel cell stack, heat exchanger, liquid pump and radiator [32–34] while a cooling system of the power electronics in hybrid electric vehicle was investigated [35]. In the literature it is still hard to find the reports on the systematic cooling performance to chill the power electronics for the fuel cell electric vehicles.

In this paper, a liquid cooling system for power electronics in fuel cell electric vehicle is systematically investigated by consecutively conducting three dimensional (3-D) and one dimensional (1-D) numerical simulations. The proposed numerical approach is effective and informative at the early stage of conceptual design to evaluate the systematic cooling performances of the electrical components coupled with the cooling devices of the liquid pump, radiator, and plumbing network. 3-D commercially available computational fluid dynamics (CFD) code characterizes the pressure drop and maximum temperature of every electrical component with respect to the coolant flow rate, while systematic cooling performance due to the operating conditions of the liquid pump and radiator is simulated by 1-D CFD code. The objective of this work is to determine the cooling system layout at the early stage of conceptual design by satisfying the thermal specification of the power electronics in the fuel cell electric vehicle of 100 kW. It is shown that the cooling system performance considerably varies with respect to the cooling layout.

2. System description

In the fuel cell electric vehicles, the power electronics is composed of many electrical components to convert and control electrical power flow to the batteries, motor, and other power consuming parts. Fig. 1 shows the schematic diagram of the electrical power flow and liquid coolant flow. Considering a fuel cell electric vehicle of 100 kW power, the electrical power with high voltage from 200 to 400 V is generated from a fuel cell stack by electrochemical reaction between hydrogen and oxygen and supplied to motor, battery, and miscellaneous parts through the high voltage junction box (HVJ) which contains multiple switching devices. Motor control unit (MCU) controls the motor to propel the vehicle while the auxiliary control driver (ACD) transports digital signals to miscellaneous parts, such as fan, pump, lamps, and other electronics. High and low voltage DC/DC converters (HDC, LDC) implement to charge battery and supply electrical power to ACD.

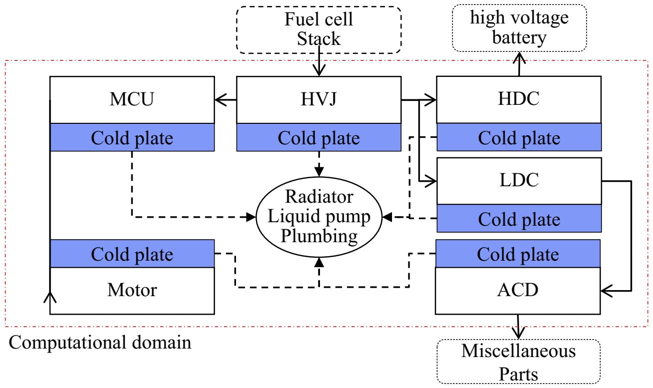

Fig. 1. Schematic diagram of power flow (solid lines) and coolant flow (dotted lines).

Table 1
Lists of the maximum heat dissipation rates and thermal design points of the electrical components.

	HDC	LDC	MCU	HVJ	ACD	Motor
Heat dissipation rate (W)	650	320	1800	450	500	6600
Heat flux (W/cm^2)	3.3	6.4	12.7	0.6	4.3	5.7
Thermal designpoint (°C)	85	85	85	85	85	120

The liquid cooling system is composed of the liquid pump, cold plates, radiator, and plumbing network as shown in Fig. 1. The liquid pump circulates the liquid through a plumbing network and cold plates to the radiator where the heat is rejected by air flow. The cold plates are individually designed with regard to the heat fluxes from the electrical components. The generated amount of heat and thermal design points in the components are summarized in Table 1. The heat fluxes of the components are estimated by assuming the power conversion efficiency of 90% at the power conversion.

3. Numerical

3-D and 1-D CFD codes are used to simulate the fluidic and thermal characteristics of the liquid cooling system. Fig. 2 illustrates the flowchart of the numerical simulation process proposed in the paper. From the 3-D simulation, the computed relationships between the pressure drop (ΔP), the maximum temperature difference (ΔT) and flow rate (Q) become the input parameters to conduct the 1-D simulation. To account for the pressure loss due to the fluid connectors, the plumbing network is categorized into straight, deflected, and branched parts to define the length (L), diameter (D), curvature (R), deflection angle (θ_1), branched angle (θ_2) and surface roughness (ε) as shown in Fig. 3. The characteristic curves of liquid pump and radiator are also used to simulate the operating conditions of the cooling system. It is denoted that the radiator is thermally modelled as a heat exchanger with the specified inlet air flow rate (Q_a) and temperature (T_a). With these, 1-D numerical simulations are conducted to predict the pressure drops, liquid flow rates, and maximum temperatures of all the components in the liquid cooling system.

3.1. 3-D simulation

3-D commercial CFD code (STAR-CCM + version 7.04) is used to compute the pressure drops and maximum temperatures with respect to the liquid flow rate for all the electrical components. The geometries for the CFD simulations are imported from 3-D CAD models to generate mesh as shown in Fig. 4. For each component, it is assumed that heat is generated from the electrical parts of integrated circuits (ICs) or printed circuit boards (PCBs) and transferred to liquid without loss. The liquid flow paths are designed at the upper side of the illustrated figures whereas the heat sources are modeled at the bottom side. The cold plates of HDC, LDC, and MCU have plural fins or pins in their internal compartments to enhance heat transfer, while the liquid flow paths of HVJ and ACD are simply structured to reduce flow resistances. A coiled