Yongchun Liang[1], Jingnan Zhang[1], Mo Li[2], Yanping Guo[1], Jiansheng Yuan[2]

1.School of Electrical and Information Science, Hebei University of Science and Technology
2. Department of Electrical Engineering, Tsinghua Universigy

Thermal Analysis of the Heat Exchanger for Power Electronic Device with Higher Power Density

***Abstract**. Liquid cooling system has been used in high power electronic device systems to cool down the temperature of power electronic device. Heat exchanger is an important part of liquid cooling system to transfer the heat generated by power electronic device into air. In this paper, a Streamline-upwind/Petrov-Galerkin (SUPG) stabilized finite element analysis method was proposed to solve the water and air governing formulas including the mass conservation equation, the momentum conservation and the energy conservation equation. Furthermore, the thermal characteristic of a heat exchanger is simulated and the result was compared with an experiment. The comparison shows that this method is effective.*

***Streszczenie.** W artykule przedstawiono zagadnienie kontroli systemów chłodzenia w urządzeniach energoelektronicznych. Zaproponowano metodę SUPG (ang. Streamline-Upwind Petrov-Galerkin) do obliczania rozpływu cieczy i powietrza, z uwzględnieniem oszczędności ilości, energii oraz pędu środka chłodzącego. Otrzymane charakterystyki wymiany ciepła zostały przebadane symulacyjnie, a wyniki porównano z badaniami eksperymentalnymi. (**Termiczna analiza wymiany ciepła w urządzeniach energoelektronicznych o dużej gęstości mocy**)*

Keywords: power electronic device, liquid cooling system, heat exchanger, Streamline-upwind/Petrov-Galerkin, finite element analysis
Słowa kluczowe: urządzenia energoelektroniczne, system chłodzenia cieczą, SUPG, analiza metodą elementów skończonych.

Introduction

The demand for high voltage and high current in hybrid electrical vehicles and electrical locomotive, and so on, presents technical challenges for power conversion beyond those normally associated with electrical and electronic systems. In these applications, power electronics reduced system volume and costs are required to stay competitive. These requirements impose the use of high switching frequencies in order to reduce passive component sizes. As a result, semiconductors are dissipating more heat with increased density. But the maximum junction temperature provided by the manufacturer has to be followed at any time of operation when using power semiconductors. The heat potential due to energy losses has to be dissipated by a cooling system, which may also serve as a constructional element [1-4].

There are many ways to remove heat from a power electronic device. However, nearly all of them are based on the same common principle: to move heat away from the device to the ambient medium (in most cases air) by convection, conduction and radiation.

With the drastically increased power dissipation of HV switches only high-performance cooling systems should be used to improve converter dimensions.

Natural air cooling is mostly used in low power range applications. However, it can be used as well in high power range applications if extremely large cooling surface are available in the device. But this will increase the size of whole converter device.

In contrast to natural air cooling, forced air cooling is the most widely used technique in the present power electronic systems. Highly efficient forced air cooled heat sinks have a hollow-fin structure that consists of basic extruded aluminium profiles with extruded ribs pressed into the profiles. The fin geometry is optimized in terms of flow resistance and thermal efficiency. Such a shpe provides an increased surface area and improved turbulence of the air, while one or more fans provide necessary air volume to absorb the heat from the active surfaces of the heat sink. This is a relatively inexpensive thermal management solution for medium- and high-voltage IGBTS packaged in brick-style standard module formats and it is generally used for cooling converts with rated power up to 1 MW [5-7].

The most efficient way of cooling > 1 MW converters is to utilize liquid cooling. Liquid-cooling takes advantage of liquid's higher heat density, capacity and thermal conductivity in comparison to air. This enables cooling concentrated heat sources in remote locations, providing more freedom in the physical design of power electronic systems, allowing more compact higher power density systems to be developed [8-11].

An actual liquid-cooling system is shown in Figure 1. A liquid cooling loop for contact typically consists of a heat sink, a pump, a heat exchanger, and pipe or hose. A heat exchanger is a device designed to efficiently transfer heat from liquid to air. Pump, along with a heat exchanger, is the second power consuming active component in a typical closed loop liquid cooling system. The pump must deliver the required liquid flow rate, while overcoming the pressure drop in the cooling system. A heat sink is in contact with high heat flux components that must have waste heat to be removed to prevent overheating.

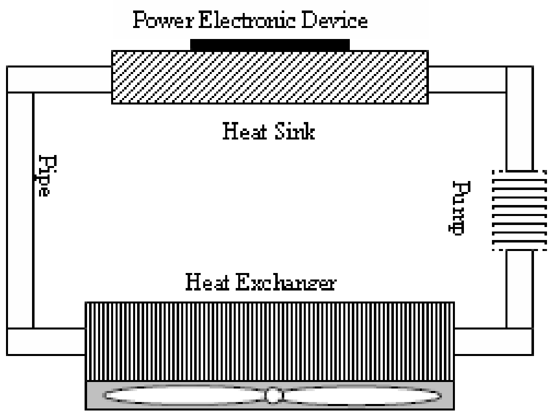

Fig.1 Typical structure of closed loop liquid cooling system

In electrical locomotive, high power density always appears in electronic converters. A water-cooling system is mostly provided to dissipate the heat from IGBT converters. Because almost all the heat is transferred by the heat exchanger into air, the thermal transfer effectiveness of the heat exchanger will influence the work of the whole cooling system. In this paper, Streamline-upwind/Petrov-Galerkin stabilized finite element analysis method was used to analyze the thermal effectiveness of the heat exchanger.

Calculation Method

The length, width and height of the heat-exchanger are 1051mm, 795mm, and 260mm respectively. It is difficult for an ordinary computer workstation to build the whole structure of heat exchanger and calculate the temperature distribution of the whole heat exchanger. There are 43 water branches and 44 air branches in a heat exchanger. In each water branch, there are 104 water channels. In each air branch, there are 318 air channels. It is still difficult for an ordinary computer workstation to calculate a model with one water branch and one air branch. Because the water branches and the air branches are all parallel connection, we can make an assumption that there is no heat exchange between different branches.

Therefore, we can study and calculate some small model first. The total effect of heat exchanger can be obtained by multiplying the effect of the small branch by the number of branches. The heat in the water is transferred by two air branch in contact with the water branch. Of course, one air branch transfers the heat from two water branch in contact with the air branch. We suppose that the heat transfer in air branch and water branch is symmetrical. So a small model with three semi-air-channels and three semi-water-channels is shown in Figure 2.

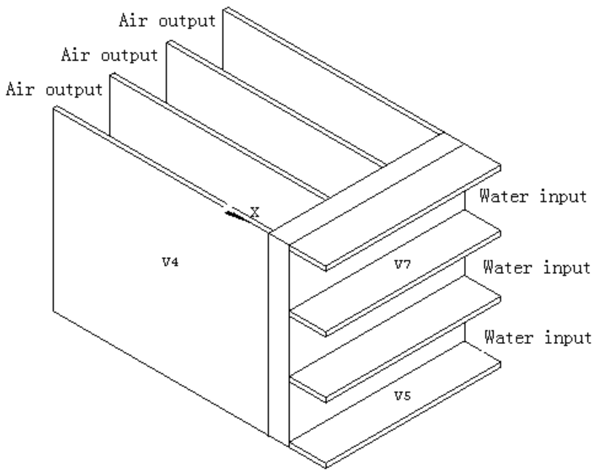

Fig.2. A small simulation model of heat exchanger

We must find an equivalent approach to evaluate the effect of one branch based on the calculation results of a few channel. If we can get the heat exchange coefficient of a simple simulation model with two air fins (one air channel) and two water fins (one water channel). The heat exchange ability of the whole branch can be calculated by follow method and the heat exchange ability of the whole heat-exchanger can be obtained.

The process of heat exchange between water and air is that the heat of water is transferred from water to fins and from fins to air by heat convection. The heat convection formula is Newton formula as shown in formulas (1) and (2).

When the heat is transferred from wall to fluid

$$q = h_1(t_w - t_f) \tag{1}$$

When heat is transferred from fluid to wall

$$q = h_2(t_f - t_w) \tag{2}$$

where q is the heat, h_1 and h_2 is heat exchange coefficient, t_w is the temperature of wall, t_f is the temperature of fluid.

The heat exchange coefficient is influenced by the temperature of fluid, the shape of the wall, the length of the wall, the velocity of fluid, and so on. When the velocity of fluid and the shape of the wall are constants and the temperature of the fluid doesn't vary too large, the heat exchange coefficient is a constant. Here, we suppose the heat exchange coefficient of one air channel and one water channel model is constant.

We suppose two heat exchange coefficient for one air channel and one water channel model:

h_{w-a} represents the heat loss of water and h_{a-w} represents the heat increase of air. The heat loss of water can be described by

$$h_{w-a}(\overline{T}_w - \overline{T}_a) = c_w m_w \Delta T_w \tag{3}$$

where $\overline{T}_w$ is the average temperature of outlet water, $\overline{T}_a$ is the average temperature of outlet air, c_w is the heat capacity coefficient of water, m_w is the weight of water that flow through the simulation model per second, ΔT_w is the different temperature of water between inlet and outlet.

The heat capacity coefficient and the weight of water are all constants. Formula (3) can be rewrite by another mode:

$$h'_{w-a}\left(\frac{T_{win} - T_{wout}}{2} - \frac{T_{ain} - T_{aout}}{2}\right) = (T_{win} - T_{wout}) \tag{4}$$

where T_{win} is the average temperature of inlet water, T_{wout} is the average temperature of outlet water, T_{ain} is the average temperature of inlet air, T_{aout} is the average temperature of outlet air， and

$$h'_{w-a} = \frac{h_{w-a}}{c_w m_w}$$

The heat increase of air can be described by

$$h_{a-w}(\overline{T}_w - \overline{T}_a) = c_a m_a \Delta T_a \tag{5}$$

Where c_a is the heat capacity coefficient of air, m_a is the weight of air that flow through the simulation model per second, ΔT_a is the different temperature of air between inlet and outlet.

The heat capacity coefficient and the weight of water are all constants. Formula (5) can be rewrite by another mode:

$$h'_{a-w}\left(\frac{T_{win} - T_{wout}}{2} - \frac{T_{ain} - T_{aout}}{2}\right) = (T_{aout} - T_{ain}) \tag{6}$$

where

$$h'_{a-w} = \frac{h_{a-w}}{c_a m_a}.$$

If we know h'_{a-w} and h'_{w-a} and the inlet temperature of water and inlet temperature of air are given, the outlet temperature of every air channel T_{aout} and the outlet temperature of every water channel T_{wout} can be calculated by Formula

When T_{ain} and T_{win} are know, the outlet air average temperature and the outlet water average temperature of a branch can be calculated step by step by solving formula(4) and formula (6).

The procedure is constituted by two segments. In the first segments, we simulate a simple model to get the heat-exchanging coefficient h'_{a-w} and h'_{w-a}. In the second segments, we calculate the outlet air average temperature and the outlet water average temperature of a whole branch.

Segment 1:

Step 1: calculate thermal distribution field of a simple simulation model.

Step 2: calculate average outlet temperature of water channels, T_{wout} and average outlet temperature of air channels, T_{aout}.

Step 3: set $\overline{T}_w = \dfrac{T_{win} - T_{wout}}{2}$ and $\overline{T}_a = \dfrac{T_{aout} - T_{ain}}{2}$

Step 4: h'_{a-w} and h'_{w-a} can be calculated by formula (4) and formula (6).

Segment 2

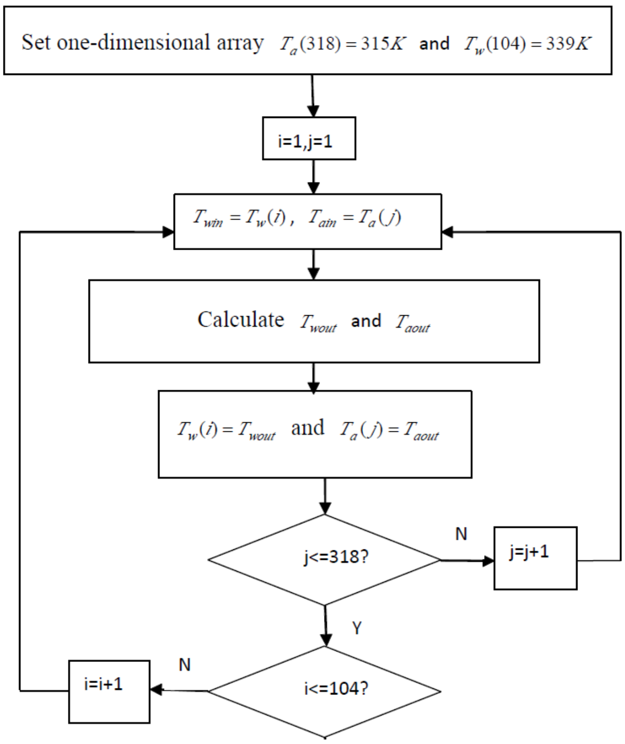

Fig.3 Procedure of outlet temperature calculation of a whole branch

The temperature drop of water and the temperature rise of air are calculated and the heat exchange ability of the whole heat-exchanger is obtained.

Basic Control Equations

The temperature computation of the heat exchanger requires solving the heat transfer equation which defines a functional relationship between the heating and the temperature with the heat exchanger.

The heat is transferred from water to fins and from fins to air. Finally, the heat is taken away by forced cooling air. In this process, there are two kind of heat transfer mode: heat conduction in the fins and heat convection between water and fins and between fins and air.

In the fins, the heat conduction equation for three-dimensional model can be described as

(7) $$K\frac{\partial^2 T}{\partial x^2} + K\frac{\partial^2 T}{\partial y^2} + K\frac{\partial^2 T}{\partial z^2} = 0$$

where K is the thermal conductivity of the materials, T is the unknown temperature.

In water or air, to solve the heat transfer problem need to solve the coupling of the mass conservation equation, the momentum conservation equations and the energy conservation equation.

The mass conservation equation is defined by (8).

(8) $$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

where u is the velocity of the water or air in the x direction, v is the velocity of water or air in the y direction, w is the velocity of water or air in the z direction.

The momentum conservation equations are defined by (9), (10) and (11).

(9) $$\rho(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}) = -\frac{\partial p}{\partial x} + \eta(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial u}{\partial z^2})$$

(10) $$\rho(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}) = -\frac{\partial p}{\partial y} + \eta(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial v}{\partial z^2})$$

(11) $$\rho(u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}) = -\frac{\partial p}{\partial z} + \eta(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial w}{\partial z^2})$$

where ρ is the density of the water or air, η is the kinematic viscosity of the water or air, p is the pressure of water or air.

The energy conservation equation is defined by (12)

(12) $$u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z} - K(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}) = 0$$

Subdividing the field domain is usually the most tedious work during the implement of FEM.

In this study, quadrilateral mesh is used to subdivide the simulation model as shown in Figure 4 and Figure 5.

With the weak Galerkin's procedure, the finite element model of equation (7) becomes,

(13) $$[K]^e[T]^e = [P]^e$$

The SUPG finite element modelling of equation (9), (10) and (11) becomes

(14) $$Mu + Cu + Ku \cdot Qp = F$$

The finite element modelling of equation (8) and (12) is similar to equation (13) [12].

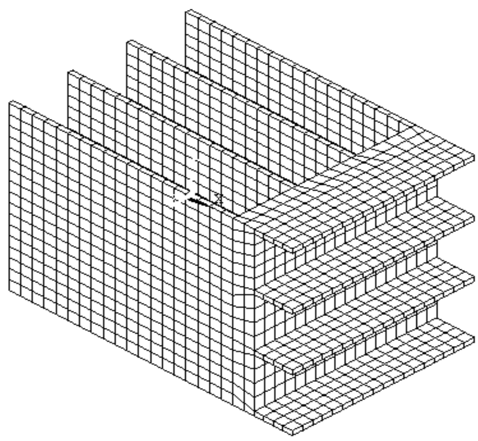

Fig.4 Subdivide field domain of fins

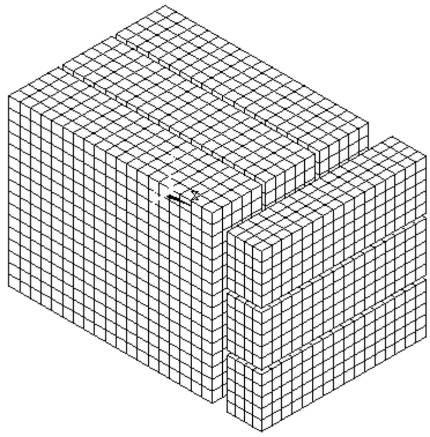

Fig.5 Subdivided field domain of water and air

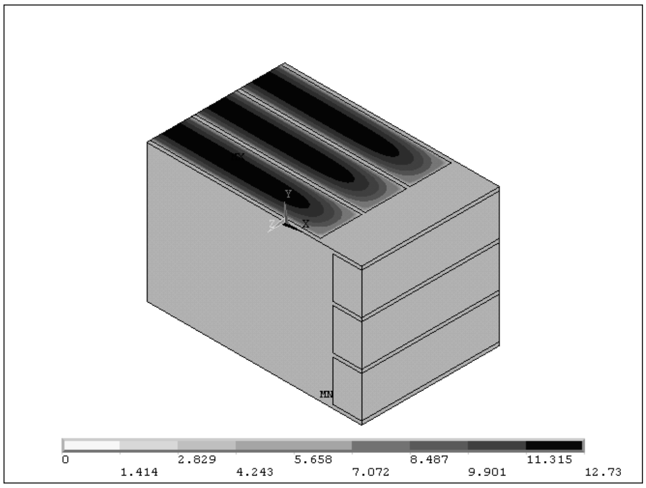

Fig.6 The velocity distribution of simulation model

Result and Analysis

In this simulation, we must know the water velocity, the air velocity, the inlet air temperature and the inlet water temperature. The certain condition of the same air-machine is translated to the same air-flow, i.e., the air-flow equals to 390m^3/min. In this condition, the air velocity of heat-exchanger equals to 10.7m/s with the air channel of 2.3 mm. furthermore, the water-flow equals to 200l/min and the water velocity of heat-exchanger equals to 0.1m/s. The inlet air temperature T_{ain} equals to 315K and the inlet water temperature T_{win} equals to 339K.

For the simulation model shown in Figure 2, we use FLUENT to calculate the temperature distribution and velocity distribution of the model. The results are shown in Figure 5 and Figure 6.

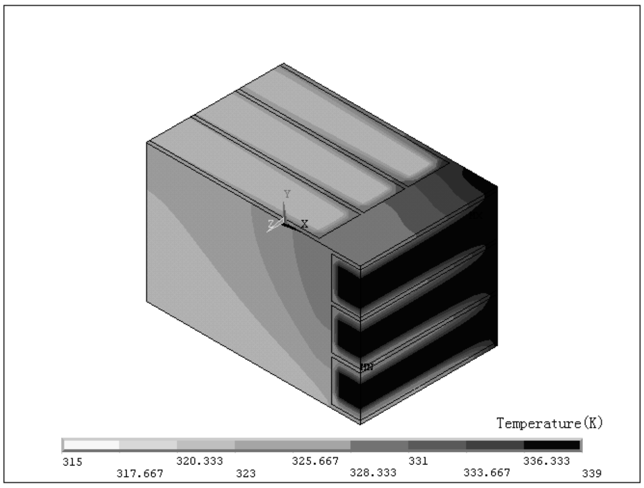

Fig.5 The temperature distribution of simulation model

From Figure 5, we know that the water temperature decreased along the water flow direction and the air temperature increased along the air flow direction. The temperature of air in contact with the fin is higher than other places. The temperature of water in contact with the fin is less than other places. The heat in water is transferred into air.

From Figure 6, because the air velocity is very larger than the water velocity, we can only distinguish the air velocity. Because there is friction between air and fins, the velocity of air in contact with fins almost equals to 0. The maximum velocity of air equals to 12.73m/s.

The key problem to calculate the thermal transfer ability of the whole heat exchanger with high accuracy is to get h'_{a-w} and h'_{w-a} with high precision. Because we use a small model in the simulation, the difference between middle fins and end fins will lead to some error.

In order to reduce these errors, we calculate a series of models, from a small model shown in Figure 2 to a large model with 58 air channels and 35 water channels. The h'_{a-w} and h'_{w-a} results are shown in Figure 7.

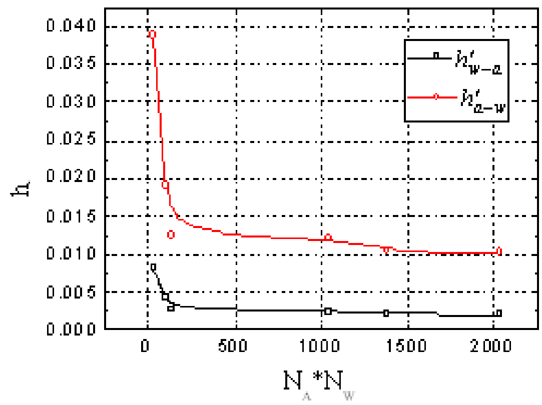

Fig.7 The relation between the heat transfer coefficient of single channel and the size of simulation model

From Figure 7, we know that the heat transfer coefficient changes little with the size of simulation model when the size is bigger than 1500.

In this paper, we adopt a simulation model with 58 air channels and 35 water channels. We get h'_{w-a} =0.0027 and h'_{a-w} =0.0120. Using these results, we can calculate the air outlet temperature and the water outlet temperature of the whole branch with the method provided in above. The maximum air outlet temperature is 332.1 K. The minimum air outlet temperature is 326.4. The average air outlet temperature is 329.2K. The maximum water outlet temperature is 333.4 K. The minimum water outlet temperature is 326.1K. The average water outlet temperature is 330.1 K.

In order to validate the accuracy of the method, we carry out a test. The average water outlet temperature of the test is 331 K and the average air outlet temperature of the test is 329 K. The water temperature error is 0.9 K and the air temperature error is 0.2 K. This is acceptable in engineering.

Conclusion

It is almost impossible to simulate the whole structure of heat exchanger with ten thousands of channels by FLUENT. We must find an equivalent approach for the evaluation.

A recursive method has been presented to simplify the simulation model for a normal computer to calculate the temperature and velocity distribution of a whole heat exchanger.

The comparison between simulation method result and test result has proved that this method is acceptable in engineering and can be used to analyze the thermal transfer ability of any heat exchanger.

Acknowledgments

The authors wish to thank the support of innovation team funds of Hebei University of Science and Technology, Electrical Machine Laboratory of the Hebei University of Science and Technology .

REFERENCES

[1] Hsueh-Rong Chang, Jiankang Bu, George Kong and Ricky Labayen, 300A 650V 70um Thin IGBTs with Double-Side Cooling, Proceedings of the 23rd International Symposium on Power Semiconductor Device & IC's, 2011 San Diego, CA, 2011

[2] John Vetrovec, Active Heat Sink for Automotive Electronics, International Journal of Passengar Cars-Electronic Electrical System 1(2009) 336-343.

[3] Chan-Su Yun, Paolo Malberti, Mauro Ciappa, Wolfgang fichtner, Thermal Component Model for Electrothermal Analysis of IGBT Module Systems, IEEE Transactions on Advanced Packaging 3(2001) 401-406

[4] M.Bartram, M.Schrey, H.Kuhn, Rik W. De Donchker, A New Approach to Optimize IGBT-Heatspreader-combinations Allow "real-time" Simulation of IGBT-Chip-Temperature, 35th Annual IEEE Power Electronics Specialists Conference, Aachen, Germay, 2004

[5] Javier Valenzuela, Thomas Jasinski, Zahed Sheikh, Liquqid Cooling for High-Power Electronics, Power Electronics Technology (2005) 50-56

[6] Andrei Blinov, Dmitri Vinnikov, Cooling Methods for High-Power Electronic Systems, Scientific Journal of Riga Technical University 29(2011) 79-86

[7] J.Biela, J.W.Kolar, Cooling concepts for High Power Density Magnetic Devices, Power Conversion Conference, Nagoya, 2007.

[8] C.Simpson, Liquid cooling for wide bandgap semiconductor technology, presented at Adv. Liquid Cooling Workshop, 2003.

[9] Jeremy C.Howes, David B.Levett, Shawn T. Wilson, et al, Cooling of an IGBT Drive System with Vaporizable Dielectric Fluid (VDF), 24th IEEE SEMI-THERM Symposium, San Jose, CA, 2008

[10] Schulz-Harder, Exel Karl, Meyer Andreas, Direct Liquid Cooling of Power Electronics Devices, 4th International Conference on Integrated Power Systems, Naples, Italy, 2006

[11] Avijit Bhunia, Sriram Chandrasekaran, Chung-Lung Chen, Performance Improvement of a Power Conversion Module by Liquid Micro-Jet Impingement Cooling, IEEE Transactions on Components and Packaging Technology 2(2007) 309-316

[12] T.Wenquan, Numerical Transfer, Xi'an Jiaotong University Press, Xi'an, China, 2001.

***Author**:*

Dr. Yongchun Liang, He is with the school of Electrical and Information Science, Hebei University of Science and Technology, No.70 Yuhuadonglu, Shijiazhuang, Hebei China, 050018. E-mail: yongchunliang@hotmail.com.