Basic Study on Flow and Heat Transfer Control around Heating Components in Rectangular Duct by Pulsating Flow

Takashi Fukue[1,*], Hidemi Shirakawa[2], Wakana Hiratsuka[3]

[1]Kanazawa Institute of Technology, Kanazawa, Japan
[2]National Institute of Technology, Toyama College, Toyama, Japan
[3]Iwate University, Morioka, Japan
*corresponding author: fukue@neptune.kanazawa-it.ac.jp

Abstract
This paper aims to develop a novel heat transfer enhancement method for high-density packaging electronic equipment by using the pulsating flow. Our research tries to apply the pulsating flow to the novel heat transfer control method of electronic equipment. This report especially investigated flow and heat transfer characteristic of pulsating flow around an array of several heating components mounted in the rectangular enclosure through 2-Dimentinal CFD (Computational Fluid Dynamics) analysis. In order to investigate the possibility of heat transfer control by the pulsating flow in electronic equipment, transient flow and heat transfer analysis was performed by using OpenFOAM Ver. 4.1, which is an open-source CFD toolbox. Working fluid was air which simulated fan-cooled electronic equipment. The dimensions of the calculation model of the rectangular enclosure simulated an enclosure of 1U-size electronic equipment mounted in a 19-inch rack. The multiple rectangular components, which simulates an electronic component, was mounted in the enclosure while changing the dimensions and the clearance between the components. The constant heat dissipation was generated from the surfaces of the components. As the analytical conditions, the time-averaged flow rate of the pulsating flow was set to the flow rate of the steady flow at which the time-averaged Reynolds number was 1,000. This simulated the airflow condition in high-density packaging electronic equipment. The pulsating frequency of the pulsating flow was 1 Hz. In this report, visualization of flow and temperature field around the array of the components was especially conducted. By the comparison of the flow and heat transfer phenomena around the components, the effectiveness of the pulsating flow was evaluated. It was found that the application of pulsating flow changes the flow pattern between the component and this affects to the temperature distribution around the components. Generally, the flow separation causes between each component. However, when the pulsating flow is caused, the counter flow from the surrounding to the rear of the component. This removed the heated air between the components and decreased the temperature. On the other hand, when the clearance between the components become narrower, the speed of the counter flow was restricted. Therefore, we concluded that there is the possibility of the heat transfer enhancement around the electrical components mounted in the enclosure by the pulsating flow. However, the level of the heat transfer enhancement by the pulsating flow is dependent on the mounting position of the electrical components.

Keywords: Pulsating Flow, Cooling Device, Heat Transfer Enhancement, Thermal Design of Electronic Equipment

1 Introduction

Recent years have witnessed an increase in the demand for the reduction of power consumption in electronic equipment in order to save energy. In data centers, 38 percent of input power is used for cooling systems [1]. On the other hand, in laptop computers, it is critical to prolong battery life and reduce noise in order to satisfy users' demand regarding various usage environments. Controlling the operation of a cooling fan mounted in laptop computers is vital to reducing power consumption and noise. The improvement of cooling technology for electronic equipment is the key to reducing power consumption in electronic equipment.

Forced convection cooling by a fan is the most common method for dissipating heat from electronic equipment [2] – [4]. However, the fan generally requires additional electricity and its operation decreases battery life. In addition, the high rotation speed of the fan generates not only a high flow rate but also much noise. In order to reduce both noise and power consumption of the fan, it is necessary to control its rotation speed to optimize operation.

Flow and heat transfer phenomena induced by pulsating flow have shown that the application of pulsating flow can enhance heat transfer according to the pulsating conditions [5]. Hence, if we could control fan rotation speed and generate a pulsating flow, the cooling performance of the fan would be improved while reducing net power consumption. The reduced net power consumption of the fan would lead to reductions of environmental load and fan noise. Therefore, the combination of a fan and a pulsating flow could be the next-generation technique for heat transfer enhancement in electronic equipment. However, in order to achieve the heat transfer enhancement in electronic equipment by using the pulsating flow, the effectiveness of the pulsating flow should be evaluated. In densely packed electronic equipment, a number of components, including power devices, capacitors, resistance blocks and heat sinks are mounted. In order to control the temperatures of the components, the flow and heat transfer performance of the pulsating flow around the components should be investigated.

Against this background, we are now investigating the flow and heat transfer characteristics of a pulsating airflow both experimentally and analytically, in order to apply the pulsating airflow to a novel cooling method [6] [7]. In our previous reports, we have already reported that the pulsating flow can enhance heat transfer around heating components while decreasing time-averaged supply flow rate of the coolants. In this report, we additionally investigated the flow and heat transfer phenomena of the pulsating flow around several heating components arranged in series mounted in the rectangular enclosure through 2-Dimentional CFD analysis. Through the investigation, we evaluated the possibility of the heat transfer enhancement around the obstructions arranged inline by the pulsating flow.

2 Analytical Model and Method

Figure 1 shows the schematic of the analytical model investigated in this study. 2-Dimentional flow and heat transfer analysis was performed. Working fluid was air. A width of the flow passage is 60 mm. 3 square pillars were mounted in the model. The length of the side of the pillars was 30 mm respectively. The pillars were arranged inline. The clearance between the pillars *c* was changed between 10 mm and 30 mm. Then the relationship between the heat transfer enhancement by the pulsating flow and the clearance was investigated. The first pillar (pillar 1) was mounted 60 mm after the passage inlet boundary. Hence the entrance region, which has the length of 60 mm, was prepared in front of the pillar 1. In addition, the downstream region, which has the length of 300 mm, was prepared behind the last pillar (pillar 3). The constant heat flux was generated from each side of the pillars. This simulates the heat dissipation from electrical components.

As the CFD code, OpenFOAM® Ver. 4.1, which is the open-source CFD toolbox, was used. In order to evaluate the transient change of the flow pattern and heat transfer performance in the flow passage, transient laminar flow and heat transfer analysis was conducted. The mesh number of the analytical model was about 200,000. About flow boundary conditions of the analytical model, in order to generate the pulsating flow, velocity boundary at the inlet was changed as shown in Fig. 2. An outlet was the free flow outlet and other walls were the no-slip wall condition. About heat transfer boundary conditions, the flow inlet was the constant temperature boundary, the flow outlet was the free outlet boundary, the heating surface, which is the side of the pillars, was the constant heat flux boundary and the other walls were the insulation boundary.

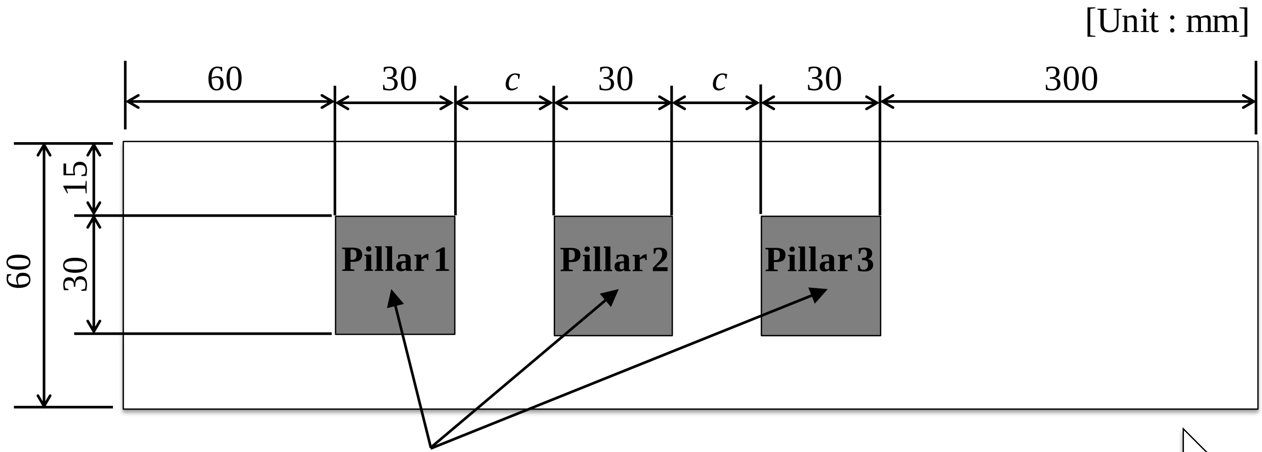

Fig. 1 Analytical model

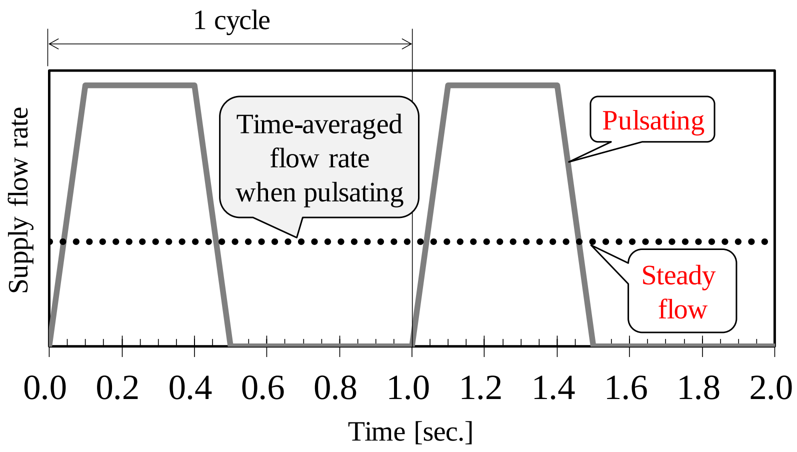

Fig. 2 Pulsating wave pattern.

Figure 2 shows a time variation of flow rate of pulsating flow in this report. In this research, as the pulsating wave pattern, the trapezoidal wave was used [7]. The time-averaged velocity of the pulsating flow and the steady flow was set to 0.32 m/s. This corresponds to the time-averaged velocity when the following time-averaged Reynolds number was 1,000.

$$Re_{\mathrm{m}} = \int_{t} \frac{u_t d}{\nu} \qquad [-] \qquad (1)$$

Where u_{t} [m/s] is the bulk mean velocity, and ν[m^2/s] is kinematic viscosity of the water. d [m/s] is the hydraulic diameter of the rectangular duct which has 60 mm in width and 40 mm in height. These dimensions simulated fan-cooled 1U-size electronic equipment mounted in a 19-inch rack. The pulsating frequency was 1 Hz. To evaluate the effectiveness of the pulsating flow, the steady water flow analysis, that the supply flow rate was the same as the time-averaged supply flow rate of the pulsating flow analysis, was also performed. Therefore, the maximum supply flow rate of the pulsating flow was higher than the supply flow rate of the steady flow analysis because the time-averaged supply flow rate became the same.
In addition, in order to evaluate the time-averaged cooling performance around the pillar, the following time-averaged Nusselt number was used.

$$Nu = \frac{hb}{\lambda} \; [-] \qquad (2)$$

where h [W/(m^2·K)] is time-averaged heat transfer coefficient by the temperature difference between mean temperature rise on each surface of the pillars and initial air temperature, b [m] is the width of the pillars and λ [W/(m·K)] is thermal conductivity of the air.

3 Analytical Results and Discussions

Figure 3 shows the relationship between time-averaged Nusselt number of each pillar and the clearance between the pillars. In almost all analytical conditions, Nusselt number of the pulsating flow becomes higher than the steady flow. Especially, the heat transfer performance of Pillar 2 and 3 improved significantly. About Pillar 1, the change of the heat transfer by the flow pulsation is relatively small. However, Pillar 1 is first in line. The cooling air impinges firstly and the net heat transfer performance is originally high. Therefore, the effect of the pulsating flow on the heat transfer enhancement becomes relatively small. In addition, when the clearance between the pillars becomes wider, the heat transfer enhancement by the pulsating flow becomes higher.

Here, the flow pattern and the temperature distribution around the pillars will be discussed. Figure 4 shows the difference of the flow pattern around the pillars between the pulsating flow and the steady flow in the case of c = 30 mm. In addition, Fig.5 shows the temperature distribution around the pillars in the case of c =

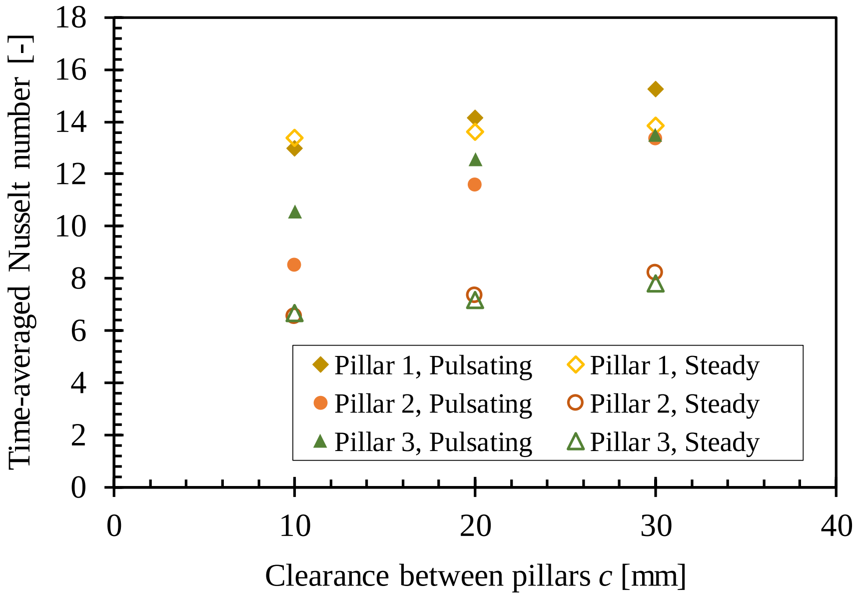

Fig. 3 Relationship between Nusselt number and clearance between pillars.

30 mm. In the case of the steady flow, the flow separation caused between the pillars. Due to the separation, the air temperature becomes higher regardless of the elapsed time as shown in Fig. 5. Especially, temperature on the surface around the flow separation area becomes higher. The cause of the flow separation at the clearances and the cause of the stagnation in the flow in the clearances decreases the net heat transfer performance around the pillars. On the other hands, in the case of the pulsating flow, the flow separation in the clearance did not cause. Especially, in the deceleration period, the flow from the main passage to the clearance caused regardless of the position of the clearance. This generates the pair vortices between the clearances and the air between the clearance was disturbed. Due to the generation of the flow in the clearance, the low-temperature air in the main flow passage was supplied to the clearance and the temperature between the clearances decreased as shown in Fig. 5. This enhances heat transfer between the pillars. And the increase of whole heat transfer performance around the pillars by the pulsating flow is caused.

Here, in the case of the steady flow, generally the heat transfer performance behind the pillars decreases because of the flow separation. However, because of the flow pulsation generation, the air around the pillars flows to the rear of the pillars as the static pressure behind the pillars decreases due to the flow separation. This mechanism increases the heat transfer behind the pillars. Therefore, we can say that the generation of the pulsating flow may enhance heat transfer in the clearance between the pillars where the flow separation generally occurs and heat transfer performance decreases. This mechanism enhances whole heat transfer performance around the pillars regardless of the position of the pillar. However, the level of the heat transfer enhancement by the pulsating flow is dependent on the mounting position of the pillar.

4 Summaries

In this paper, the flow and heat transfer characteristics of the pulsating flow around several heating components arranged in series mounted in the rectangular enclosure was investigated. Especially, the difference of the flow pattern and the heat transfer characteristics around the pillar array between the steady flow and the pulsating flow was investigated while changing the clearance between the pillars. In the range of the present research, the following summaries were obtained.

When pulsating, in almost all conditions, the heat transfer performance around the pillars is enhanced by the flow pulsation regardless of the position of the pillar. Especially, when the clearance between the pillars becomes wider, the level of the heat transfer enhancement becomes higher.

This is caused by the generation of the flow from the main flow passage to the clearance between the pillars

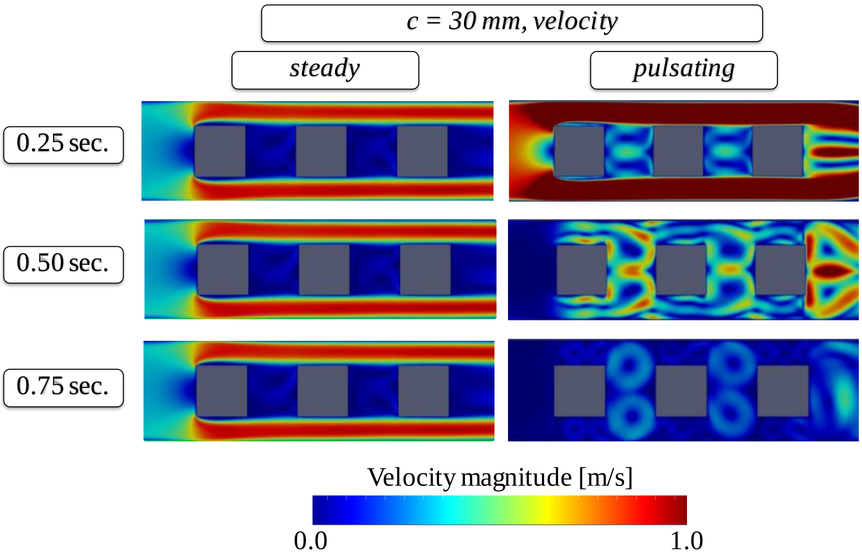

Fig. 4 Flow pattern around the pillars in the case of c = 30 mm.

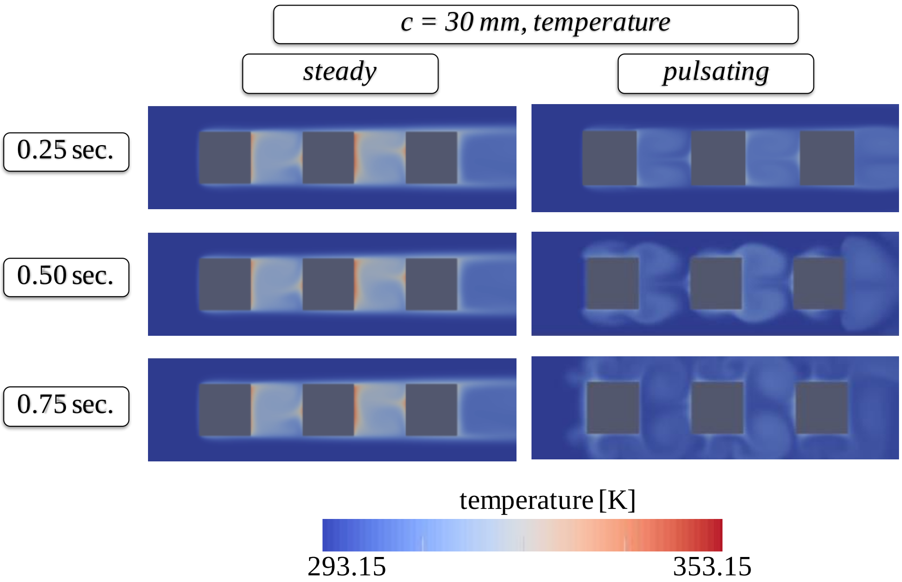

Fig. 5 Temperature distribution around the pillars in the case of c = 30 mm.

by the flow pulsation. The generation of the additional flow supplies new air to the clearance and ejects the heated air. This mechanism enhances heat transfer on the pillar surfaces. The level of the heat transfer enhancement by the pulsating flow is dependent on the mounting position of the pillar and the clearance between the pillars.

Acknowledgement

This work was supported by Grant-in-Aid for Encouragement of Young Scientists (B) from JSPS KAKENHI (Grant Number 16K18022).

References

[1] Emerson Electric Co., (2012) Energy Logic: Reducing Data Center Energy Consumption by Creating Savings that Cascade Across Systems.

[2] Alkharabsheh, S. A., Sammakia, B., Shrivastava, S., Ellsworth, M., David, M. and Schmidt, R. (2013) A Numerical Steady State and Dynamic Study in a Data Center Using Calibrated Fan Curves for Cracs and Servers, In Proceeding of the ASME InterPACK2013 Conference, Paper No. IPACK2013-73217.

[3] Egan, V., Stafford, J., Walsh, P. and Walsh, E. (2009) An Experimental Study on the Design of Miniature Heat Sinks for Forced Convection Air Cooling, ASME Journal of Heat Transfer, Vol. 131, Paper No., 071402.

[4] Fukue, T., Hatakeyama, T., Ishizuka, M., Hirose, K. and Koizumi, K. (2013) Relationships between Supply Flow Rate of Small Cooling Fans and Pressure Drop Characteristics in Electronic Enclosure, In Proceeding of the ASME InterPACK2013 Conference, Paper No. IPACK2013-73089.

[5] Saitoh, H. and Yoshioka, Y. (2010) Effect of Pulsating Amplitude on Flow Structure and Associated Heat Transfer around the Flat Plate Installed in Pulsating Duct Flow, In Proceeding of the 21st International Symposium on Transport Phenomena.

[6] Fukue, T., Yatsu, N., Obata, K. and Hirose, K., (2014) Flow and Heat Transfer of Pulsating Airflow around Obstructions mounted in Rectangular Enclosure, In Proceeding of the 25th International Symposium on Transport Phenomena, Paper No., 98.

[7] Fukue, T., Hiratsuka, W., Shirakawa, H., Hirose, K. and Suzuki, J., (2017) Numerical Investigation of Effects of Pulsating Wave Pattern on Heat Transfer Enhancement around Square Ribs by Pulsating Flow, In Proceeding of the 28th International Symposium on Transport Phenomena, Paper No., 20.