A Test Bench for Validation of CFD Analysis for Air Flow in Thin Electronic Casing Model

Tomoyuki Hatakeyama, Masaru Ishizuka, Risako Kibushi, and Shinji Nakagawa

Department of Mechanical Systems Engineering, Toyama Prefectural University, Kurokawa 5180, Imizu, Toyama, JAPAN

Abstract. This paper describes accuracy between experimental and calculated results of air flow pattern in thin electronic equipment. To measure flow patterns in the model of a thin electronic equipment, PIV measurement technique was employed. Dummy components (obstacles in the flow path in the thin electronic equipment) were set in the model and those configurations were altered for evaluating several flow patterns. For a CFD calculation, “OpenFOAM” was employed and the results of flow patterns from the CFD calculation were compared with the experimental results. A heat source was set in the model and temperature rise of the heat source was also measured. Then the relationship between flow patterns and the cooling performance were examined. As a result, the numerical results showed a good agreement with the experimental results when seeing in the over all velocity filed.

1. Introduction

In recent years, electronic equipments, such as computers, cellular phones, are becoming more and more small. However, their function is still improving. Therefore, heat generation density of electronic equipments is increasing, and the effective cooling method is required for thermal designers. Shortening the development period and reducing costs, on the other hand, is required. To fulfill these requirements, Computational Fluid Dynamics (CFD) software is becoming useful thermal design tool for electronic equipments. However, unfortunately, the capability of the present CFD software is inadequate to resolve complex heat and fluid flow in electronic equipment at a reasonable cost. Recently improvement of accuracy of CFD analysis has been attempted [1, 2, 3, 4]. The research committee of the Japan Society of Mechanical Engineers (JSME) have been working on this issue, and discussions were reported by Nakayama et al., [5, 6, 7, 8]. Further, if flow field can be calculated accurately, temperature rise in electronic equipments can possibly be estimated by using thermal network analysis [9].

To improve CFD analysis, benchmark data is useful for discussing the accuracy of CFD analysis. The objective of the present study is the comparison the results of CFD analysis with the experimental results in a simple model. This data becomes first step of benchmark data to discuss the accuracy of CFD analysis. We focused on a notebook computer and made the thin casing model. For the CFD analysis, the open source CFD toolkit “OpenFOAM” [10] was employed. For the experiment, particle image velocimetry (PIV) was conducted to monitor flow patterns in the model. Dummy components were placed in the model and those configurations were altered. A heat source was set in the model and the temperature rise was also measured, and the cooling performance depending on the flow pattern was examined.
2. Experiments

2.1. Experimental Apparatus

Fig. 1 shows the experimental apparatus. The thin electronic equipment model imitates a notebook computer. This is shown as “Casing” in Fig. 1. As shown in Fig. 1, to measure accurate flow rate through the model, a wind tunnel was employed. Flow rate is measured by using an orifice with a differential manometer. Air flow is generated from a fan attached to the exit of the tunnel. Flow rate from the fan can be controlled by changing supply voltage of the power supply unit 1 to the fan. The casing includes a heat source (details will be shown later) and the power supply unit 2 applies power to the heater. Temperature rise of the heat source is measured using a thermometer through a thermocouple.

Figure 1. Experimental apparatus

Figure 2 shows details of the model (“Casing in Fig. 1). The model has a simple geometry yet retains essential features that are found in notebook computers. The dimension of the model is $200 \times 235 \times 10 \text{ mm}^3$. The model was made of acrylic resin in order to visualize flow in the model. A printed circuit board (PCB) is placed in the model and a 672-pin plastic ball grid array (PBGA) package is mounted on the PCB as a heat source. The dimensions of PCB and PBGA are $110 \times 110 \times 1.2 \text{ mm}^3$ and $45 \times 45 \times 2.5 \text{ mm}^3$, respectively. The PBGA package contains a thermal chip having embedded heaters and temperature sensing diodes. Thus, the temperature of the chip can be measured by the sensing diodes through their temperature dependence of voltage-current relationship. Dummy components placed in the model are made of polystyrene foam. As shown in Fig. 3, the configuration of the dummy components is altered to change the air inlet position. In Fig. 3, gray blocks in the figure shows the dummy components. We evaluated three configurations. The horizontal distance $X_{in}$ between the lower left corner and the centerline of the inlet was varied from 100 to 150 mm as shown in Fig. 3.

The width of the inlet opening is 20 mm. Dummy components in the right side were made of acrylic resin to enable the laser light sheet to penetrate this side.

2.2. Experimental Procedure

In this study, to monitor the flow pattern in the model, a double pulse PIV system is employed. For the PIV measurement, liquid droplet of propylene glycol and YAG laser were employed as a tracer and a illuminating laser light sheet. Diameters of droplets are less than 1 $\mu$m. Density of propylene glycol is about $1040 \text{ kg/m}^3$. The rate of sedimentation $u_{ps}$ can be estimated using the following equation [11],

$$
    u_{ps} = \frac{1}{18} \left( \frac{\rho_p}{\rho_f} - 1 \right) \frac{g \rho_p d^2}{\nu} \left( 1 + \frac{2d}{d} \right)
$$

(1)
where \( \rho_p \) and \( \rho_f \) are density of particle and fluid respectively, \( g \) is acceleration of gravity, \( d \) is the diameter of particle, \( \nu \) is kinematic viscosity, and \( al = 9 \times 10^{-8} \) for air at standard atmosphere. For our experimental condition, \( u_{ps} \) becomes \( 3.5 \times 10^{-2} \) mm/s and it can be said that sedimentation can be ignored. In order to disperse tracer particles in air uniformly, a mixing part is prepared and well mixed air with tracer particles flow through the model.

Then the flow including tracer particles is illuminated with a laser light sheet of about 1 mm thick. The power of the YAG laser is 15 mJ/pulse. The laser light sheet is parallel to the bottom of the model and located on 7 mm above the bottom wall. The illuminated air is recorded using CCD camera, and the resolution of a CCD camera is 1008 × 1018 pixels. The spatial resolution of acquired images is about 0.16 mm/pix in our condition. The laser and the CCD camera are synchronized and digital images are recorded at a personal computer. Two images with arbitrary time intervals \( \Delta t \) are analyzed with an interrogation window of 32 × 32 pixels.

In this study, temperature measurement is also conducted in order to evaluate the relationship between the flow patterns and the temperature rise in the model. Temperature of the PBGA package and ambient air are measured. Temperature rise \( \Delta T \) can be obtained by subtracting
ambient air temperature from the PBGA temperature. Dissipated power of the PBGA package is set at 2.6 W. Electric power supplied to the fan is set at 0.4 W and the flow rate becomes about $5.64 \times 10^{-4}$ m$^3$/s. Reynolds number based on the inlet velocity and the hydraulic diameter of the inlet is about 2400 in our condition.

All data was obtained in a steady-state condition.

3. NUMERICAL SIMULATION

For CFD analysis, we employed the open source CFD toolkit “OpenFOAM”. This toolkit was developed by Weller et al. [11] The components of OpenFOAM are written in C++ programming language and it is easy to construct and manage the code due to the object oriented techniques used in OpenFOAM. Basic CFD tools are included in OpenFOAM. We applied the SIMPLE algorithm based on the finite volume method to the calculation of steady laminar flow, and air flow in the model was numerically simulated. As mentioned above, Reynolds number at the inlet is about 2400 and this is the boundary between laminar flow and turbulent flow. However, the assumption of laminar flow is valid in this case as shown later.

Figure 4 shows an example of the simulation model for $x_{in} = 100$ mm. In this model, the number of mesh is about 350,000 and unstructured mesh nodes were adopted.

![Numerical simulation model (x_{in} = 100 mm)](image)

Figure 4. Numerical simulation model ($x_{in} = 100$ mm)

As the boundary condition of velocity, fixed velocity $V_{in}$ which corresponded to the experimental condition was given at the inlet. Experimentally measured flow rate was divided by the cross-sectional area ($20 \times 10$ mm$^2$) of the air inlet in order to obtain $V_{in}$. For the surfaces, no slip wall condition was applied.

As the boundary condition of pressure, the pressure outlet condition in which the pressure was kept constant and the velocity gradient was zero was applied at the three outlets. For other parts, zero gradient of pressure was applied.

4. Results and Discussions

First, we discuss the results of flow patterns.
Figure 5 shows vector maps obtained from the PIV measurements. Figure 5 (a) is the case of $X_{in} = 100$ mm, (b) is the case of $X_{in} = 130$ mm, and (c) is the case of $X_{in} = 150$ mm. Magnitude of velocity vectors is shown in color and vector length. The value of each color is shown on the right of the map. Details of the value of velocity can be also read from Fig. 6. Squares in the center of each map show the position of the PBGA package. Cooling air flows from the bottom side of the figures (the inlet part of the model) to the top side of the figures (the outlet of the model).

Figure 5. Vector map at $Z = 7$ mm (PIV measurement)

As can be seen in Fig. 5 (a), in the case that the inlet, the outlet and the PBGA package are placed in a straight line (the case of $X_{in} = 100$ mm), cooling air directly passes through the PBGA package. The model has three outlets, and cooling air from the inlet to the center of the outlet is the mainstream in this case. In the case of $X_{in} = 130$ mm, cooling air bends slightly to the center of the model, and the cooling air is divided into two streams as shown in Fig 5.
As a result, a part of cooling air (air from the inlet to the center of the outlet) flows over the PBGA. In the case of $X_{in} = 150$ mm, almost all of cooling air linearly flows to the outlet on the right, and the flow over the PBGA is weak.

Before discussing the relationship the flow patterns and the temperature rise, we compare the discrepancy between the experimental results and the numerical results. Velocity distributions along the $X$ direction at various $Y$ positions are shown in Fig. 6. The points are the experimental results and the line means the numerical results. Fig. 6 (a) is the case of $X_{in} = 100$ mm, Fig. 6 (b) is the case of $X_{in} = 130$ mm, and Fig. 6 (c) is the case of $X_{in} = 150$ mm. It is confirmed that high velocity region shifts to the right side as the inlet moves to the right side. Negative velocities near the sidewall can be observed in both the experimental and the numerical results.

![Profiles of streamwise velocity component at $Z = 7$ mm](image)

Figure 6. Profiles of streamwise velocity component at $Z = 7$ mm

Velocity profiles from the numerical results show almost good agreement with the experimental results. However, the discrepancy can be observed. The cause of the discrepancy will be assumptions in the present numerical simulation. Although the model includes obstacles, laminar flow has been assumed in the numerical simulation. Flow in the model would be in
transitional regime because Reynolds number based on the inlet velocity and the hydraulic diameter of the inlet is about 2400. However, it is concluded that the present calculation will simulates the flow in the experiment.

Then we discuss the relationship between the air flow patterns and the temperature rise. Figure 7 shows air velocity dependence of the temperature rise. The horizontal axis shows the inlet position ($X_{in}$), the left vertical axis shows the temperature rise, and the right vertical axis shows the average air velocity over the PBGA package. Solid square means temperature rise and open circle is average air velocity. From this figure, it can be seen that temperature rise becomes higher with decreasing average air velocity as we expected. In other words, maximum temperature on the PBGA package higher as the inlet position shifts to the right side. From those results, it is concluded that the inlet, outlet and PBGA package should be in straight line for effective cooling in the case that many outlets are located in an enclosure.

![Figure 7. Relation between the inlet position and temperature rise $\Delta T$ of the PBGA](image)

In this study, experiments were conducted with a relatively simple configuration. In this case, simple CFD calculation can well simulate the flow pattern in the experiment. As a result, we can estimate the inlet location for the most effective cooling from simple CFD calculation.

Benchmark data with more complex and realistic configurations will be obtained in the successive work of JSME project. For example, the case of several heat sources, outlets on the side walls and more obstacles will be discussed. Further, higher Reynolds number case will also be considered and the limit of the assumption of laminar flow will be evaluated.

5. Conclusions
The relationship between the cooling performance and air flow pattern has been examined experimentally in the case of the model of a thin electronic equipment. Air flow depending on the position of the air inlet was clarified using a PIV. It is confirmed that cooling performance can be improved by leading flow to heat source. CFD analysis was also conducted and the results of the flow pattern showed good agreement with the experimental results. It is concluded that effective cooling performance can be estimate from simple CFD analysis. In the present study, a relatively simple configuration was examined. In our future study, more complex and realistic configurations will be discussed. The configurations are, for example, the case of several heat sources, outlets on the side walls and more obstacles. Further, in the case of higher Reynolds number, the assumption of laminar flow will show large discrepancy between the calculation and the experimental results. Therefore, relations between the limit of the assumption of laminar flow and Reynolds number should be evaluated, and laminar flow analysis will be compared with turbulent flow analysis. From the present results, it is obvious that flow pattern is one of the important factors for efficient cooling. So as to estimate flow pattern in complex cases, details
of the relationship between flow patterns and the pressure of outlets will be discussed in our future work.

References
[1] H. Nakamura, “Cooling Fan Model for Thermal Design of Compact Electronic Equipment: Improvement of Modeling Using PQ Curve”, Proc. of InterPACK’09, IPACK2009-89010, 2009.
[2] G. V. Shankaran, M. B. Dogruoz, “Validation of an Advanced Fan Model with Multiple Reference Frame Approach”, Proc. of ITherm 2010, 2010.
[3] T. Fukue, T. Hatakeyama, M. Ishizuka, K. Koizumi, “Relationship Between Flow Pattern in Front of Fans and Decreases of Fan Performance”, Proc. of ISTP22, No. 152, 2011.
[4] K. Koizumi, “Thermal Analysis of Switch Mode Poser Supply”, Proc. of ECO-MATES 2011, TAE-2, 2011.
[5] W. Nakayama, “An approach to fast thermal design of compact electronic systems: A JSME project”, Proc. of IPACK’01, IPACK2001-15532, 2001.
[6] W. Nakayama, R. Matsuki, Y. Hacho, and K. Yajima, “A new role of CFD simulation in thermal design of compact electronic equipment: application of build-up approach to thermal analysis of a benchmark model”, Proc. of ASME IMECE 2003, IMECE2003-42181, 2003.
[7] W. Nakayama, T. Nakajima, H. Koike, and R. Matsuki, “Heat Conduction in Printed Circuit Boards - Part I; Overview and the Case of a JEDEC Test Board”, Proc. of IPACK2007, IPACK2007-331605, 2007.
[8] W. Nakayama, T. Nakajima, H. Koike, and R. Matsuki, “Heat Conduction in Printed Circuit Boards - Part II; Small PCBs Connected to Large Thermal Mass at Their Edge”, Proc. of IPACK2007, IPACK2007-331606, 2007.
[9] Masaru Ishizuka, Shinji Nakagawa and Tatsuro Yoshida, and Wataru Nakayama, “Synthesis of CFD Analyses and Experiments in Developing a Thermal Network Model of A Simulated Heat Spreader Panel”, Proc. of IMECE2008, IMECE2008-66091, 2008.
[10] http://www.openfoam.com/
[11] H. G. Weller, G. Tabor, H. Jasak and C. Fureby, “A Tensorial Approach to CFD using Object Oriented Techniques”, Computers in Physics, Vol. 12, No. 6, pp 620 - 631, 1998.