

**SYSTEM LEVEL THERMAL
SIMULATION OF A
COMPUTER CHASSIS
USING
FLOTHERM SOFTWARE**

by

Tom Lee & Mali Mahalingam
Advanced Packaging Development Center
Motorola, Inc
5005 E. McDowell Road, Mail Drop B-136
Phoenix, Arizona 85008

The above paper was published as part of the FLOTHERM International User Conference Presentations, May 1993

FLOTHERM® is a registered trademark of Flomerics Limited.

SYSTEM LEVEL THERMAL SIMULATION OF A COMPUTER CHASSIS USING FLOTHERM SOFTWARE

Tom Lee and Mali Mahalingam
Advanced Packaging Development Center
Motorola, Inc.
5005 E. McDowell Road, Mail Drop B-136
Phoenix, Arizona 85008

ABSTRACT

A CFD tool, FLOTHERM, is used to evaluate the velocity and the temperature fields of air flow in a computer system enclosure. Simulations focused on the six printed circuit board regions where approximately 37 W of power was generated on the component side of the board. Three-dimensional, steady-state, turbulent air flow is assumed. Both straight and swirl flow cases at fans were considered. The actual air flow rate at fans was determined by matching the predicted pressure rise across the fan to the given fan characteristics.

Experiments were performed under real life conditions. The actual computer chassis was tested inside a temperature-controlled chamber. Air temperatures between two printed circuit boards were measured in various positions and compared with the simulations.

Results demonstrate a reasonable agreement between the temperatures obtained by simulations and measurements. In all comparisons, average errors were less than 10%. The introduction of swirl at fans made no significant effect on temperatures. Hot spot appeared in the inner side of the air passage between boards; this hot spot was created due to the recirculation of low velocity air. Some recommendations on improvements of the simulation will also be presented.

I. INTRODUCTION

In recent years, many powerful Computational Fluid Dynamics (CFD) tools have been developed and applied in the electronic cooling industry to help reduce the dependency on extensive and costly experimentation. By introducing this numerical analysis tool in the early system design stage, the number of prototype hardware revisions can be minimized and the overall turnaround time from design to production can be reduced.

The need for experimentation exists; in most cases, experimental data are required to provide necessary information to the simulations. It is also used to verify the predicted results. Once the physical models are built and levels of confidence in predicted results are established, the simulation tool can be applied in parametric studies (such as factorial design) leading to an optimum product design in a shorter design time.

One unique feature of a CFD tool lies in its global system level simulation by providing accurate boundary conditions for detailed component level analysis. Thermal simulations at component level packages typically use detailed 3D software (finite element or finite difference based) to analyze the package. The boundary conditions that control the cooling of the package are typically treated using heat transfer coefficients derived from suitable correlations. Flow fields (free or forced air) that exist near the component package can be greatly different in an actual equipment compared to what is typically assumed in component thermal modeling. CFD software enables one to determine the velocity flow fields in an equipment leading to more accurate boundary conditions. This paper will describe such analysis methodology with a case study including supporting experimental verification.

The following sections describe the processes of using FLOTHERM (V. 1.2) to evaluate the velocity and the temperature fields of air flow in a computer system enclosure. The computer system to be studied is described initially, followed by the simulations, experimentation, results, discussion, and conclusions.

II. SIMULATED COMPUTER SYSTEM

The computer system enclosure to be simulated is a Motorola's Delta Series Model 8440. The Delta Series provides UNIX-based, multi-user distributed and networked systems. The basic structure of this computer is shown in Fig. 1. In the front, the enclosure has six VME compatible Printed Circuit Boards (include main logic boards, memory boards, and system controller boards); each board can generate an average power of 37 W. In the back, the enclosure includes an 8-slot transition module for I/O connectors. On the top, the enclosure (both front and back) houses peripheral bays for mass storage devices. At the bottom of the enclosure is a 400 W power supply. The enclosure is cooled by forced air. Each multifan can deliver up to 80 m³/h (47.1 CFM) of flow rate. This enclosure is 574 mm high, 203 mm wide, and 483 mm deep.

The direction of air flow in the computer chassis can be visualized by the side and top views of the enclosure (Figs. 2a, b). As shown in Fig. 2(a), air was drawn from outside, blew towards the VME PCBs through a converging duct. The position of each VME PCB is numbered; air passage #1 defines the gap between PCBs #1 and #2, etc. (Fig. 2b). Since two edges of the PCBs were sealed, air was forced to flow downwards in the gap between two PCBs. As the air reached the bottom of the enclosure, the flow was divided into two directions: forward flow releasing through a vent; backward flow through the power supply releasing from another vent.

III. SIMULATIONS

Basic Assumptions

Three-dimensional, steady-state turbulent air convection in the electronic enclosure was assumed. The Reynolds number based on the fan inlet velocity and the air gap between the PCBs was larger than 3000. Preliminary studies assuming laminar air flow overestimated air temperatures and created large errors when compared to the experimental data. Turbulent air flow was assumed subsequently. Constant properties for air were assumed in all computational domains (based on 20°C ambient temperature). It was anticipated that the air temperature within the enclosure was less than 50°C, and the variation of fluid properties in this temperature range was insignificant. Buoyancy effect was included. Contributions from radiation were ignored.

Overall Computational Domain

The focus of this study is to predict air flow and temperature fields in the VME PCBs region; the mass storage devices and the I/O cards were neglected in the simulations. Several built-in menus such as Cuboid, Triangle, Internal Plate, External Fan, Vent, and PCB were used to construct the model. All the internal structures of the enclosure were modeled as thin plates. These plates were smooth on both sides and treated as adiabatic boundary conditions. Power supply was modeled as blocks.

A total of ~43,300 grid points were assigned to the overall domain. Cells of large aspect ratio were not created. The grid spacing was carefully selected till the computational results were least affected by the change in grid spacing. The air gap between centers of two PCBs is 20 mm. Fine

grids were chosen in the air gap near the component side of the PCB; they were located 2 mm, 3 mm, and 7.7 mm away from the components.

PCBs

A PCB menu was used to simulate the VME PCBs rack. The rack contained six, equally spaced PCBs. For each PCB, a smooth surface on the non-component side was assumed. On the component side, the presence of the components was accounted by assuming surface friction; the surface was treated with roughness height equal to the average component height (≈ 4 mm). Simulations were performed six times with only one PCB being powered at a time.

Total heat generation from the components was 37.6 W per board. The PCB menu required an input of the percentage of heat dissipated to the air on the non-component side of the PCB. Experiments were performed to estimate this proportion. Details of the experiments are discussed in Sec. IV Temperature Measurements.

Fans

The actual air flow rate delivered from two multifans was unknown and controlled by the flow impedance of the enclosure. To estimate the actual air flow rate at fans, two methods were used and discussed here. In the first method, for each multifan, it was assumed two external fans sandwiched a block. Both external fans have the same flow direction and deliver the same mass flow rate. It is necessary to adjust the flow rate to correspond to the predicted pressure rise across a fan to match the given fan characteristics (provided by the manufacturer). Three flow rates (34, 51, and 68 m³/h) were assumed and the intersection of the calculated system operating curve and the fan characteristic curve determined the actual flow rate at the fan.

The second method used the Fan Characteristic Option of the Internal Fans in FLOTHERM v. 1.3. For each multifan, a cuboid block represented the hud with four plates represented the cowl. The internal fan was located at the center of the block and bounded by the plates. Parameters P_0 and m_0 were given from the fan characteristics. Fine grids were applied on both the intake and discharge faces of the fan. FLOTHERM 1.3 computed the actual air flow rate automatically.

Results from these two methods are shown in Fig. 3. The first method predicted the actual air flow rate at the fan to be 51 m³/h. This value gives an average normal velocity of 1.71 m/s at the discharge face of the fan. The second method obtained a similar result: the estimated air flow rate is 48 m³/h with a normal velocity of 1.61 m/s. Data from method 1 is used in this study.

Fans with swirl are also considered in the simulation. Generally, the amount of rotational air at fans is small (compared to the normal flow), unpredictable, and is usually ignored. To verify the effect of swirl on the system, each multifan was replaced by four small, equal-area fans. Each small fan delivered a straight flow with the same normal velocity of 1.71 m/s; it also delivered two by-directional flows with velocities equal to 10% or 20% of the normal velocity (1.71 m/s). The plane of these by-directional flows is perpendicular to the normal velocity. With this arrangement, the combination of angular flows from four small fans forms a swirling flow while maintaining the same total mass flow rate as in the straight flow case.

Mathematical Model

FLOTHERM solves fundamental equations for conservation of fluid mass (continuity), fluid momentum, and thermal energy in fluid and solid. A control-volume, finite-difference procedure was used to solve the governing equations [1]. For turbulent flow, the effects of turbulence are expressed through turbulent viscosity (μ_t) and turbulent conductivity (k_t). The constant turbulent

viscosity model is used. The turbulent conductivity was calculated by assuming constant turbulent Prandtl number, $Pr_t=0.9$.

In the near wall region, turbulence is damped, and the turbulent viscosity (μ_t) becomes comparable to, and even smaller than, the laminar viscosity (μ). A well-established "universal" relationship is used to describe the velocity profiles in this region. Special treatment on rough walls was also applied to components on PCBs. For heat transfer in the near wall region, a "Taylor-Prandtl analogy" is applied to calculate "Stanton number". Details of these models are explained in Ref. [2].

Boundary Conditions

The external surfaces of the enclosure were assumed to be adiabatic. For the two outlet vents (Fig. 2a), a constant external static pressure boundary condition was applied. For the inlet vent, a constant external stagnation pressure boundary condition was applied. Since either a grille or a perforated plate was located at the vent, the amount of pressure drop across the vent needs to be considered. In the menu, the device velocity was chosen and the loss coefficient was set to be 1.5. It gave a pressure drop equivalent to ~4 dynamic heads for the outlet vents and ~3 dynamic heads for the inlet vent.

IV. TEMPERATURE MEASUREMENTS

Experiments were performed on a Delta Series computer chassis to measure air temperature in the passages between the PCBs. The mass storage devices and I/O PCBs were removed from the enclosure. Six PCBs were inserted into a 6-slot VME compatible card cage. Each board had a 10x10 array of resistors with a total power dissipation of 37.6 W. Each resistor was 4 mm high. These resistors simulated the actual components on the board. During the experiment, only one PCB was powered and this powered PCB was switched from slot 1 to slot 6 (Fig. 2b).

Nine T-type (copper-constantan) thermocouples (with a $\pm 0.1^\circ\text{C}$ uncertainty) were attached along the component side of the powered PCB. The front ends of these thermocouples were tilted normal to the PCB, and the tips were positioned in the air ~2 mm above the components. From these nine thermocouples, air temperatures between boards were measured. As mentioned in Sec. III. PCBs, experiments were run to estimate the proportion of heat dissipated through the non-component side of board. This was done by measuring board temperatures on both sides and then calculating the conduction heat loss through the board. Board temperatures were measured by additional six T-type thermocouples that were positioned on both sides of the powered PCB (three thermocouples on each side). The tips of the thermocouples were bonded against the board by a high-conductivity epoxy. The locations of all these thermocouples (15) on the PCB are shown in Fig. 4.

The whole computer chassis was put into a temperature-controlled chamber; the chamber temperature was set to 20°C ($\pm 0.5^\circ\text{C}$). The computer system was turned on and the power was supplied to the resistors and multifans. When the temperature reached steady-state, five consecutive readings at each thermocouple position were recorded and averaged. The uncertainty of the average value was less than $\pm 0.7^\circ\text{C}$ (combination of uncertainties from the thermocouples, thermocouple meter, and the standard deviation of the five readings).

Results showed that 8%-24% of the total heat from the components dissipated through the non-component side of the board to the air; this variation depended on the location of the powered PCB. These values were used in the simulations.

V. SIMULATION RESULTS

Velocity Field

Fans Without Swirl. The top view of the velocity field is shown in Fig. 5. This view is located on a plane 20 mm above the VME card cage. The locations of each PCB and air passage are referred to Fig. 2(a). Clearly, as the flow reached the front end of the enclosure, a recirculation zone was created and reverse flow was introduced towards the PCB regions. It is expected that as the flow moved down closer to the top of the PCB regions, some reverse flow will entrain into the passages between PCBs. Fig. 5 suggested that the strongest reverse flow with entrainment exists in air passage #4.

The side views of the velocity fields in air passages #1 to #6 are shown in Figs. 6(a)-(f). These velocity fields are located on a plane 2 mm away from the components. They indicate that reverse flow existed only at the top of air passages #1 to #4. In Fig. 6(a), the forward flow still dominated the flow field above the PCBs region and no reverse flow entrained into passage #1. Figs. 6(b)-(d) show the reverse flow merging with the forward flow at the top of the air passages, and the strongest entrainment appearing in passage #4. No entrainment occurred in passages #5 and #6 (Figs. 6e, f).

Fig. 6 also shows that large stagnation zones existed in the inner portions of passages #1 to #5 and died out in passage #6. These stagnation zones lead to recirculation. Since low velocity air exists in the recirculation zone, it is expected that hot spot will appear in these regions.

Fans With Swirl. With 10% or 20% of swirl at fans, the velocity field was changed. The top view of the velocity fields for both swirls are shown in Fig. 7. Clearly, due to the introduction of swirl, the recirculation zone as observed in Fig. 5 shifted toward the positive-z direction and the strongest entrainment may exist in air passage #1. Both 10% and 20% of swirl have similar flow patterns; though 20% of swirl (Fig. 7b) shows a greater penetration of reverse flow beyond passage #1 (in the positive-z direction).

Examples of velocity field at side view are shown in Fig. 8. These are velocities in passages #1 and #4 with 10% of swirl. In contrast to the cases without swirl (Figs. 6a, d), a strong penetration from reverse flow appeared in passage #1 (Fig. 8a), and no reverse flow with entrainment existed in passage #4 (Fig. 8b). Similar patterns were seen with 20% of swirl.

Large stagnation zones were also found in passages #1 to #5 for both swirl cases and no recirculation zone existed in passage #6.

Temperature Field

Fans Without Swirl. The side views of the temperature contours in the flowing air stream in passages #1, #4, and #6 are shown in Fig. 9. After air entered the passages (~20°C at inlet), temperature increased as the air passed downward over the heated boards and evident temperature gradients were observed. For passages #1 to #5, similar patterns of temperature contours are seen; temperature field was distorted on the inner side of the passage by the recirculation, which created a hot spot. This hot spot ranged from 37°C to 45°C. For passage #6, the temperature field was uniform due to the absence of the recirculation. In this case, air temperature reached ~38°C as it left the PCB.

Fans With Swirl. With 10% of swirl, examples of temperature contours are shown in Fig. 10. Generally, they have similar patterns as the cases without swirl. Fig. 10(a) shows that the hot spot expanded from the inner side of the passage to the lower region of the passage; this is different from Fig. 9(a). Similar results were obtained for 20% of swirl.

VI. EXPERIMENTAL VALIDATION AND DISCUSSION

Comparisons of temperatures between simulations and measurements in air passages #1 and #4 are shown in Fig. 11. In general, the calculated results agreed well with the measured data. The simulated results indicated that no significant difference existed between the cases with and without swirling flow (for both 10% and 20% swirl). It is noticed that although thermocouples #3, #6, and #9 were not necessarily located in the hot spot region, they measured warmer temperatures in this area.

The accuracy of the temperature simulations is summarized in Table 1. The average error is defined as the average of percentage errors (in nine thermocouple locations) between measured and calculated data. The maximum and minimum errors define the largest and the smallest percentage errors in the nine thermocouple locations. In all six passages (for both cases with and without swirl), the average predicted errors from simulations were less than 10%. Data from passage #4 (without swirl) indicated that large errors would be created if the computational plane did not match the measurement plane. By introducing swirl (10% or 20%), the results did not improve. This suggested that the effect of rotational flow was damped out as the flow entered the PCB regions.

As described earlier, several assumptions were made in the simulation. These assumptions such as the estimate of heat dissipation through the non-component side of the board, the pressure drop across the vent, and the amount of swirl at the fans can be further refined or justified by detailed preliminary experiments. Velocity measurement is also important to verify the computed air flow rate at the fans and the flow patterns in the PCB regions.

The constant turbulent viscosity model has a good representation of average turbulence in a large scale computational domain. As the computational domain becomes small, for example, focusing on a single PCB region or on component level, an improved turbulent model (such as the k- ϵ model) is recommended which can characterize local turbulence more accurately.

The current model that assumed the component side of the PCB as a rough surface (with roughness height equal to the component height) can be improved by treating the components as an array of blocks. Once the heat source is assigned to each block, both the component and flow temperatures can be calculated simultaneously. However, in a global system level analysis, detailed thermal simulations on PCBs at component level are usually prohibitive. This is due to the limitations of computational time and the available computer memory for fine grids. One of the alternatives is to use the calculated velocity and temperature fields (located near the PCB regions) to be the boundary conditions and solve the components on the PCBs as a conjugate problem separately. The other option is using these boundary conditions making an estimate of local heat transfer coefficients and solving the components on the PCBs as a 3D conduction problem.

VII. CONCLUSION

This case study demonstrates the capability of using FLOTHERM software in predicting flow field and heat transfer in an air-cooled computer system. Further, it marks an enhancement in package thermal design capability to meet system level thermal design needs. Experimental data show a reasonable agreement between simulations and measurements. In all comparisons, average errors were less than 10%. Both experiments and simulations indicated that hot regions existed in the inner portion of the air passages; these hot spots were created due to the recirculation of low velocity air. Design engineers can use this information for system level revisions. If these hot regions are not desirable, or lower air temperatures in the PCB regions are required, system designers can apply simulations on various chassis models to achieve an optimum design. Only a minimum amount of prototypings would be needed.

ACKNOWLEDGEMENTS

The authors would like to thank John Gatti of Motorola Computer Group, who recommended the case study and offered the hardware chassis for simulation. Technical support from Dr. Ian Clark of Flomerics is also greatly appreciated.

REFERENCES

- [1] S. V. Patankar, *Numerical Heat Transfer and Fluid Flow*, New York, Hemisphere Publishing Corp., 1980.
- [2] "How to Use FLOTHERM Lecture Course," 1990, Document No. FLOTHERM/LC/0790, Flomerics Limited, The Kingsgate Business Centre, Kingston upon Thames, Surrey, England.

Table 1 Accuracy of the Temperature Simulations

Fans Without Swirl							
Locations of the Air Passage	#1	#2	#3	#4		#5	#6
Distance from the Components (mm)	2.0	2.0	2.0	2.0	3.0	7.7	2.0
Average Error (%)	7.1	4.5	7.9	6.7	9.5	26.7	3.7
Maximum Error (%)	13.3	10.3	16.1	13.5	18.9	35.1	10.4
Minimum Error (%)	0.5	0.2	2.2	0.6	0.0	17.4	0.3
Fans With Swirl							
Locations of the Air Passage	#1	#2	#3	#4		#5	#6
Distance from the Components (mm)	2.0	2.0	2.0	2.0	2.0	2.0	2.0
Percentage of Swirl (%)	10	10	10	10	20	10	10
Average Error (%)	6.6	4.6	7.9	6.4	7.1	4.0	7.0
Maximum Error (%)	18.2	11.5	16.9	15.1	15.1	8.4	18.8
Minimum Error (%)	3.2	0.4	2.1	0.4	1.2	0.7	1.3

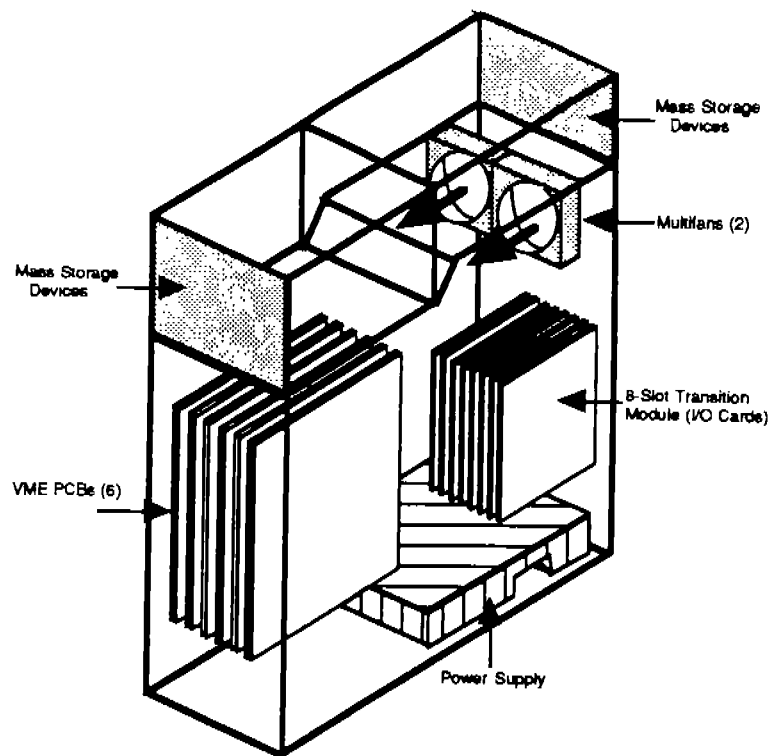


Fig. 1 Delta series computer system

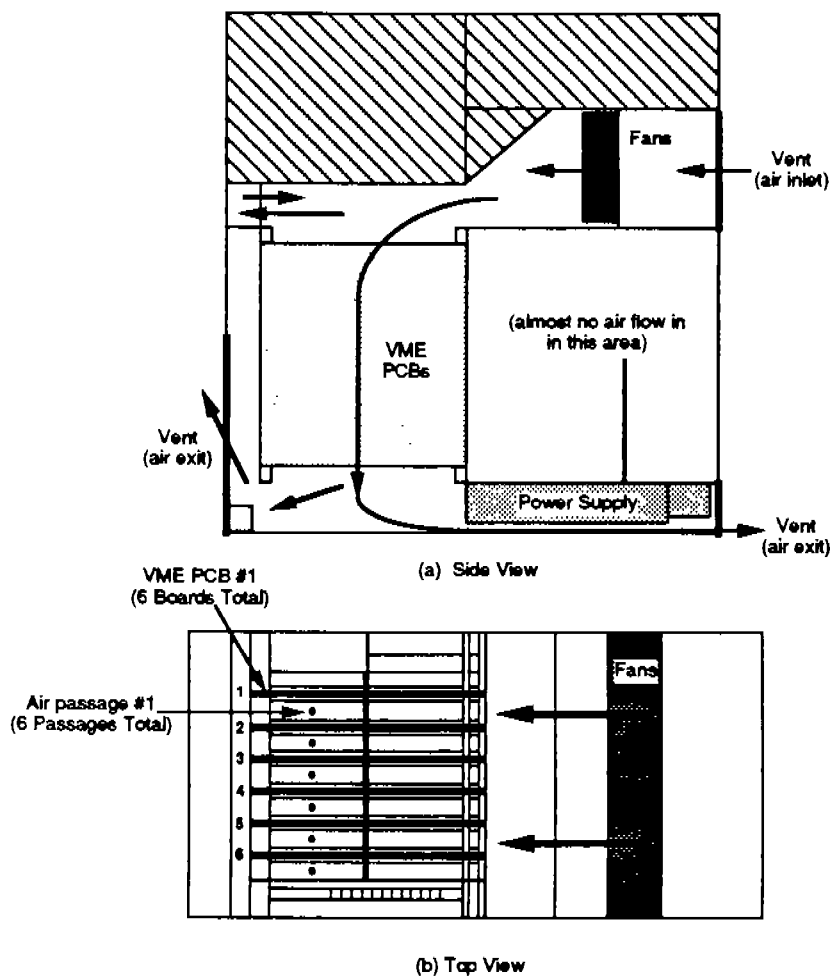


Fig. 2 Air flow in the system enclosure

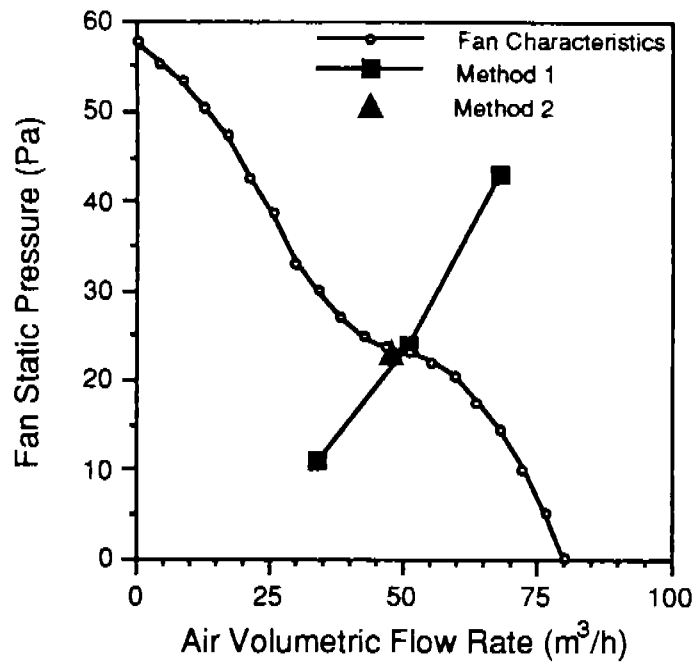


Fig. 3 Estimation of air flow rate at fan

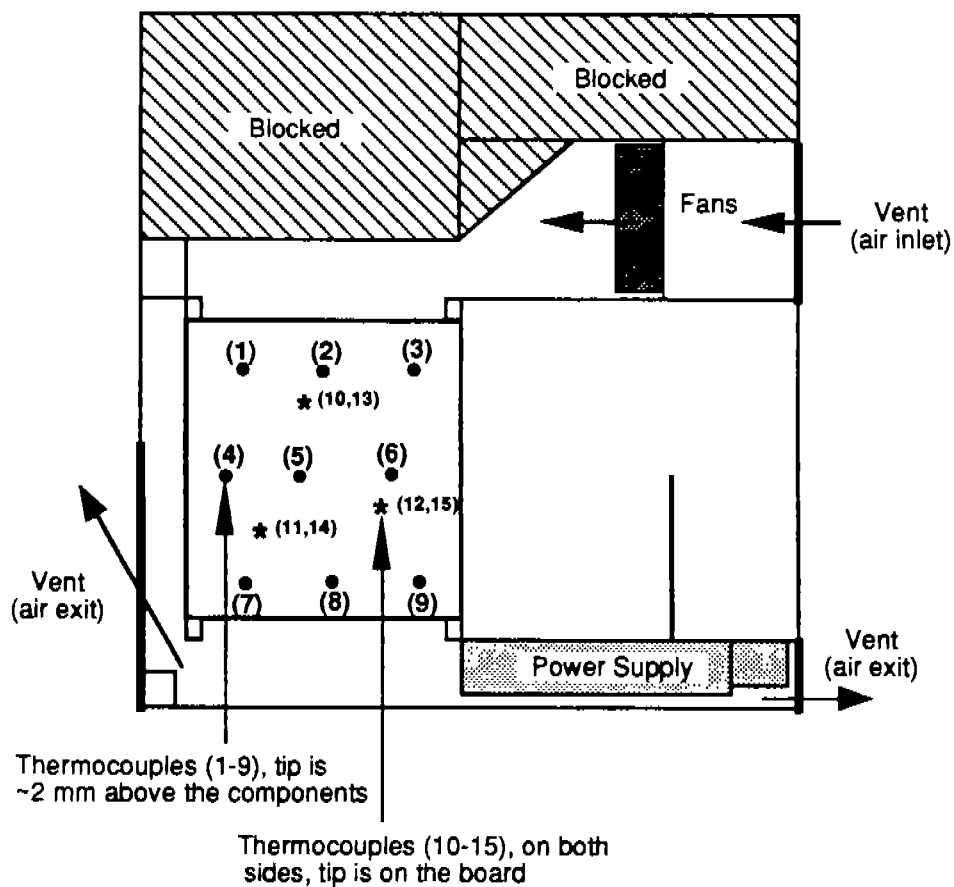


Fig. 4 Thermocouple positions

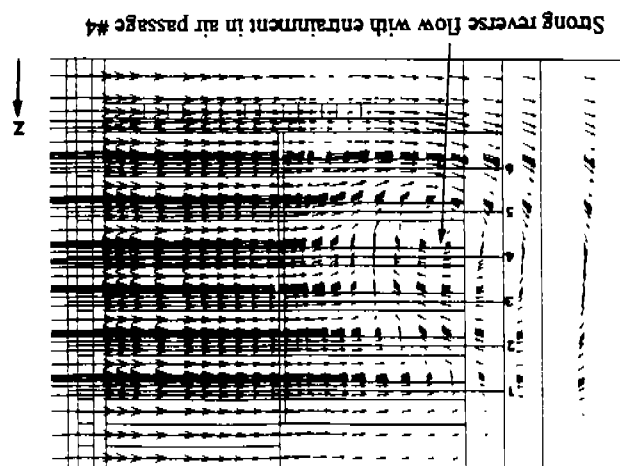


Fig. 5 Top view of the velocity field (fans without swirl)

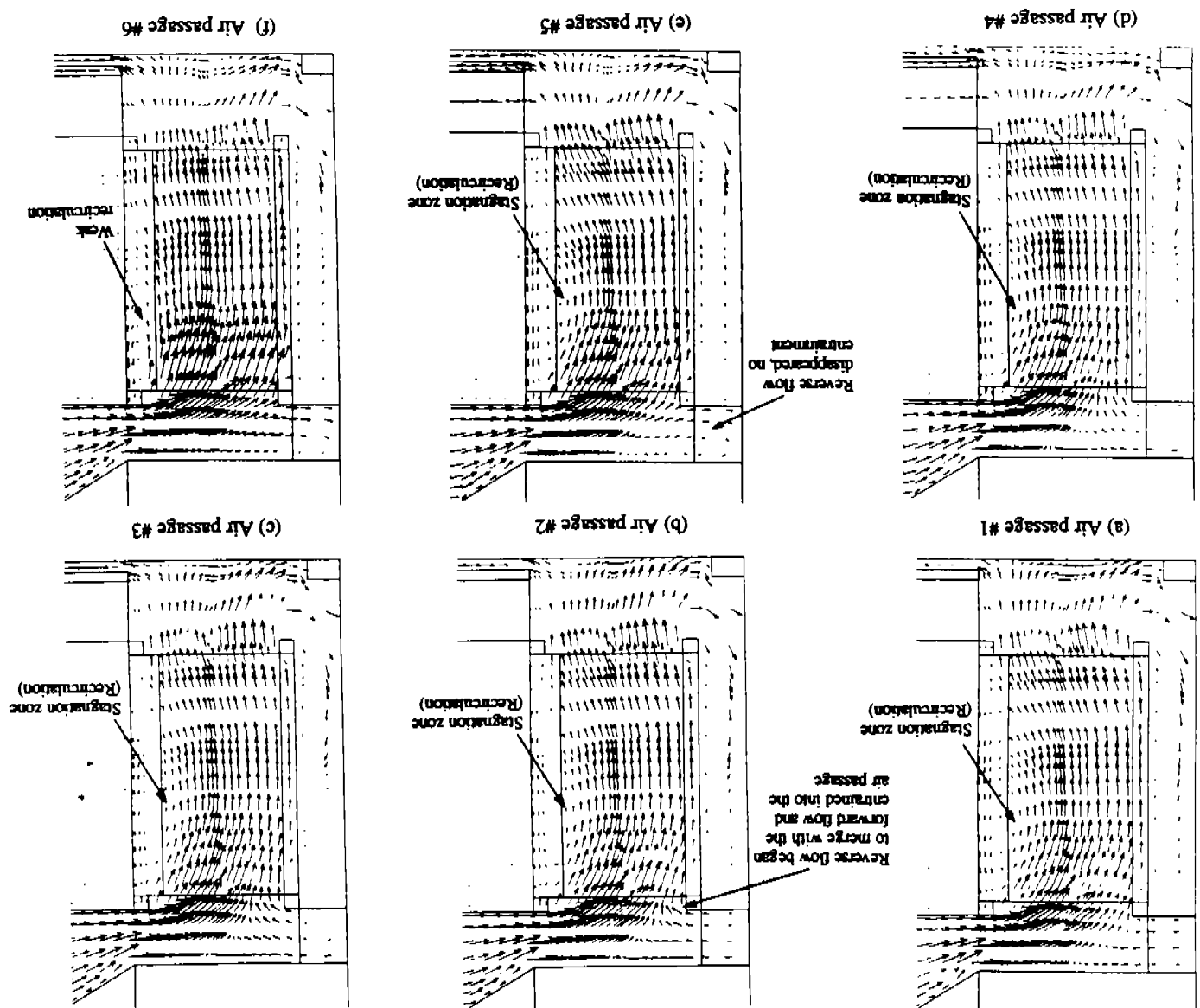


Fig. 6 Side view of the velocity field (fans without swirl)

Fig. 7 Top view of the velocity field (fans with swirl, at 20 mm above the VME card cage)

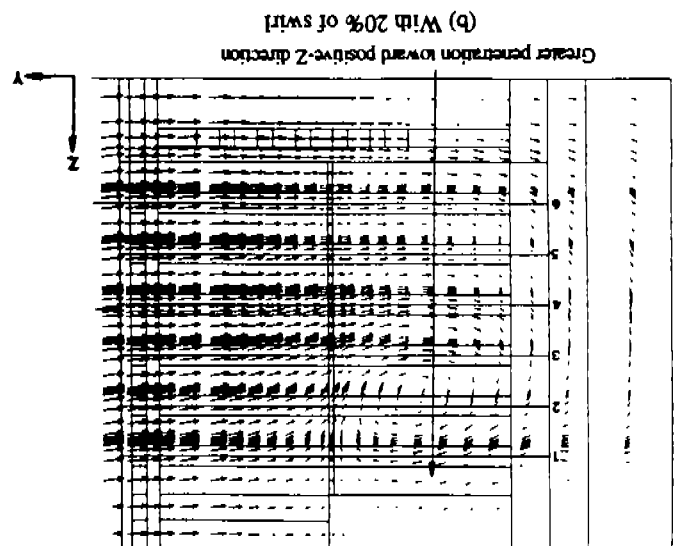
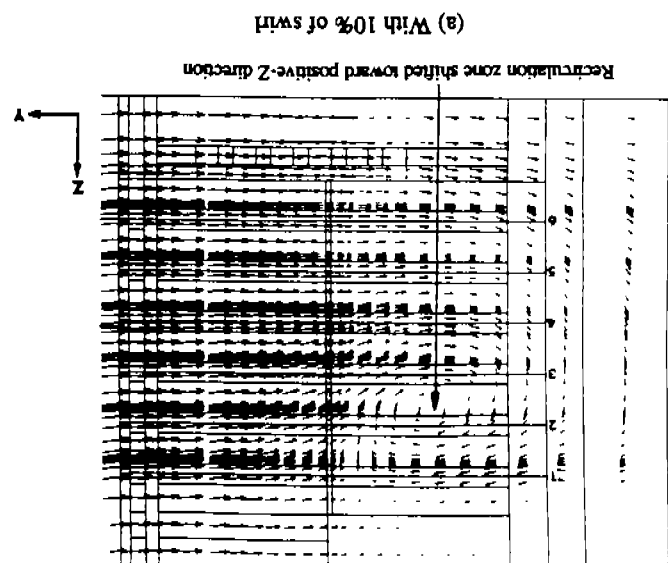


Fig. 8 Side view of the velocity field (fans with 10% swirl, at 2 mm away from the components)

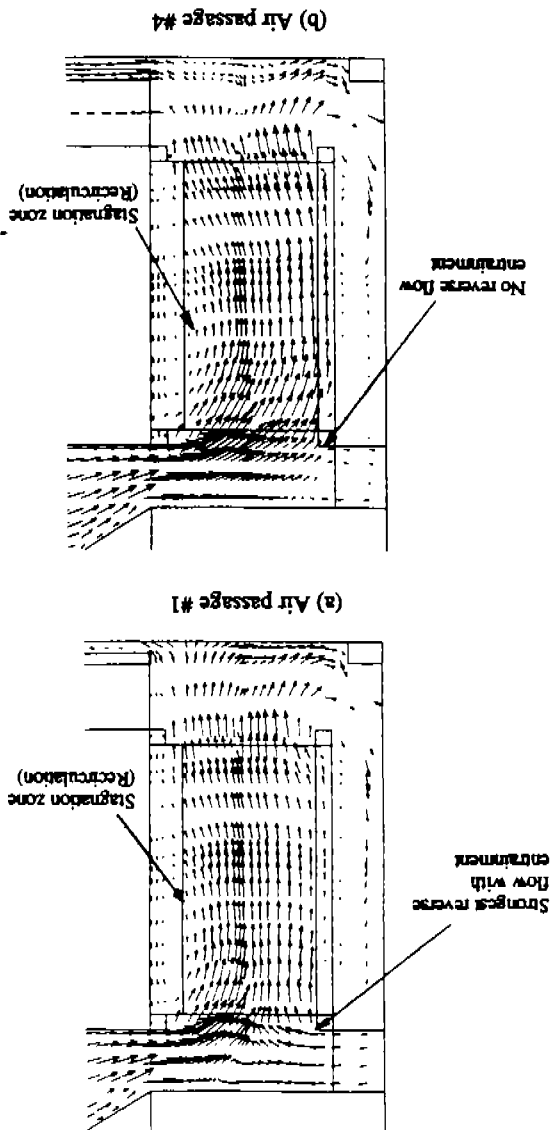
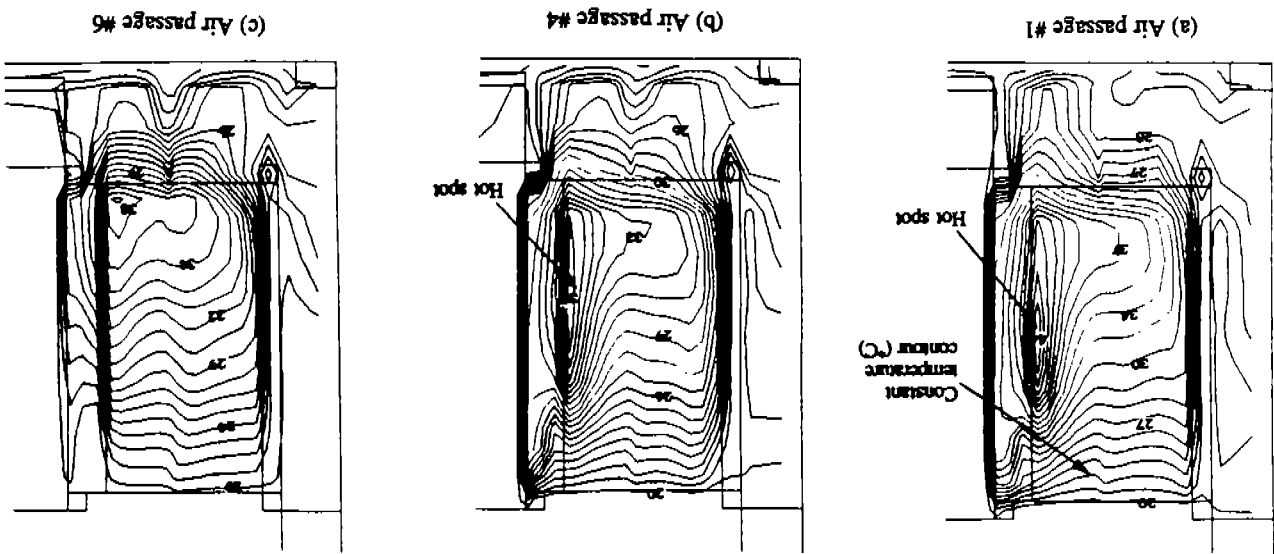
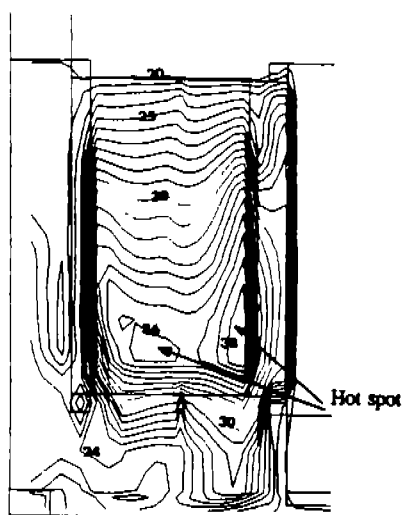
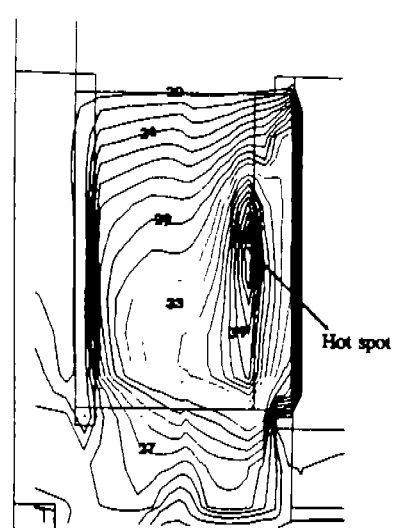


Fig. 9 Side view of the temperature field (fans without swirl, at 2 mm away from the components)



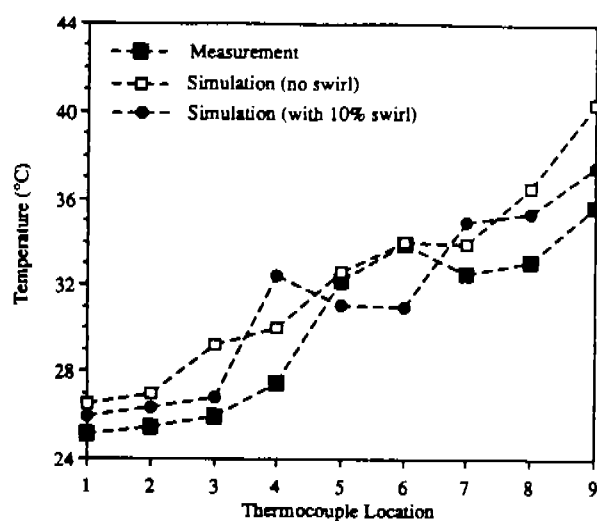


(a) Air passage #1

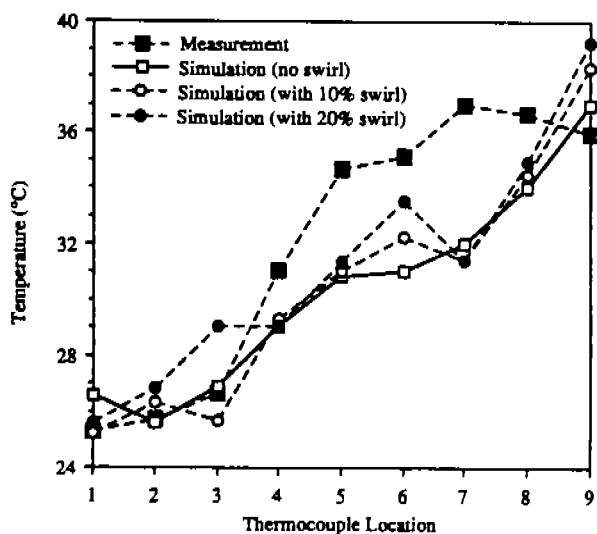


(b) Air passage #4

Fig. 10 Side view of the temperature field (fans with 10% swirl, at 2 mm away from the components)



(a) Air passage #1



(b) Air passage #4

Fig. 11 Temperature comparison (at 2 mm away from the components)