Thermal Analysis of an Electronics Enclosure: Coupling Flow Network Modeling (FNM) and Computational Fluid Dynamics (CFD)

Tom Kowalski
Fusion Design
Campbell, CA 95008
Tom@fusionnet.com

Amir Radmehr
Innovative Research, Inc.
Plymouth, MN 55447
Amir@inres.com

Abstract

The objective of the present study is to perform thermal analysis for the design of a complex electronic cabinet used as a high speed internet subscriber device. The novelty of the paper lies in the approach used for analysis of the flow distribution in the system of interest. It utilizes Flow Network Modeling (FNM) and Computational Fluid Dynamics (CFD) in a complementary and interactive manner for quick and accurate thermal analysis of the entire system. The enclosure consists of thirteen PCB's and two axial fans. In the analysis of the flow distribution, the flow through the passages formed by adjacent PCBs along with the inlet region is analyzed using the CFD technique to generate flow impedance characteristics. A flow network model of the entire system is then constructed by interconnecting the various components of the system in a manner that represents the paths followed by the air as it moves through the system. The CFD-based impedance characteristics in combination with the loss characteristics available from handbooks are ascribed to individual components in the network model. The FNM-based analysis of the entire system accounts for the interaction of the fan curve and the flow impedances to predict the flow distribution of air throughout the system. These results are, in turn, used to provide boundary conditions in the CFD analysis for the prediction of detailed flow distribution in individual card passages in order to obtain a thermal map of the PCBs. The predicted flow rates through the individual card passages are within 10% of the experimentally measured values.

The analysis approach couples the power of CFD with the speed and flexibility of the FNM technique to enable accurate prediction of the flow distribution throughout the system in the most efficient manner. Further, the modularity of the proposed approach allows quick and scientific examination of the design changes such as use of different filters, screens, or fans and easy identification of performance-limiting components. The complementary use of CFD and FNM reduces the time required for thermal analysis by an order of magnitude over the approach that uses only the CFD technique.

I. Introduction

The demand for high-speed network devices is at an all time high. The trend to accompany this demand is to design the devices to be compact and modular. Designing these high-powered systems to provide adequate cooling in a compressed design cycle frequently presents difficult challenges to mechanical engineers and analysts. Specific details of such systems that create a hostile thermal environment include elevated inlet temperatures, the use of high-powered components, high PCB density, and sometimes unconventional configurations that facilitate modularity of multiple systems installed in a facility. Often these systems are stacked in an array such that there will be identical units placed side by side, above and below.

Early in the design phase, the thermal designer must construct a model that will allow for performance evaluation, optimization, and simulation of fail safe modes such as fan failure. The traditional approach in designing and analyzing such complex systems is to model the fluid behavior and heat transfer using Computational Fluid Dynamics (CFD). While CFD provides detailed information about the flow and thermal behavior, it is also extremely time intensive and its use is considered to be impractical to analyze multiple configurations of a large thermal system such as a complex network device. Empirical testing works very well but requires that a working prototype be available early in the design cycle, which is also very impractical. A very simple and efficient method for system-level analysis of a complex system is the technique of Flow Network Modeling (FNM). In the FNM approach, the system is represented as a network of passages, fans, inlets, exits, filters, and perforated grilles for the prediction of systemwide flow and temperature distribution. Use of this technique for the design of electronics systems was first proposed by Ellison [1]. A generalization of the underlying methodology to complex systems and its use in the design of practical electronics cooling systems have been reported by Belady et al. [2], Steinbrecher et al. [3], Kelkar et al. [4], and Radmehr et al. [5]. Construction of the FNM model requires only a small fraction of the time required to set up a CFD model and the calculation of the coolant rates and temperatures occurs virtually instantaneously. However, for some systems, FNM alone is not sufficient for a complete analysis because of the lack of accurate information about the flow impedances of parts of the system. In the present study, a hybrid approach involving complementary use of CFD and FNM is used for an efficient analysis of the flow distribution in an electronic cabinet.

The cabinet analyzed in the present study involves several PCB cards oriented vertically in a cabinet. The flow through the cabinet is driven by two fans mounted above the PCBs. The electronic components and heat sinks mounted on the PCBs create complex flow passages between them. CFD analysis is used to determine the impedance characteristics of flow through the card passages along with the inlet region consisting of a tapered bottom wall and an EMI screen. These characteristics along with the data from handbooks are used to perform a network analysis of the cabinet to predict the distribution of flow rates throughout the system. A comparison between the predicted and measured values of the flow rates is also presented. This complementary approach uses the ability of the FNM technique to quickly analyze the systemwide interaction of the various parts of the system and the capability of the CFD technique to predict flow details in complex geometries. The result is that the time required for a complete analysis is reduced by orders of magnitude compared to the use of the CFD technique alone. Further, the resulting methodology is very modular in nature and is ideally suited for virtual prototyping of electronic cabinets in a rapid fashion.

II. Description of the Physical system

Figure 1 shows a rendering of the test model, a prototype of a high speed network device. The thirteen PCBs are evenly arranged with a 0.8 inches separation. All PCBs except for PCB 3 have a nearly identical component configuration.

The four heat sinks mounted on PCB 3 create additional flow restriction. The PCBs generate approximately 24 Watts of power each, totaling 300 Watts for the system. Air enters the system at the inlet located in front of and below the cards. The inlet air then passes over a 17 degree ramp and then through an EMI shield. The heated air is driven through the fans and into a plenum just above the fan tray. It is then forced out of a rearward facing exit consisting of another EMI shield and a rectangular duct. This configuration, while slightly unconventional, allows multiple units to be installed in a modular array such that most of the heated exit air will not be drawn into the inlet of an adjacent device. Design variables of this system include EMI shield selection, inlet and exit geometry, fan selection, and plenum geometry. The EMI shield selected was fabricated from standard perforated sheet having an open area of 51% and an orifice diameter of 0.31 inches.

III. Overview of the Analysis Methodology

The steps involved in the overall methodology, which utilizes FNM and CFD in a complementary manner, are discussed below:

Characterization of an Individual Passage Using CFD

Air flows through the passages formed by the successive PCBs. Each card has various electronic components and heat sinks mounted on them. The resistance for the flow through a card passage can be determined approximately by treating it as a channel of a rectangular cross-section with an appropriate wall-roughness factor. In the present study, for a more accurate characterization of the flow behavior of the individual passages, CFD analysis is used. The result of this analysis provides the variation of the pressure loss with the flow rate for each type of passage (with and without heat sinks).

System-Level Analysis of the Entire Cabinet Using FNM

The cabinet is represented as a network of components through which the air flows as it moves through the system. The flow characteristics of standard components such as screens, filters, and fans are available from handbooks or vendor data. This data, in combination with the impedance characteristics determined from the detailed CFD analysis, are used to represent the cabinet. Flow network analysis considers the interaction of the various flow impedances and the fans to determine the total induced flow and its distribution in all components through various parts of the system.

Detailed Predictions in the Card Array Using CFD

Network analysis predicts the flow rates and the inlet temperatures for individual card passages. This information is used to specify the boundary conditions in the CFD analysis of the critical passage(s) to predict the detailed flow distribution throughout the card passage and the flow rates through individual heat sinks. This information is necessary for the prediction of the temperatures of the electronic components. The present study focuses on the prediction of the flow distribution alone and a calculation of the temperature distribution is not attempted.

Figure 2 shows a schematic representation of the steps described above. The individual steps involved in the analysis are now described in detail.

IV. Characterization of Individual Passages Using CFD

Individual cards have a number of electronics components and heat sinks mounted on them. Therefore, the geometry of the passage formed between successive cards is complex. Further, because the flow enters the cabinet in a direction parallel to the width of the PCB card, velocity over the inlet cross-section of the card passage can be very non-uniform. Use of a taper in the bottom wall of the cabinet is included to reduce the nonuniformity. Presence of a screen at the bottom of the card passages further facilitates a uniformization of the inlet velocity. However, some nonuniformity may remain especially when the screen is not very restrictive or the taper is not properly designed. Therefore, CFD is used to predict the details of the flow in the region consisting of the bottom taper, the screen, and the card passage for determining its overall flow impedance.

An important feature of the CFD analysis is the treatment of the heat sinks. Due to the small gap between the fins in the extruded heat sinks, resolution of the flow in the interfin passages will require a very fine grid and make the CFD model of the passage computationally expensive. In the present study, the heat sink is treated as a volume with a distributed resistance. Further, this resistance is determined from a flow network model of the extruded heat sink. It involves representation of flow through the heat sink as multiple flow paths in parallel, with each path consisting of an area contraction upstream of the heat sink, a duct of rectangular cross-section corresponding to the interfin passage, and an area expansion downstream of the heat sink. A more detailed description of this treatment is presented in the study by Radmehr et al. [5].

The control-volume method of Patankar [6] is used to perform CFD analysis. A commercial program COMPACT [7] that incorporates this method is used for developing the CFD model. The computational domain consists of the portion of the tapered bottom passage of a width corresponding to the separation between the cards, the screen at inlet of the card passage, and the card passage itself. The boundary conditions for the analysis involve specification of the velocity over the inlet of the computational domain and use of an outflow boundary treatment over the exit. The impedance characteristics are determined by running the CFD model for a range of inlet velocities to determine the corresponding loss of total pressure across the domain. There are two types of card arrays in the cabinet – those with and without heat sinks. The resulting variation of the pressure drop with the flow rate is shown in Figure 3.

V. System-Level Flow Distribution Using FNM

The technique of Flow Network Modeling is used for the prediction of the flow distribution throughout the system. This technique involves a graphical representation of the paths followed by the fluid as it moves through the system. The various components (fans, screens, filters etc.) and flow paths (e.g. ducts) in the network are represented using overall characteristics that specify the variation of pressure drop and rate of heat transfer in the component with the flow rate through it. Solution of the conservation of mass and energy in combination with the use of the component characteristics (momentum equations) enable prediction of the flow and temperature distribution throughout the system. More details of the FNM technique are presented in the studies by Belady et al. [2] and Steinbrecher et al. [3]. In the present study, a commercial software package MacroFlow™ [8] which incorporates this technique is used for flow network analysis.

Figure 4 shows the flow network of the cabinet. The flow enters the system through an opening at the front of the system. The losses through this opening are represented using an inlet component that is exposed to the ambient pressure. The flow then moves in parallel streams through individual card passages after passing through the tapered reservoir and the EMI screen at the bottom of the PCB cards. The flow impedance of each card passage assembly (bottom taper, screen, and card passage), as determined in the CFD analysis, is represented using a flow resistance element. Note that the circular icons are junctions that denote the locations of a constant total pressure. Placement of the card assembly creates a bypass channel between the last card surface and the corresponding wall of the cabinet. This bypass channel is represented as a rectangular duct with a screen at the bottom. The flow through the cabinet is driven by two symmetrically placed fans at the top of the card assembly. Thus, the streams from each half of the card assembly meet upstream of the respective fans. The combined stream goes through a 90-degree turn and leaves the system through a screen exposed the ambient pressure. Note that the impedance characteristics of the open inlet, screened outlet, bends, and ducts available in MacroFlow are taken from handbooks by Idelchik [9] and Blevins [10]. The fan curve is specified from the data provided by the fan manufacturer. Each fan provides a maximum head of 0.24"of water and maximum flow rate of 45 CFM.

Analysis of the flow network of the cabinet provides information about the flow distribution throughout the system. Thus, the flow rates of the individual card passages and the extent of flow bypass are known. Similarly, a comparative evaluation of the pressure losses through the card cage, and the inlet and the exit regions can be readily carried out. Note that network analysis automatically accounts for the interaction between the flow impedances and the fans in determining the induced flow through the system. Thus, the fan operating points are predicted from the analysis. Finally, due to the modular nature of the network representation, effect of changes to the system such as a different placement of the fans or incorporation of a more open screen at the exit are readily carried out for efficient optimization of the thermal performance of the system.

VI. Results

The analysis approach proposed in the present study has been applied to predict the distribution of the flow throughout the system. Comparison between the predicted and the measured values of the flow rates and results of the CFD-based analysis of the flow distribution in individual card passages are now discussed.

Flow Distribution in the Cabinet and Its Comparison with Experiments

The bar chart in Figure 5 shows the volumetric flow rates through card passages. Note that Passage 1 is formed by the leftmost chassis sidewall and the back surface of PCB 1. It has the largest cross-sectional area and no obstructions on the walls. Therefore, the flow resistance of this passage is much lower than the other card passages and has the highest flow rate through it. The reduced flow rate in Passage 4 is largely due to the additional impedance caused by the presence of heat sinks on PCB 3. The flow rate is fairly constant in the center passages because they are virtually identical.

Detailed experimental measurements are carried out for determining the flow distribution in the cabinet. The flow rate for each passage is determined experimentally by taking velocity measurements at the inlet of each passage just downstream of EMI shield 1. Measurements are made through the depth of the passage every half-inch. These velocity measurements are used to determine the flow rate through each passage. The total flow through the cabinet is also measured over the inlet cross-section in a similar manner. A very good agreement is obtained between the computational and experimental total passage flow rates. The total flow rate measured at the inlet was 66.6 CFM with an uncertainty of 0.5 CFM. This agrees extremely well with the predicted value of 65.8 CFM. Figure 5 also shows the comparison between the computed and measured flow rates through the card passages. The maximum discrepancy between the predicted and the measured values is only 10%. The reasons for these discrepancies include minor variations in the dimensions of the components mounted on the PCBs and physical gaps (not represented in the model) between the PCBs and the cabinet walls.

Detailed CFD Analysis of the Flow in the Card Passages

The flow rates through individual card passages for the operating condition of the system are determined in the flow network analysis. CFD analysis can then be used to determine the detailed flow distribution in the individual passages. Figure 6 shows the predicted velocity vectors and the pressure field in a plane through a card passage for the actual operating condition. The blockage effect of the electronics components is clearly seen. It is also noteworthy that the heat sinks offer a large resistance to the flow due to the narrow interfin passages. Therefore, the air bypasses the heat sink and the actual flow through the heat sink is much lower than the incoming flow. For a heat sink with densely placed fins, such a flow bypass can significantly reduce the effectiveness of the heat sink and care needs to be exercised in the design and placement of heat sinks on the circuit boards. Finally, it is noteworthy that most of the pressure drop in the PCB assembly occurs in the screen mounted at the bottom of the passage because of its low fractional open area. Further, the large screen resistance also causes the flow velocity to be uniform over the entire inlet cross-section of the card passage. The tapering of the bottom wall is normally used to minimize the manifold effect and achieve a uniform velocity over the inlet of the card passages. However, such a taper is necessary only when the screen resistance is small (percentage open area >80%).

VII. Conclusions

The present study demonstrates a complementary use of the techniques of Computational Fluid Dynamics (CFD) and Flow Network Modeling (FNM) for efficient analysis of the flow distribution in a practical telecommunications cabinet. The physical system analyzed in the present study involves a forced-flow air-cooled cabinet containing a stack of PCB cards. CFD analysis is used to determine the impedance characteristics of the individual card passages for use in the network analysis of the entire system. Heat sinks are represented as volumes with a distributed resistance to derive computational efficiency. These distributed resistances are, in turn, determined using FNM analysis of a heat sink. A network model of the entire system is then constructed for the prediction of the flow distribution through the entire system. It accounts for the interaction between the various flow impedances and the fans to determine the actual operation of the system. Subsequent use of the CFD analysis of individual card passages enables prediction of the detailed flow distribution within individual card passages needed for determining the effectiveness of various heat sinks and a thermal map of the PCBs. A detailed measurement of the airflow velocities in various parts of the cabinet has been carried out. The predicted variation of the flow rates through the card passages and total flow through the system are within 10% of their experimentally measured values.

The hybrid approach proposed in this study is very general in nature and it can be utilized for analysis of any complex electronics cooling system. The complementary approach utilizes the strengths of individual methods to minimize the time required for a complete analysis of the flow system and to incorporate modularity of flow analysis that is very beneficial for its integrated use in the design process.

VII. References

[1] G.N Ellison., *Thermal Computations for Electronic Equipment*, Robert E. Krieger Publishing Co., Malibar, Florida, 1989.

[2] C. Belady, K.M. Kelkar, and S.V. Patankar, "Improving Productivity in Electronics Packaging with Flow Network Modeling (FNM)," *Electronics Cooling*, Vol. 5, No. 1, pp. 36-40, January 1999.

[3] R. Steinbrecher, A. Radmehr, K.M. Kelkar, and S.V.Patankar, "Use of Flow Network Modeling (FNM) for the Design of Air-Cooled Servers," *Advances in Electronics Packaging*, *Proceedings of the Pacific RIM/ASME Intersociety Electronics and Photonic Packaging Conference*, Vol. II, pp. 1999-2008, 1999.

[4] K.M. Kelkar., A. Radmehr, P. Kelly, S.V. Patankar, C. Belady, "Use of Flow Network Modeling (FNM) for enhancing the design process of Electronic Cooling Systems," in *Proc. International Systems Packaging Symposium (ISPS)*, San Diego, January 11-13, 1999, pp. 58-63.

[5] A. Radmehr, K.M. Kelkar, P. Kelly, S.V. Patankar, and S.S. Kang, "Analysis of the Effect of Bypass on the Performance of Heat Sinks Using Flow Network Modeling (FNM)", in *Proc. Semiconductor Thermal Measurement and Management Symposium (SEMI-THERM), San Diego, March 9-11 1999, pp*. 42-47.

[6] S.V. Patankar, *Numerical Heat Transfer and Fluid Flow*, Hemisphere, 1980.

[7] Compact User's Manual, Innovative Research, Inc., 3025 Harbor Lane N., Suite 300, Plymouth, MN 55447.

[8] MacroFlow User's Manual, Innovative Research, Inc., 3025 Harbor Lane N., Suite 300, Plymouth, MN 55447.

[9] I.E. Idelchik, *Handbook of Hydraulic Resistance*, CRC Press, Florida, 1994.

[10] R.D Blevins, *Fluid Dynamics Handbook*, Krieger Publishing Comp any, 1992.

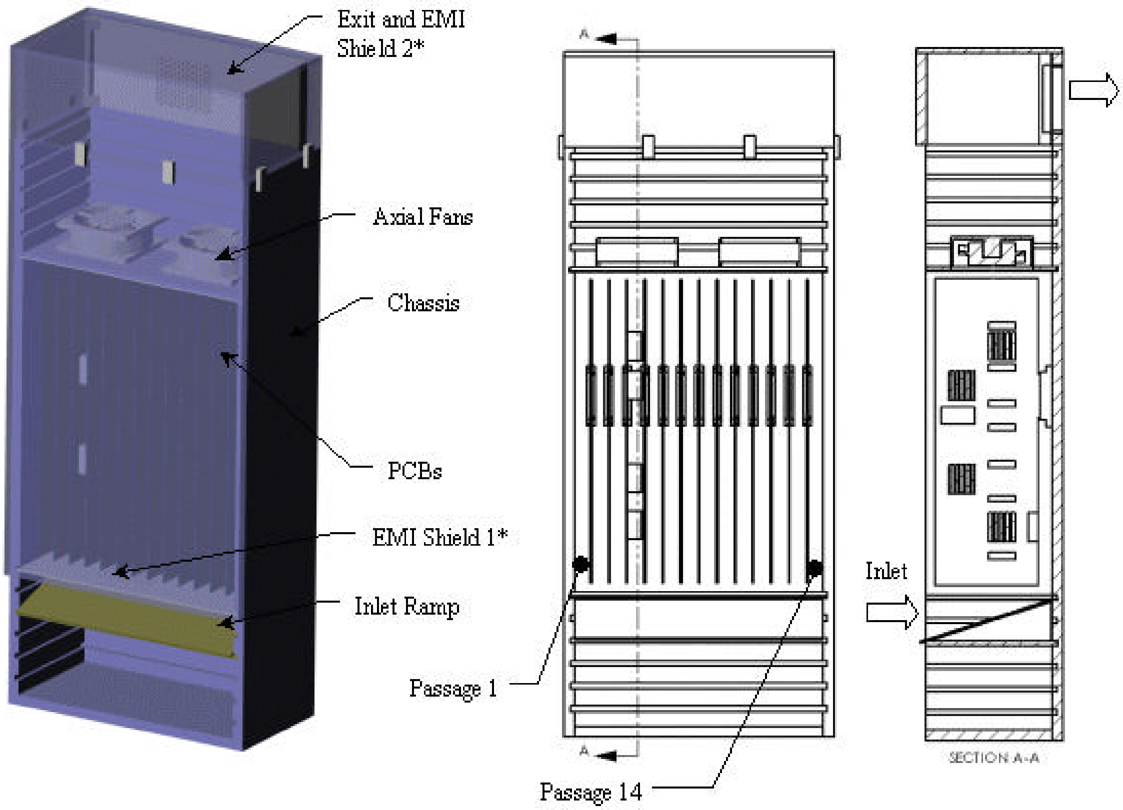

Figure 1 – Physical system analyzed in the present study

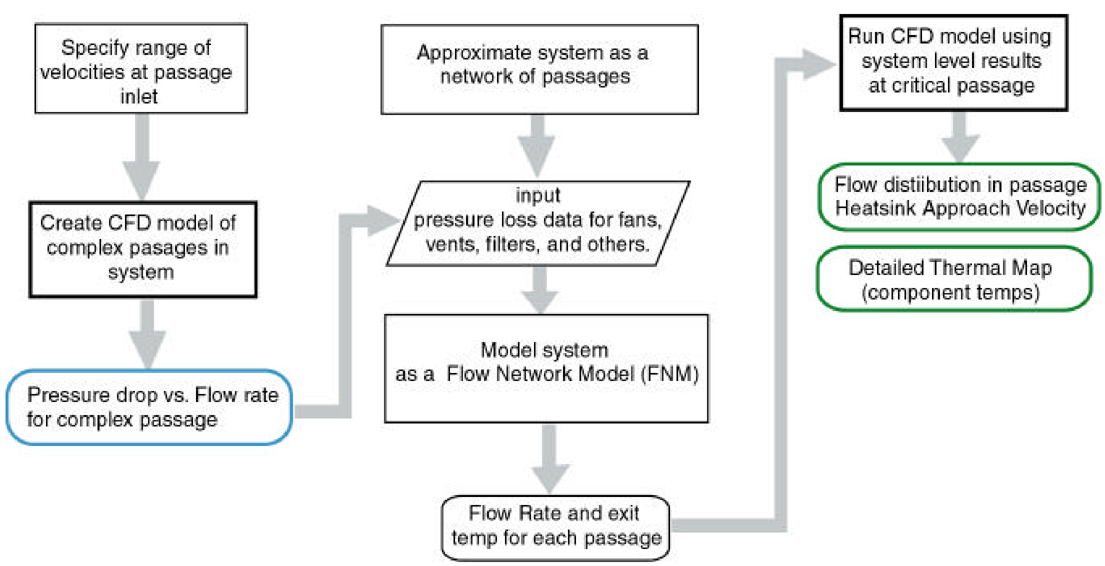

Figure 2 – Flow chart of the analysis technique used the

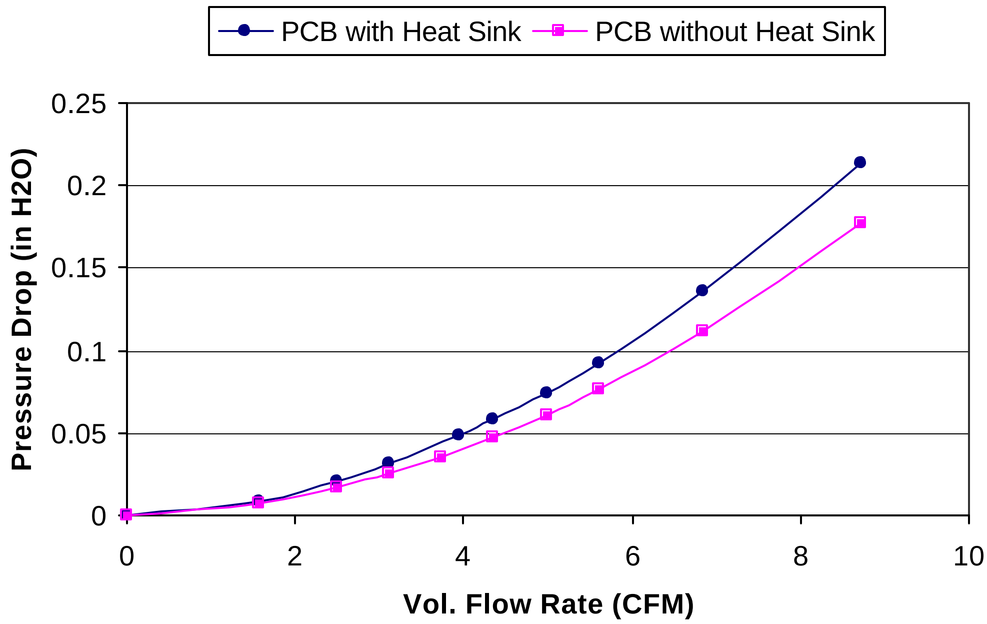

Figure 3 – Impedance characteristics of the card passages determined using CFD

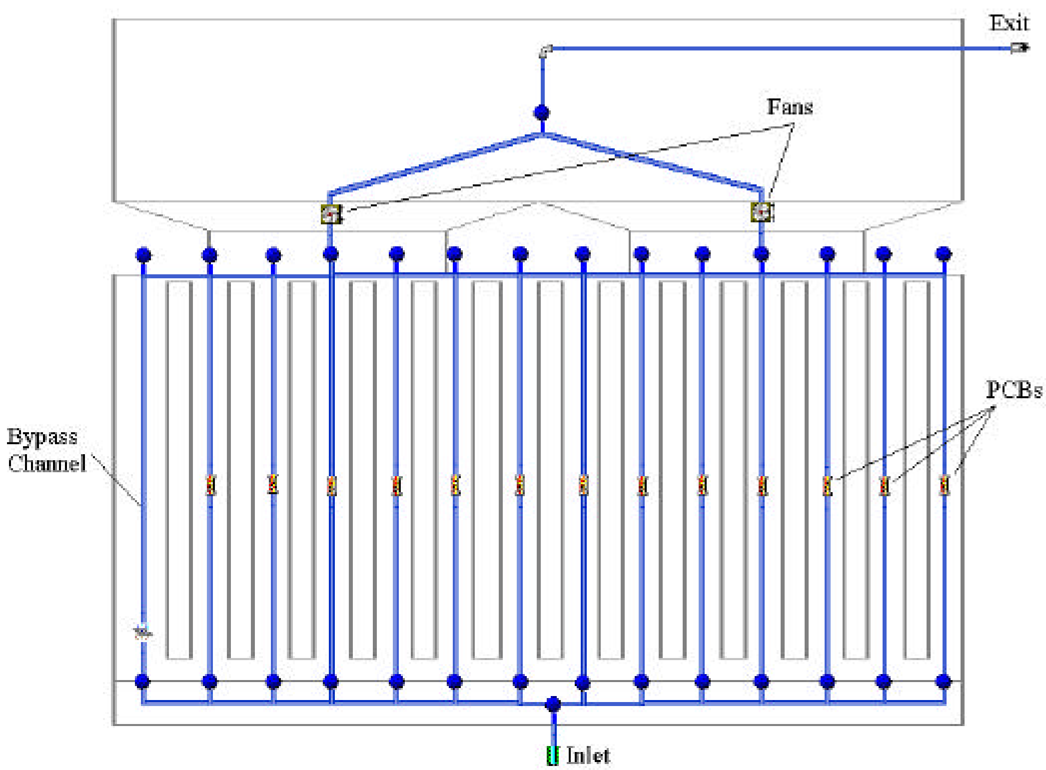

Figure 4 – Flow network representation of the flow through the cabinet

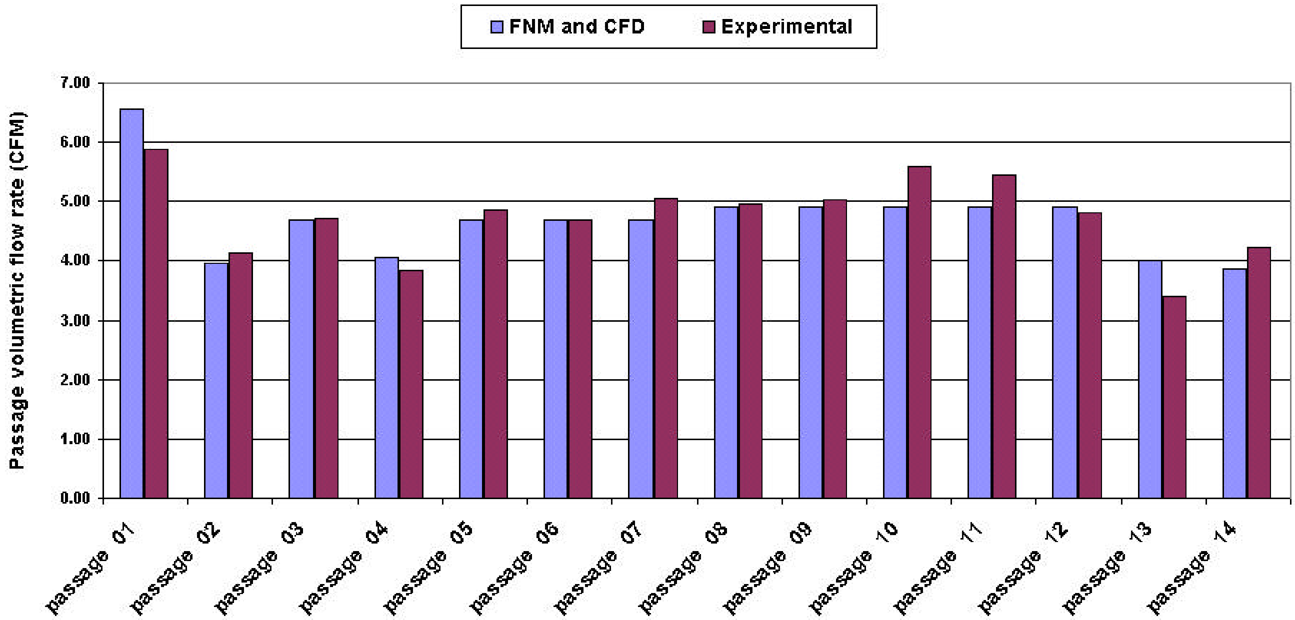

Figure 5 – Variation of the volumetric flow rates through the card passages and its comparison with experimental measurements

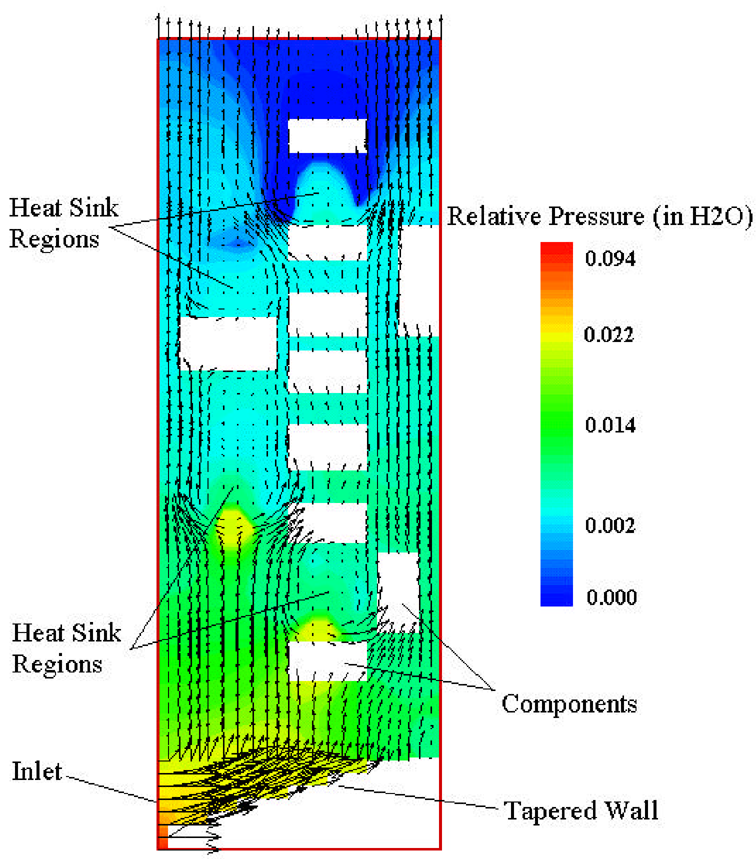

Figure 6 – Velocity vectors and the pressure contours in a plane through a card passage.