VOL. 5, NO. 1, JANUARY 2010 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

NUMERICAL INVESTIGATION OF NATURAL CONVECTION HEAT TRANSFER FROM MULTIPLE HEAT SOURCES IN A SQUARE ENCLOSURE

S. Venkatachalapathy and M. Udayakumar
Department of Mechanical Engineering, National Institute of Technology Tiruchirappalli, Tamil Nadu, India
E-Mail: svc@nitt.edu, venkatpjj@yahoo.co.in

ABSTRACT

Natural convection cooling using air as a fluid is commonly used in the cooling of electronic equipment and many other devices. A three-dimensional numerical study of natural convection heat transfer from multiple protruding heat sources simulating electronic components is conducted. Computational fluid dynamics (CFD) software, FLUENT is used in this analysis. A 4 by 5 array of heat sources are embedded in the bottom wall of an adiabatic square enclosure. The heat sources with a constant heat flux source at the bottom are of square cross-section and arranged in an in-line manner. Each heat source is attached with one thermocouple, which is connected to a data acquisition system and a computer. The steady state temperatures of heat sources, air inlet, outlet and enclosure walls are measured. The analysis is carried out by varying the heat fluxes and outlet areas. The heat transfer coefficient, Nusselt number and Grashof number are obtained. Results indicate that the heat sources inside the array are hotter and the heat transfer coefficient increases almost linearly with heat source surface temperatures. Grashof number and outlet opening areas strongly influence the Nusselt number. The heat transfer coefficient for the inner heat sources in a row is lower than those near the enclosure walls. The results of numerical analysis are compared with the experiments and there is a good agreement between the two.

Keywords: air cooling, numerical analysis, square enclosure, heat sources, Nusselt number.

Nomenclature

A	heat source surface area, m^2
C_p	specific heat at constant pressure, J/kg.K
g	acceleration due to gravity, m/s^2
Gr_H	Grashof number based on heat source height
Gr_L	Grashof number based on heat source length
H	heat source height, m
h	heat transfer coefficient, $W/m^2.K$
L	heat source length, m
Nu_H	Nusselt number based on heat source height
Nu_L	Nusselt number based on heat source length
Pr	Prandtl number
q	heat flux, W/m^2
T	temperature, °C
u, v, w	x, y and z components of velocity, m/s

Subscript

avg	average
b	bulk
in	inlet
wall	heat source wall

Greek symbols

α	thermal diffusivity, m^2/s
β	thermal expansion coefficient, K^{-1}
ν	kinematic viscosity, m^2/s
μ	dynamic viscosity, $N.s/m^2$
ρ	density, kg/m^3

INTRODUCTION

The satisfactory performance of electronic equipment depends on their operating temperature. In order to maintain these devices within the safe temperature limits, an effective cooling is needed. High heat transfer rate, compact in size and reliable operation are the challenges of a thermal design engineer of electronic equipment. Air cooling is suitable for low heat dissipating devices. Natural convection and forced convection are the two types of air cooling used. Natural convection is simple and does not require additional components to create the air flow and is widely used for cooling of low heat dissipating electronic and other equipment.

Experiments were performed by Salat *et al.*, [1] on turbulent natural convection in a cavity. Two guard cavities were used to ensure adiabatic conditions for the test cavity. The problem was also studied numerically with inputs from experiment, which greatly reduced the discrepancy between the experimental and numerical analysis. Temperature and velocity profiles were obtained for various planes in the cavity. Numerical studies were performed on the convection heat transfer characteristics of a heat conducting body in an enclosure by Hsu and How [2]. Constant heat flux was applied and the orientation and size of the heat source was varied. Isotherms and stream lines were obtained for various Richardson numbers and heat source positions.

Effect of inlet port locations, Reynolds number and Prandtl number on heat transfer characteristics of a cavity was numerically studied by Rahman *et al.*, [3]. The right vertical wall of the cavity was subjected to uniform heat flux. Streamlines and isotherms were obtained and a correlation was proposed for Nusselt number. Convection heat transfer from an 8 by 4 array of heat sources in a rectangular channel was experimentally investigated by Baskaya *et al.*, [4, 5]. The discrete heat sources were subjected to uniform heat flux. Row averaged Nusselt number and temperature were obtained. The Nu was

VOL. 5, NO. 1, JANUARY 2010 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

lowest for fifth row and maximum for first row. Bhowmik *et al.,* [6] studied numerically and experimentally steady state convective heat transfer from in-line chips with water as a fluid. Effect of flow velocity, chip numbers on heat transfer characteristics was investigated.

Alawadhi [7] investigated numerically the forced convection heat transfer from obstacles in a channel with heat generation in the developing region of flow. Three heat sources with equal spacing was analysed using Finite element technique. The results indicate that maximum temperature occurred at the heat source centre and heat transfer coefficient was higher for top walls than the side walls. Natural convection from an array of heat sources in a cavity was numerically analyzed and verified with experimental results by Heindel *et al*., [8]. Laminar, conjugate heat transfer was considered with water and FC 77 as fluids. Temperature of each heat source was obtained. Effect of Rayleigh number and substrate - fluid conductivity ratios on heat transfer characteristics were investigated.

Natural convection cooling of a 3 × 3 heater array in a rectangular enclosure was investigated numerically by Tou *et al*., [9]. Various liquids were used and the effect of Rayleigh number, enclosure aspect ratio and Prandtl number on the heat transfer characteristics was studied. Madhavan and Sastri [10] performed a numerical study on conjugate natural convection from protruding heat sources in an enclosure using the CFD software, FLUENT. The effect of Rayleigh number on Nusselt number was obtained and a correlation was proposed for Nusselt number. A numerical study on natural convection from discrete heat sources was performed by Yu and Joshi [11]. The heat sources were mounted on a substrate kept in a vented enclosure. Different vent locations and vent configurations were used and the combinations of vents on top and right wall gave the maximum cooling effect.

Bazylak *et al*., [12] conducted a numerical study on natural convection heat transfer from flush mounted heat sources in a rectangular enclosure. Uniform heat flux was applied to the heat sources and periodic boundary condition was used for the enclosure side walls. Effect of heat source length and spacing on heat transfer was investigated and optimized. A 3-Dimensional experimental and numerical study on forced convection heat transfer from PLCC package was conducted by Yousoff *et al*., [13]. CFD software, FLUENT was used for numerical analysis. Effect of chip spacing, input power and air velocity on heat transfer was investigated. A 2-Dimensional numerical study was conducted by Bilgen and Balkaya [14] on natural convection heat transfer from flush mounted heat sources in a rectangular enclosure with ventilation ports. The heat sources were subjected to uniform heat flux and an optimum heater position and spacing were arrived.

It was found from the available literature that not much work has been done about the influence of inlet and outlet opening areas on the heat transfer characteristics of multiple heat sources in an enclosure. Also, the heat transfer coefficient variations for individual heat sources with a specified heat flux and opening area was not discussed much.

PROBLEM DEFINITION AND EXPERIMENTAL PROCEDURE

The present work investigates numerically the natural convection heat transfer from multiple heat sources and the results are validated by experiments. The total number of heat sources is 20, arranged in an in-line manner with 5 in each row and thus there are 4 rows and 5 columns. The heat source number and the row or column of the array it belongs is given in Table-1. Aluminium is used as the material for heat sources due to its high thermal conductivity. This ensures uniform temperature for a heat source under steady state conditions. Each heat source has a cross-section of 20 mm × 20 mm and the surfaces are highly polished to minimize emissivity and thus radiation effects. The heat sources are mounted on the bottom wall of a square enclosure of side 350 mm and height 96 mm. The enclosure aspect ratio is 3.5. In order to have some realistic dimension for the enclosure, this size is chosen since this is the general dimension for the cabin of desk top PCs. The enclosure walls are made up of wood and the walls are assumed adiabatic. The arrangement of heat sources is shown in Figure-1.

Table-1. Heat source number and the row/column of the array it belongs.

Heat source number (row wise)		Heat source number (column wise)	
Row number	**Heat source number**	**Column number**	**Heat source number**
1	1-5	1	1, 6, 11, 16
2	6-10	2	2, 7, 12, 17
3	11-15	3	3, 8, 13, 18
4	16-20	4	4, 9, 14, 19
		5	5, 10, 15, 20

Figure-1. Arrangement of heat sources-4 rows and 5 columns.

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

Inlet is provided at the bottom of left vertical wall parallel to the heat sources row. The inlet has a fixed cross-section of width 310 mm and height 15 mm whereas the outlet area and location is varied. In the first case, outlet is located diagonally opposite to inlet and on top wall with an opening area of 310 × 15 mm^2 (case 1-rectangular open). In the second case, the outlet is in the form of ventilation ports of circular cross-section, with one port on top of each heat source. Thus there are 20 numbers of ventilation ports, each having a diameter of 10 mm. The heat sources are supplied with uniform heat flux by a heater. The back surface of the heater is insulated to minimize heat losses to the surroundings. A K-type thermocouple is attached with each heat source, 1.5 mm under its surface. A hole is drilled through the bottom of each heat source and the threaded thermocouple is inserted and tightened to ensure perfect contact with the heat source surface. Hence, there are 20 thermocouples attached to the heat sources and 4 more to monitor the air inlet, outlet and enclosure wall temperatures. The thermocouples are connected to a 60-channel Data Acquisition System and a computer. An auto transformer is used to supply power to the heater and the heater input is calculated by measuring the voltage and current. The experimental setup is shown in Figure-2.

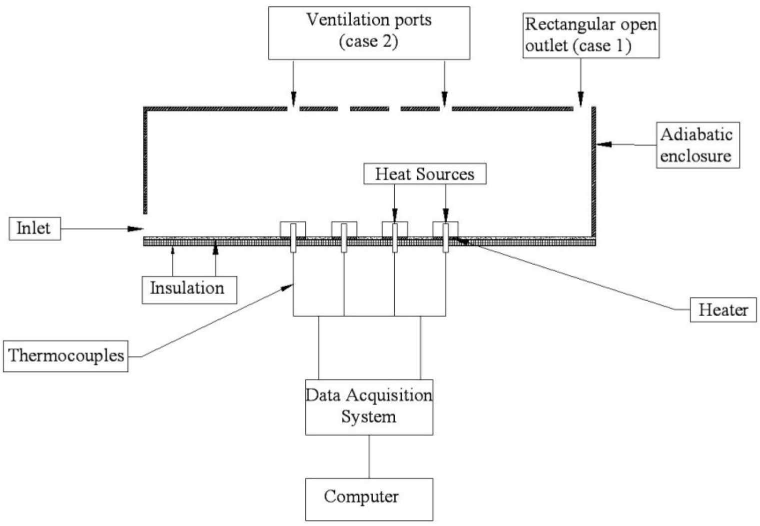

Figure-2. Experimental setup for the analysis.

This problem can be studied by two methods. In the first method, a prescribed heat flux is applied to the heat sources and the parameters like heat source temperature, heat transfer coefficient, Nusselt number, etc. are obtained. The second method which is used in this study maintains the heat sources at prescribed temperatures and finds the heat transfer rate and other parameters. This approach gives a better idea about the rate of heat dissipation from the heat sources at various surface temperatures.

In order to minimize the external disturbances in natural convection, the experimental setup is kept inside a large enclosed space. Uniform heat flux is applied to all the heat sources. The parameters varied in this study are air outlet opening areas, viz, rectangular open outlet (case 1), ventilation ports outlet (case 2) and applied heat fluxes. The heat fluxes are adjusted such that the average surface temperatures of heat sources are approximately 50°, 60°, 70° and 80°C in both the cases. The maximum temperature is restricted to 80°C as most of the electronic equipment operates below this temperature. The inlet, outlet, enclosure walls and heat source temperatures and heater power input are measured at every 30 seconds interval. Steady state conditions are obtained after 3 to 4 hrs of heating and the steady state readings are stored in the computer.

THE NUMERICAL MODEL

Governing equations

The flow is assumed as steady, incompressible and three-dimensional. Also, the conjugate heat transfer with conduction in the heat source is taken into account. Thermophysical properties of the fluid are assumed constant except density variations, which causes buoyancy force in the momentum equation and Boussinesq approximation is used. The governing equations of mass, momentum and energy can be written as

$$\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)\frac{\partial}{\partial z}(\rho w)=0 \tag{1}$$

$$\frac{\partial}{\partial x}(\rho uu)+\frac{\partial}{\partial y}(\rho vu)+\frac{\partial}{\partial z}(\rho wu)=\frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial u}{\partial z}\right) \tag{2}$$

$$\frac{\partial}{\partial x}(\rho uv)+\frac{\partial}{\partial y}(\rho vv)+\frac{\partial}{\partial z}(\rho wv)=\frac{\partial}{\partial x}\left(\mu\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial v}{\partial z}\right) \tag{3}$$

$$\frac{\partial}{\partial x}(\rho uw)+\frac{\partial}{\partial y}(\rho vw)+\frac{\partial}{\partial z}(\rho ww)=\frac{\partial}{\partial x}\left(\mu\frac{\partial w}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial w}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial w}{\partial z}\right)-\frac{\partial p}{\partial z}-\rho g \quad (4)$$

$$\frac{\partial}{\partial x}(\rho uT)+\frac{\partial}{\partial y}(\rho vT)+\frac{\partial}{\partial z}(\rho wT)=\frac{\partial}{\partial x}\left(\frac{\mu}{Pr}\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{\mu}{Pr}\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(\frac{\mu}{Pr}\frac{\partial T}{\partial z}\right) \quad (5)$$

The heat conduction equation for the heat source (solid region) reduces to

$$\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}=0 \quad (6)$$

All the walls of the enclosure are assumed adiabatic with no slip conditions.
Hence, for the enclosure walls,

$u = v = w = 0$

$q = 0$ and the normal temperature gradient for any wall is zero,

$$\frac{\partial T}{\partial x}=\frac{\partial T}{\partial y}=\frac{\partial T}{\partial z}=0 \quad (7)$$

All the thermophysical properties are evaluated at the bulk temperature of the fluid, defined as

$$T_b=\frac{T_{wall}+T_{in}}{2} \quad (8)$$

The Nusselt number is defined based on heat source length, Nu_L, and heat source height, Nu_H.

$$Nu_L=\frac{hL}{k}=\frac{qL}{kA(T_{wall}-T_{in})} \quad (9)$$

$$Nu_H=\frac{hH}{k}=\frac{qH}{kA(T_{wall}-T_{in})} \quad (10)$$

The Grashof number based on heat source length, Gr_L, and heat source height, Gr_H are respectively defined as

$$Gr_L=\frac{g\beta qL^4}{k\nu^2} \quad (11)$$

$$Gr_H=\frac{g\beta qH^4}{k\nu^2} \quad (12)$$

Computational procedure

Numerical solutions for the governing differential equations with the associated boundary conditions are obtained using CFD software, FLUENT. The equations are discretized using finite volume technique. Boussinesq approximation is used in the analysis. Pressure based solver is used with implicit scheme. Outlet temperature is obtained from the experiment and given as input for FLUENT. SIMPLE algorithm is used for pressure-velocity coupling. Second order upwind scheme is used for momentum and energy discretization. The convergence criterion for continuity, velocity components is 1×10^{-3} and for energy, it is 1×10^{-6}. When adiabatic conditions for the walls with back flow temperature equal to atmospheric temperature are used, there is a considerable difference between experimental and numerical values. This difference is minimized by using the data from experiments that is, enclosure wall temperature and air outlet temperature.

Grid independence test

A grid independence check is carried out to choose the proper grid size for the numerical analysis. The surface temperatures and heat transfer coefficients are compared for various grid sizes. It is found that a grid size of 140 × 140 × 38 for the enclosure is optimum. A finer grid is used for the heat sources.

RESULTS AND DISCUSSIONS

The surface temperature variations of heat sources with rectangular open outlet for various heat fluxes (W/m^2) are shown in Figures 3 and 4. The measured steady state temperatures are compared with those obtained by numerical analysis. The heat fluxes are chosen to get average surface temperatures of 50, 60, 70 and 80°C. The numerical results are close to experimental values with a maximum error of less than 10%. It is found that maximum temperature occurs in the third row and minimum in first row. All the heat sources in the first row are almost at the same temperature because this row is fully exposed to incoming fresh air. From second row onwards, the interior heat sources of a row are at higher temperatures than those near the enclosure walls because, these heat sources are surrounded on all sides by other heat sources and air motion is restricted.

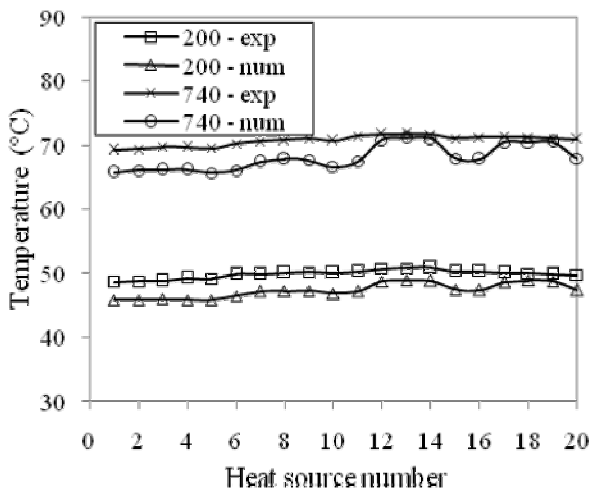

Figure-3. Steady state temperatures of heat sources for rectangular open outlet with applied heat fluxes in W/m^2 (experimental and numerical).

VOL. 5, NO. 1, JANUARY 2010 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

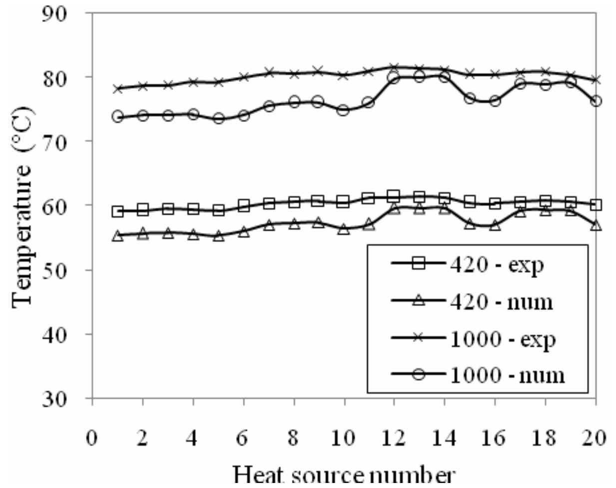

Figure-4. Steady state temperatures of heat sources for various heat fluxes (W/m^2)-rectangular open outlet.

Figures 5 and 6 shows the steady state surface temperatures of heat sources for ventilation ports outlet. There is a close agreement between experimental and numerical values. Third and fourth rows are almost at the same temperatures and the interior heat sources are at higher temperatures. As the first row is exposed to cooler fluid, its temperature is the lowest. The area of ventilation ports is one-third of rectangular open outlet. But the heat fluxes required for the same surface temperature is around 75 to 85% of rectangular open. In other words, the heat transferred with ventilation ports is 2 to 2.5 times more than rectangular open outlet for the same area. The reason for this can be explained as follows: the heated fluid raises due to buoyancy forces and most of this fluid escape through the ports which are located on top of each heat source thus minimizing the recirculation of hot air inside the enclosure, which may be the case in rectangular open outlet. Thus ventilation ports are preferred than a single opening of any shape for outlet.

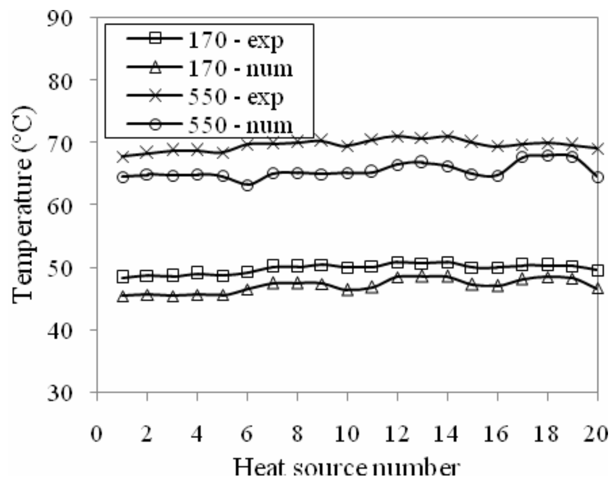

Figure-5. Steady state temperatures of heat sources for ventilation ports outlet with applied heat fluxes in W/m^2 (Experimental and Numerical).

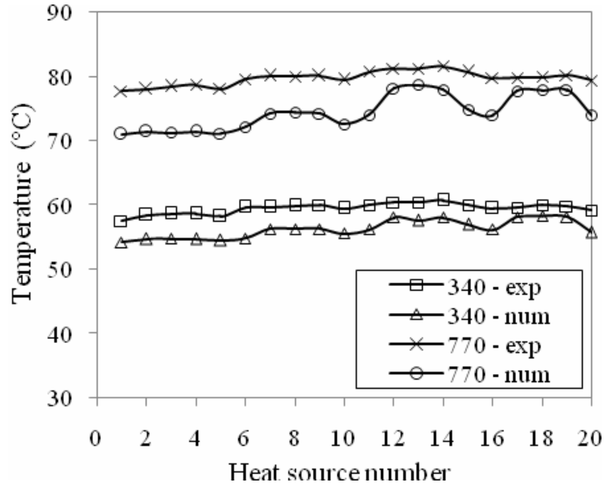

Figure-6. Comparison of steady state temperatures of heat sources for various heat fluxes-ventilation ports outlet.

The row-averaged surface temperatures of heat sources for both the cases are plotted in Figure-7. The applied heat flux is 500 W/m^2 and the numerical method slightly under predicts the temperatures. Maximum temperature occurs in third row and minimum temperature in first row. Column-averaged surface temperatures are plotted in Figure-8 for the same heat flux. Heat sources in columns 2, 3 and 4 are slightly at higher temperatures than those in columns 1 and 5. The average heat transfer coefficients based on experiments and numerical analysis are compared in Figure-9. The heat transfer coefficient increases linearly with surface temperatures in both the cases.

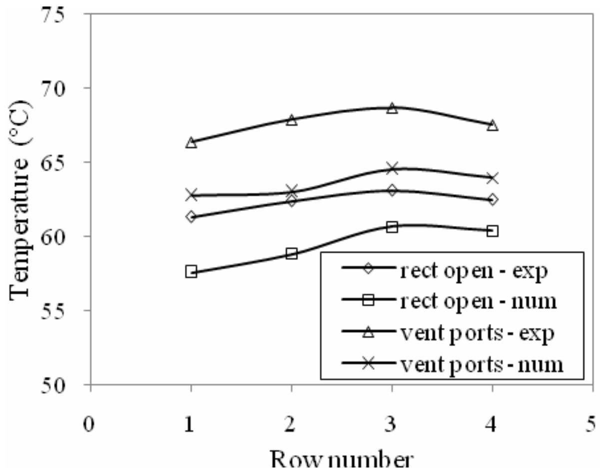

Figure-7. Row-averaged temperatures for both cases (Heat flux-500 W/m^2).

VOL. 5, NO. 1, JANUARY 2010 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

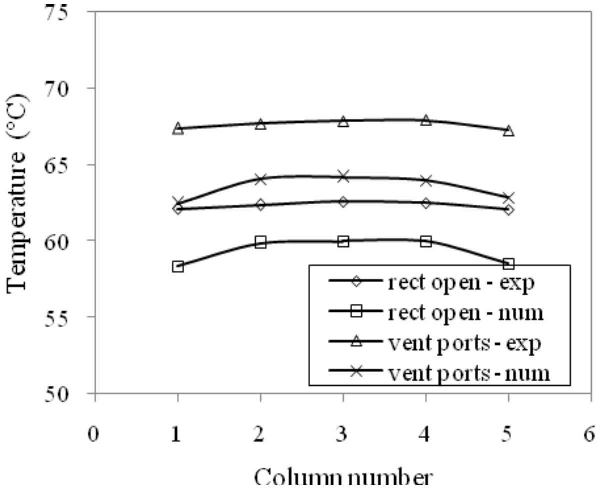

Figure-8. Column-averaged temperatures for both cases- Heats flux 500 W/m^2.

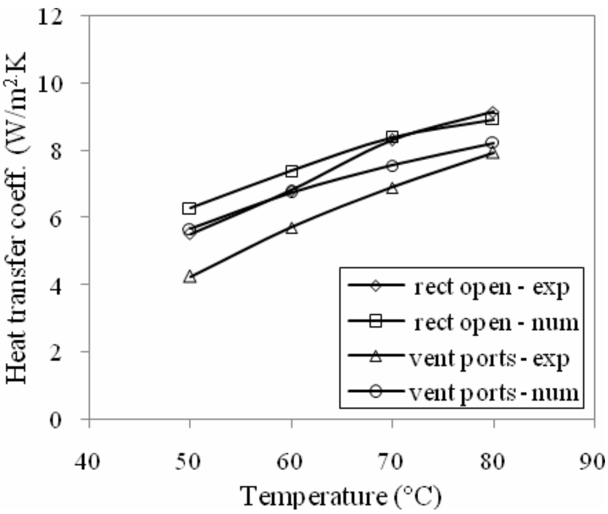

Figure-9. Average heat transfer coefficient for various surface temperatures.

Heat transfer coefficients for individual heat sources are compared for various heat fluxes in Figures-10 and 11. The heat fluxes in ascending order corresponds to heat source surface temperatures of 50, 60, 70 and 80°C in both the figures. Due to restriction in the air motion, the inner heat sources have lower heat transfer coefficients.

The nature of the curve remains same for all surface temperatures. Figures 12 and 13 show the variations of heat transfer coefficients with row number. Since the first row is exposed to fresh air and air motion is not restricted, heat transfer coefficient is higher while the minimum occurs in third row.

In the column averaged variations shown in Figures 14 and 15, the heat transfer coefficients for columns 1 and 5 are maximum because these columns are closer to the enclosure walls and free motion of air around these heat sources takes place. The heat sources in columns 2, 3 and 4 have almost same heat transfer coefficients. The influence of Grashof number on Nusselt number is shown in Figure-16. Both the numbers are calculated along the heat source length. The maximum Grashof number is around 5×10^5 and Nusselt number increases with Grashof number. The Nusselt number based on numerical analysis is greater than that of experimental value.

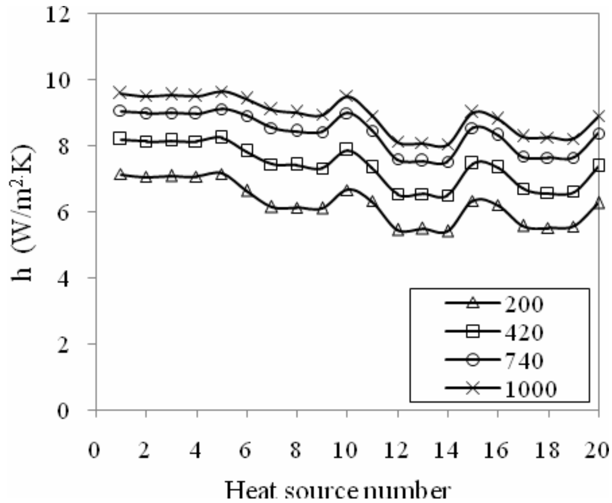

Figure-10. Heat transfer coefficient of individual heat sources based on numerical analysis for various heat fluxes- rectangular open outlet.

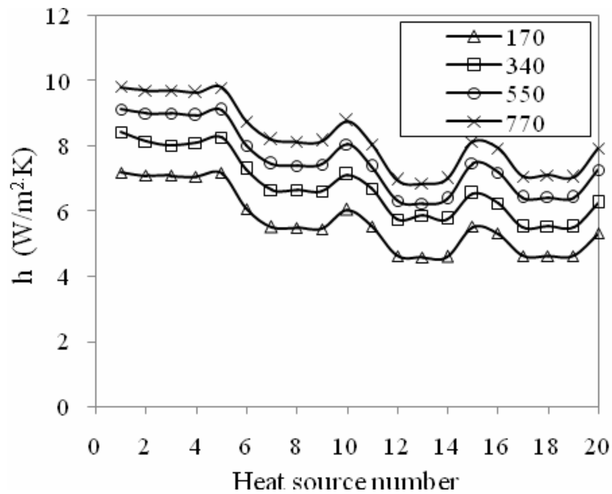

Figure-11. Numerical prediction of heat transfer coefficient of individual heat sources for various heat fluxes (W/m^2) - ventilation ports outlet.

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

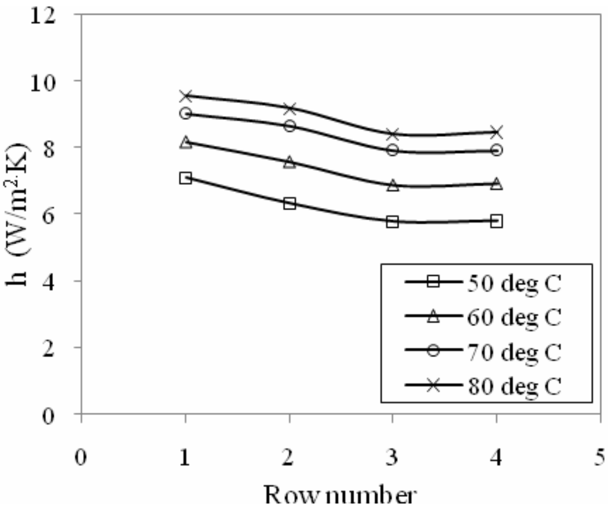

Figure-12. Row-averaged heat transfer coefficient for rectangular open outlet-numerical.

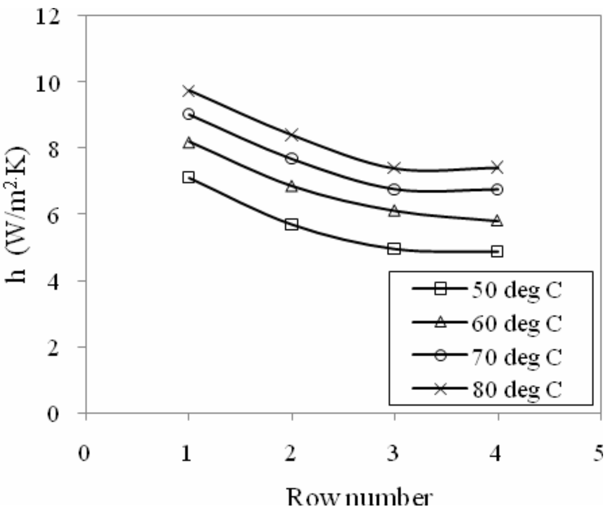

Figure-13. Row-averaged heat transfer coefficient for ventilation ports outlet-numerical.

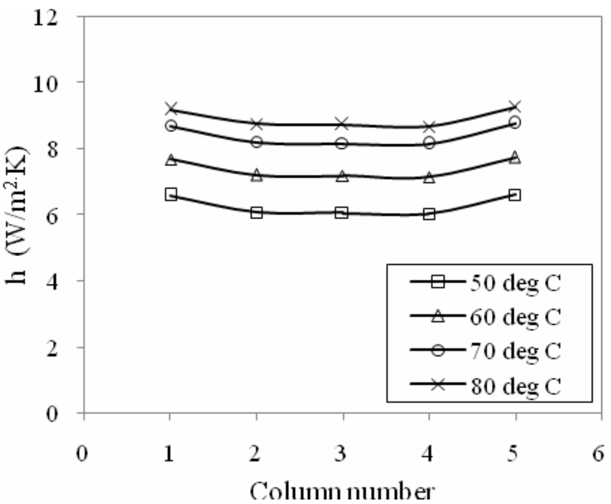

Figure-14. Column-averaged heat transfer coefficient for rectangular open outlet-numerical.

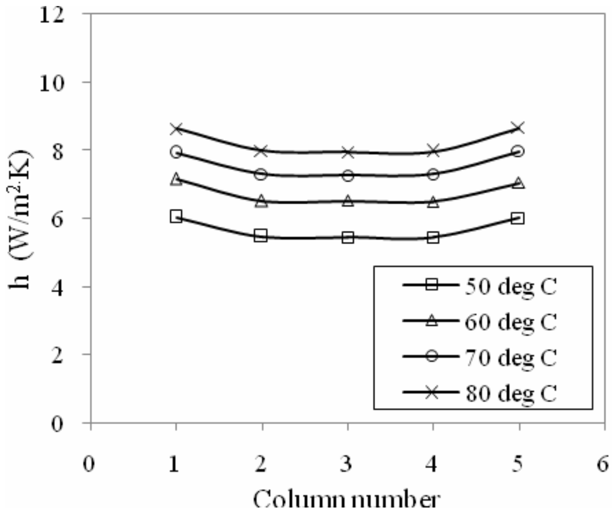

Figure 15. Column-averaged heat transfer coefficient for ventilation ports outlet-numerical.

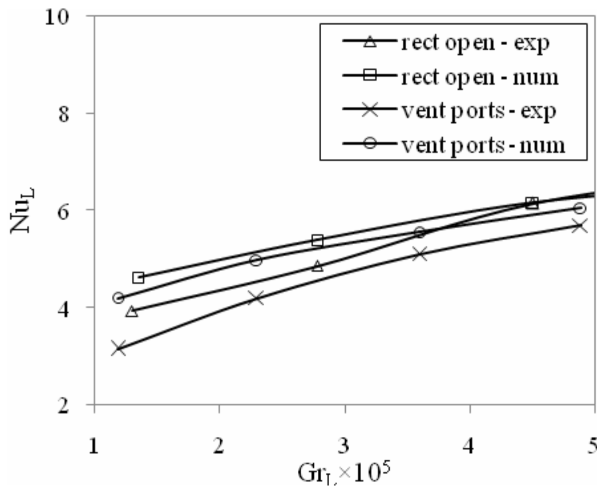

Figure-16. Effect of grashof number on nusselt number.

CONCLUSIONS

Natural convection heat transfer from a 4 by 5 array of heat sources has been investigated numerically and the results are compared with experiments. Uniform heat flux is applied to each heat source. The analysis is carried out with fixed inlet and two types of outlet namely, rectangular open and ventilation ports. The steady state temperatures are estimated and compared with experimental values. Heat transfer coefficients of individual heat sources, Grashof number and Nusselt number are obtained. There is a good agreement between the results obtained by numerical analysis and experimental data.

It is found that minimum temperature occurs in the first row and maximum temperature in third row. Ventilation ports enhance the heat transfer rate significantly. There is a considerable variation in temperatures between rows whereas it is almost the same in columns. Nusselt number varies linearly with Grashof number. The heat transfer coefficients for inner heat sources in any row are lower compared with those near the enclosure walls.

REFERENCES

[1] Salat J. *et al*. 2004. Experimental and numerical investigation of turbulent natural convection in a large air-filled cavity. International Journal of Heat and Fluid Flow. 25: 824-832.

[2] Hsu T.H. and How S.P.1999. Mixed convection in an enclosure with a heat conducting body. ACTA MECHANICA. 133: 87-104.

[3] Rahman M.M. *et al*. 2007. Numerical study of opposing mixed convection in a vented enclosure. ARPN Journal of Engineering and Applied Sciences. 2(2): 25-36.

[4] Baskaya S. *et al*. 2005. An experimental study on convection heat transfer from an array of discrete heat sources. International Communications in Heat and Mass Transfer. 32: 248-257.

[5] Baskaya S. *et al*. 2005. Experimental investigation of mixed convection from an array of discrete heat sources at the bottom of a horizontal channel. Heat Mass Transfer. 42: 56-63.

[6] Bhowmik H. *et al*. 2005. Analysis of convection heat transfer from discrete heat sources in a vertical rectangular channel. Journal of Electronic Packaging. 127: 215-222.

[7] Alawadhi E.M. 2004. Thermal analysis of a channel containing multiple heated obstacles with localized heat generation. IEEE Transactions on Components and Packaging Technologies. 27(2): 327-336.

[8] Heindel T.J. *et al*. 1995. Conjugate natural convection from an array of discrete heat sources: part 1-two-and three-dimensional model validation. Int. J. of Heat and Fluid Flow. 16: 501-510.

[9] Tou S.K. *et al*. 1999. 3-D Numerical analysis of natural convective liquid cooling of a 3×3 heater array in rectangular enclosures. International Journal of Heat and Mass Transfer. 42: 3231-3244.

[10] Madhavan P.N. and Sastri V.M.K. 2000. Conjugate natural convection cooling of protruding heat sources mounted on a substrate placed inside an enclosure: A parametric study. Computer Methods in Applied Mechanics and Engineering. 188: 187-202.

[11] Yu E. and Joshi Y. 1997. A numerical study of three-dimensional laminar natural convection in a vented enclosure. International Journal of Heat and Fluid Flow. 18: 600-612.

[12] Bazylak A. *et al*. 2006. Natural convection in an enclosure with distributed heat sources. Numerical Heat Transfer. Part A. 49: 655-667.

[13] Yousoff S. *et al*. 2009. 3-D Conjugate heat transfer analysis of PLCC package mounted in-line on a printed circuit board. International Communications in Heat and Mass Transfer. 36: 813-819.

[14] Bilgen E. and Balkaya A. 2008. Natural convection on discrete heaters in a square enclosure with ventilation ports. International Journal of Heat and Fluid Flow. 29: 1182-1189.

[15] Incropera F.P. and Dewitt D.P. 1996. Fundamentals of heat and mass transfer. John Wiley and Sons.

[16] Bejan A. 2004. Convection Heat Transfer. John Wiley and Sons.

[17] FLUENT Version 6.2. 2006. Computational fluid dynamics software.