Numerical and Experimental Study of Isotherm Field Inside a Desktop P.C.

Dr. Adel Hannon Ayaal
Electromechanical Engineering Department, University of Technology/Baghdad
Email: uot_magaz@yahoo.com

Abstract:

In this paper the heat transfer analysis from a heat generation components (central processing unit CPU and power supply) in a P.C. desktop was studied. Experimental and three − dimensional numerical study of laminar steady flow with mixed convection has been analyzed(for Ra=$8.3*10^5$). The Computational Fluid Dynamics (CFD) techniques are employed in numerical solution to calculate the temperature distribution inside the computer, by solving continuity, momentum and energy equations. This technique is based on structured grid (staggered grid) and finite volume formulation to discretize the governing equations (reduced to algebraic equations). The hybrid scheme and SIMPLE algorithm have been carried out to get the final solution. Five cases are studies for different heat source and inlet air flow locations. The temperature distribution and average temperature inside the computer are calculated to reach the optimum thermal design of the five cases. The numerical results obtained are found in a satisfactory agreement with the experimental results. Further the results confirmed that the most important parameter affecting the thermal behaviors is the position of heat-generation source and inlet air flow. The numerical solution shows that the case (4) is the optimum thermal design by comparison with other cases.

دراسة نظرية و عملية للمجال الحراري داخل الحاسوب الشخصي المنضدي

الخلاصة

خلال هذا البحث تم دراسة التحليل الحراري الثلاثي الأبعاد داخل فراغ الحاسوب الشخصي المنضدي بوجود مصدر حراري(وحدة المعالجه المركزية و مجهز القدرة). انجز هذا التحليل من خلال دراسة نظرية وعملية لجريان مستقر ومختلط الحمل (Ra=8.3*10^5). استخدمت تقنية ديناميك الموائع الحسابي لحل معادلات الاستمرارية ، الزخم والطاقة لأيجاد توزيع درجات الحرارة في داخل الحاسوب. بواسطة هذه التقنية وبالاعتماد على طريقة الحجوم المحددة(FVM) و staggered grid تم تحويل المعادلات الحاكمة للجريان الى معادلات خطية. استخدم نظام SIMPLE و hybrid scheme للحصول على الحل العددي النهائي. درست خمسة حالات تختلف عن بعضها باختلاف مواقع المصدر الحراري وفتحة دخول الهواء لأختيار التصميم الحراري الافضل. أوضحت النتائج ان النتائج النظرية كانت متقاربة مع النتائج العملية، علاوة على ذلك اوضحت النتائج ان موقع المصدر الحراري وفتحة دخول الهواء من اهم العوامل المؤثرة على السلوك الحراري داخل الحاسوب المنضدي الشخصي وكذلك اظهرت نتائج التحليل العددي ان الحالة رقم(4) هي افضل تصميم حراري من بين الحالات الخمسة.

Eng. & Tech. Journal , Vol.30, No.18, 2012

**Numerical and Experimental Study of
Isotherm Field Inside a Desktop P.C.**

Nomenclature

symbol	*Description*	Dimension
g	Acceleration due to gravity	m/s^2
p	Pressure	N/m^2
p$_r$	Prandtl number	
R$_a$	Rayleigh number	
T	Temperature	^oC
$T_{\exp}$	Experimental measured temperature	^oC
T_{ave}	Average Temperature of air inside the computer	^oC
u	Velocity component in x coordinate direction	m/s
v	Velocity component in y coordinate direction	m/s
w	Velocity component in z coordinate direction	m/s
b	Volumetric coefficient of expansion	K^{-1}
m	Dynamic viscosity of fluid	kg/m sec
r	Density of fluid	kg/m^3
Γ	General Diffusion coefficient	kg/m s

INTRODUCTION

In recent years, electronic devices have become indispensable part in every aspect of our daily life. In operating these devices, it is essential to maintain the temperature of electronic components below the recommended upper limit level to achieve optimum performance, maximum efficiency and reliability of the components. Inability in maintaining recommended temperature range will reduce the performance, efficiency and life span of the system and may even lead to catastrophic system failure [1].

Heat transfer problems with heat-generation components such as central processing unit (CPU), hard disk drive (HDD), and adaptive cards have been getting more and more significant since they are in our daily lives. The geometric configuration of heat sources and the cooling fans must be selected so as to generate an air flow rate which minimizes the average and temperature rise inside the enclosure itself [2].

Several works have been done for the cooling of electronic component located in an enclosure, Papanicolaou and Jaluria[3] simulated the laminar mixed convective cooling of electronic component located in an enclosure under conjugate heat transfer, Jayakanthan et al. [4] carried out a simulation under conjugate heat transfer using FLUENT (commercial CFD code) on single and two packages with a Printed Circuit Board (PCB) in wind tunnel for various flow conditions. Sattar[5] presented an experimental and computational study of three dimensional turbulent mixed convection for cooling of electronic components in enclosure. Heat transfer analysis with heat source in 3-D desktop P.C. was investigated by Chiang et al, [2] using CFD technique. Baïri et al, [6] presented numerical and experimental study to determine the thermal behavior in an enclosure, Yusoff et al, [7] presented a three dimensional heat and fluid flow analysis of two Plastic Leaded Chip Carrier (PLCC) packages mounted in tandem arrangement on a Printed Circuit Board (PCB) exposed to the free stream velocity. Jundika et al, [1] studied the heat transfer performance of various cooling channel designs; the cooling channel is designed to be placed on top of an electronic chip which dissipates heat at a constant flux. Laminar flow of a Newtonian fluid in a square cross-section channel is investigated using a three-dimensional computational fluid dynamic approach.

This paper presents a numerical and experimental study of isotherm field inside a P.C. desktop to get optimum thermal design of an electronic component located in an enclosure   . In the numerical solution the CFD technique was employed to simulate the thermal behavior inside a P.C. desktop.

EXPERIMENTAL WORK

To achieve the experimental work, a test rig is developed as shown in Figure (1). The experimental system includes a desktop P. C, several thermocouples (type K), digital thermometer (type K) and Anemometer (type AM-4206M). In order to measure the temperature distribution inside a desktop P. C computer, nine thermocouples are mounted in the space of computer. This thermocouples are distributed as net (3*3) inside the computer (which have the dimensions of (17.5 * 42 * 37.5 cm)) as shown in Figure (2), then connected to thermometer device to measure the temperature in this positions. Table (2) illustrated the locations for these thermocouples. Additional thermocouples are fitted on the surface of CPU at its center to obtain its maximum temperature, inside the power supply and near the body of computer to measure the temperature of environment.

(a)   (b)

Figure. (1) photographs of the testing system.

For more accurate measurements thermocouple wire is calibrated with thermometer for a different measurement. The measurement reading is a good agreement with error not exceeded to (0.5^{o} C). To ensure that the temperature inside the computer will be in steady-state condition, some points are chosen (all points of thermocouples and CPU) to measure its temperature value during the time. It is found that the temperature of these points will reach to the steady-state condition after (15 min), so in the experimental work a delay of (15 min) is needed before record the experimental data.

Before starting to record data, two tests are made. The first test is to measure the temperature of environment near the computer body. The second test is to measure the velocity of air flow at outlet unit position (x=11cm , y=37 , z=37.5).
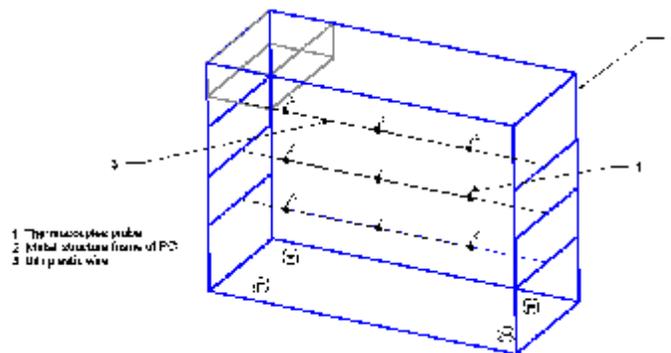
Figure (2) Schematic of the thermocouples arrangement inside a computer.

MATHEMATICAL FORMULATION

The objective of present study is to present steady incompressible, three-dimensional laminar mixed convection flow inside the desktop P.C. The governing equations describing the fluid flow are continuity, momentum and energy equations as follows [8] [9].

- Continuity equation:

$$\frac{\partial(ru)}{\partial x} + \frac{\partial(rv)}{\partial y} + \frac{\partial(rw)}{\partial z} = 0$$

- Momentum equation:

x- momentum equation

$$\frac{\partial}{\partial x}\,ruu + \frac{\partial}{\partial y}\,ruv + \frac{\partial}{\partial z}\,ruw = \frac{\partial}{\partial x}(m\frac{\partial u}{\partial x}) + \frac{\partial}{\partial y}(m\frac{\partial u}{\partial y}) + \frac{\partial}{\partial z}(m\frac{\partial u}{\partial z}) + S_u$$

y- momentum equation

$$\frac{\partial}{\partial x}\,ruv + \frac{\partial}{\partial y}\,rvv + \frac{\partial}{\partial z}\,rvw = \frac{\partial}{\partial x}(m\frac{\partial v}{\partial x}) + \frac{\partial}{\partial y}(m\frac{\partial v}{\partial y}) + \frac{\partial}{\partial z}(m\frac{\partial v}{\partial z}) + S_v$$

z- momentum equation

$$\frac{\partial}{\partial x}\,ruw + \frac{\partial}{\partial y}\,rvw + \frac{\partial}{\partial z}\,rww = \frac{\partial}{\partial x}(m\frac{\partial w}{\partial x}) + \frac{\partial}{\partial y}(m\frac{\partial w}{\partial y}) + \frac{\partial}{\partial z}(m\frac{\partial w}{\partial z}) + S_w$$

- Thermal energy equation :

$$\frac{\partial}{\partial x}\,ruT + \frac{\partial}{\partial y}\,rvT + \frac{\partial}{\partial z}\,rwT = \frac{\partial}{\partial x}(\Gamma\frac{\partial T}{\partial x}) + \frac{\partial}{\partial y}(\Gamma\frac{\partial T}{\partial y}) + \frac{\partial}{\partial z}(\Gamma\frac{\partial T}{\partial z}) + S_T$$

The values of S_u, S_v, S_w and S_T in the previous equations obtained as follows:

For laminar flow Ra $\leq 10^6$

$$S_u = -\frac{\partial P}{\partial x}$$

$$S_v = -\frac{\partial P}{\partial y} + rgb(T - Tc)$$

$$S_w = -\frac{\partial P}{\partial z}$$

$$S_T = \frac{Q_j}{C_p} = 0$$

Where $\Gamma = \dfrac{m}{\text{Pr}}$

NUMERICAL SOLUTION

In this study the numerical solution is achieved using CFD stechnique to simulate the thermal behavior of 3-D laminar mixed convection fluid flow inside the desktop P.C.

The transport equations for continuity, momentum, and energy which are mentioned in (Mathematical Formulation), all have the general form [10].

$$\frac{\partial(ruf)}{\partial x} + \frac{\partial(rvf)}{\partial y} + \frac{\partial(rwf)}{\partial z} = \frac{\partial}{\partial x}\left[\Gamma_f\left(\frac{\partial f}{\partial x}\right)\right] + \frac{\partial}{\partial y}\left[\Gamma_f\left(\frac{\partial f}{\partial y}\right)\right] + \frac{\partial}{\partial z}\left[\Gamma_f\left(\frac{\partial f}{\partial z}\right)\right] + S_f$$

where f is called the dependent variable while Γ_f and S_f are the diffusion coefficient and the source term, respectively.

Table (1): Source terms in the transport equations.

Equation	f	Γ_f	S_f
Continuity	1	0	0
x-momentum	u	m	$-\dfrac{\partial p}{\partial x}$
y-momentum	v	m	$-\dfrac{\partial p}{\partial y} + rgb(T - T_o)$
z- momentum	w	m	$-\dfrac{\partial p}{\partial z}$
Energy	T	m/Pr	0

The finite volume method (FVM) with staggered grid arrangement is adopted to get a numerical solution for a governing differential equation by discretized the equations (reduced to algebraic equations). The discretization equation for three dimensional based on the general differential equation is:-

$$a_P f_P = \sum_{nb} a_{nb} f_{nb} + S_f$$

where :

$$a_P = \sum_{nb} a_{nb} \ ,$$

$$a_E = \max\left[\left|0.5F_e\right|, D_e\right] - 0.5F_e$$

$$a_W = \max\left[\left|0.5F_w\right|, D_w\right] + 0.5F_w$$

$$a_N = \max\left[\left|0.5F_n\right|, D_n\right] - 0.5F_n$$

$$a_S = \max\left[\left|0.5F_s\right|, D_s\right] + 0.5F_s$$

$$a_T = \max\left[\left|0.5F_t\right|, D_t\right] - 0.5F_t$$

$$a_B = \max\left[\left|0.5F_b\right|, D_b\right] + 0.5F_b$$

where (*F* and *D*) represent the strength of the convection and diffusion term for the control volume respectively, which are equal to:

3169

$$F_e = (ru)_e A_e \quad , \quad F_w = (ru)_w A_w \quad , \quad F_n = (rv)_n A_n \quad ,$$

$$F_s = (rv)_s A_s \quad , \quad F_t = (rw)_t A_t \quad , \quad F_b = (rw)_b A_b \quad ,$$

$$D_e = \frac{\Gamma_e}{dx_{EP}} A_e \quad , \quad D_w = \frac{\Gamma_w}{dx_{PW}} A_w \quad , \quad D_n = \frac{\Gamma_n}{dy_{NP}} A_n \quad ,$$

$$D_s = \frac{\Gamma_s}{dy_{PS}} A_s \quad , \quad D_t = \frac{\Gamma_t}{dz_{TP}} A_T \quad , \quad D_b = \frac{\Gamma_b}{dz_{PB}} A_b$$

The coefficients (a_n) are derived using a hybrid scheme of Patankar [11] .

The solution of the momentum equations, which involves the velocity components (u), (v) and (w), can be obtained if the pressure field is given or estimated. SIMPLE (semi implicit method for pressure-linked equations) is one of most wildly used methods which link the velocity to the pressure in order to satisfy continuity equation procedure of Patankar and Spalding [10].

The final computations are carried out for a grid nodes (21*31*21) and the measured temperatures of central processing unit(CPU), power supply unit and environment were (37.2 ^{o} C), (34.7 ^{o} C) and (30 ^{o} C) respectively. While the velocity of exit air from computer was (2.93 m/s) with flow rate (138cm ^{3} /s).

RESULTS AND DISCUSSION:

The thermal behaviors of a ventilated desktop P.C. depends on geometric confirmation of heat source (CPU and power supply), air outlet and inlet to unit. In this study five cases are studied which different in position of CPU and inlet air flow location. The average temperature is presented for all cases to choice the optimum design.

The numerical results shows in a good agreement with the experimental results as indicated in Table (2). The simulation results of different location of CPU and inlet air flow are shown in Table (3). Since, the results show that the heat transfer efficiency of case (4) is better than the other cases due to the directly dissipative heat by forcing inlet flow on the vicinity of heat- generation component (CPU). Figure (3) illustrates the temperature isotherm contours of the flow field for case (1). It is evident that the local high temperature area is located in the vicinity of heat-generation component (CPU) and show that the temperature distribution inside the computer is very close due to the small space of computer.

One of the important factors that determine the cooling performance is the geometry of flow field as it is directly linked to the velocity and temperature profiles inside the computer. In general the flow velocity results in convective heat transfer; thus, more heat from the electronic equipments can be dissipated so as to maintain a more uniform temperature distribution within allowable limits.

In order to analyze the thermal behavior within a desktop P.C., and in particular, to obtain the character of fluid flowing several test were performed on the five cases. For the different positions of CPU and inlet air flow, the flow patterns are similar to each other because all cases included the same inlet air flow velocity as shown in Figure (4).

Table (2): comparison between numerical and experimental results.

Node No.	Node dimension (thermocouples location) (cm)			$T_{Num.}$ (^{o}C)	T_{Exp} (^{o}C)
1	x=8.75	y=31	z=8.75	35.4	34
2	x=8.75	y=31	z=18.75	35.4	34.5
3	x=8.75	y=31	z=28.75	34.9	35.5
4	x=8.75	y=21	z=8.75	35.7	34.2
5	x=8.75	y=21	z=18.75	35.7	36.7
6	x=8.75	y=21	z=28.75	35.5	35.5
7	x=8.75	y=11	z=8.75	35.7	35.5
8	x=8.75	y=11	z=18.75	35.7	35.4
9	x=8.75	y=11	z=28.75	35.7	36.4

Table(3): Average temperature for different positions of CPU and inlet flow.

Case No.	Inlet flow position(cm)	CPU position (cm)			T_{ave} (^{o}C)
1	z=37.5	x=16	y=27.5	z=28	35.61
2	z=37.5	x=7	y=27.5	z=26.8	36.42
3	z=37.5	x=16	y=28.5	z=28	36.21
4	z=37.5	x=16	y=26.5	z=28	35.23
5	z=0	x=16	y=27.5	z=28	36.66

Eng. & Tech. Journal , Vol.30, No.18, 2012

**Numerical and Experimental Study of
Isotherm Field Inside a Desktop P.C.**

a) 3-D inside the space

b) z=18 cm

c) y=30 cm

d) z=18 cm
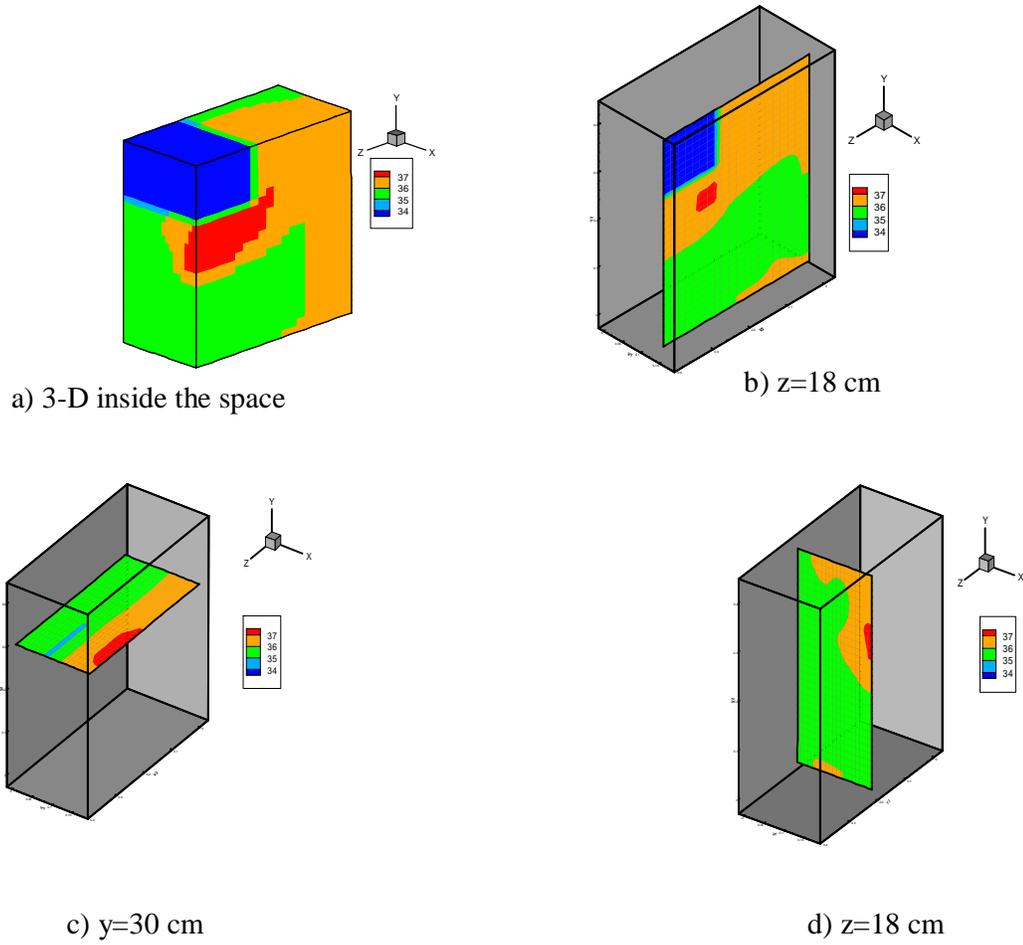
Figure.(3) Isotherm contours inside the computer for case (1).

a) 3-D inside the space   b) x=5cm

c) y=25cm

d) z=18cm
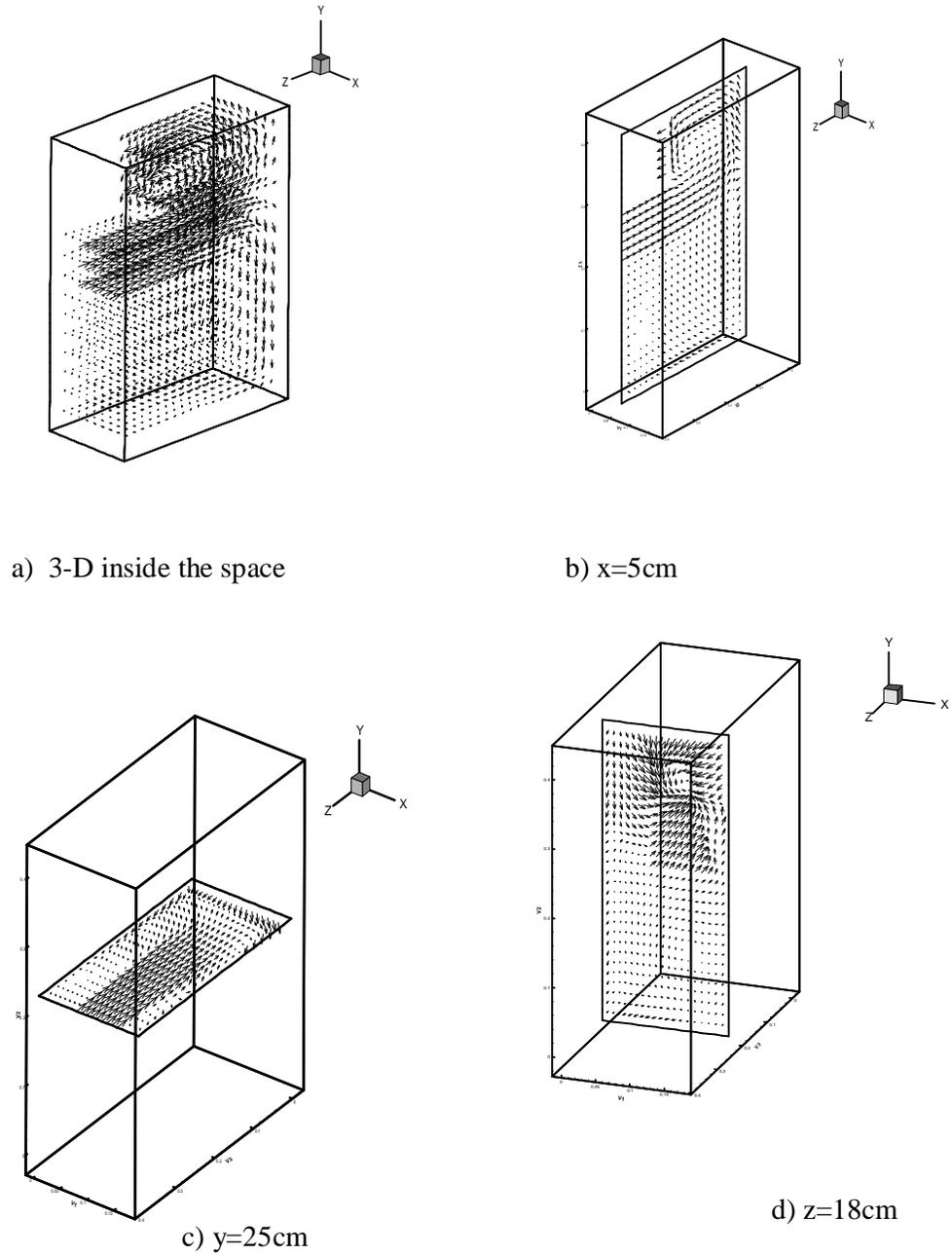
Figure.(4) Flow field inside the computer for case (1).

3173

CONCLUSIONS

To minimize the temperature inside the P.C. a desktop the location of heat source (CPU and power supply) and cooling units (inlet and outlet airflow) must be selected so as to generate an air flow. The configuration of case (4) causes the best cooling condition inside the computer. The configuration of practical case (case-1-) have a good design.

REFERENCES

[1] Jundika C. Kurnia , Agus P. Sasmito and Arun S. Mujumdar , "Numerical investigation of laminar heat transfer performance of various cooling channel designs" Journal (Applied Thermal Engineering) vol.31, pp.(1293-1304), (2011).

[2] J.S. Chiang, S.H. Chuang, Y.K. Wu and H.J. Lee, "Numerical simulation of heat transfer in a desktop computer with heat-generating components" International communication in heat and mass transfer, vol.32, pp.(184-191), (2005).

[3] E. Papanicolaou and Y. Jaluria," Mixed convection from simulated electronic components at varying relative position in a cavity" ASME Journal of Heat Transfer, vol.116,pp.(960–970),(1994).

[4] A. Jayakanthan, A.Y. Hassan and K.N. Seetharamu, "Application of CFD in cooling electronic packages", The Seventh Asian Congress of Fluid Mechanics, pp. (777–780) December 1997.

[5] Sattar J. Habeeb, "Experimental and Computational Study of Three Dimensional Turbulent Mixed Convection for Cooling of Electronic Components in Enclosure" Ph. D Thesis , Submitted to Mech. Eng. Dep., University of Technology, Iraq (2003).

[6] A. Baïri, J.M. Garc, N. Laraqi and N. Alilat "Free convection generated in an enclosure by alternate heated bands. Experimental and numerical study adapted to electronics thermal control "International Journal of Heat and Fluid Flow,vol. 29, pp.(1337–1346),(2008).

[7] S. Yusoff, M. Mohamed, K.A. Ahmad, M.Z. Abdullah, M.A. Mujeebu, Z. Mohd Ali, F. Idrus and Y. Yaakob " 3-D conjugate heat transfer analysis of PLCC packages mounted in-line on a Printed Circuit Board" International Communications in Heat and Mass Transfer,vol. 36, pp. (813–819) (2009)

[8] W. M. Rohsenow, J.P. Hartnett and E.N. Ganic, "Handbook of Heat Transfer Fundamentals" McGraw-Hill, Inc. USA, 1985.

[9] F. Chorlton, "Textbook of Fluid Dynamics" CBS Publishers & Distributors, India, 1985.

[10] Patankar, S.V. and Spalding D. B., "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows", International Journal of Heat and Mass Transfer, Vol. 15, pp. 1787-1806, 1972.

[11] Patankar, SV., "Numerical Heat Transfer and Fluid Flow", McGraw-Hill, New York (1980).