INTERNATIONAL JOURNAL OF
ARTIFICIAL INTELLIGENCE AND EMERGING TECHNOLOGY

Print ISSN 2735-4792

VOLUME 5, ISSUE 2, 2022, 49 – 62.

Online ISSN 2735-4806

CFD MODELING OF THE EFFECT OF THE AIR-COOLING ON ELECTRONIC HEAT SOURCES

A.E. KABEEL
MECHANICAL ENGINEERING DEPARTMENT-TANTA UNIVERSITY

A. M. KHALIL
MECHANICAL ENGINEERING DEPARTMENT-TANTA UNIVERSITY

G. I. SULTAN
MECHANICAL ENGINEERING DEPARTMENT – QASSIM UNIVERSITY

M. I. El-Hadary
MECHANICAL ENGINEERING DEPARTMENT-TANTA UNIVERSITY

ABSTRACT:

The target of this research is to study the performance of air cooling of an electronic cabinet including a heat sources (electronic circuit boards) by using axial fan. The cabinet is cooled from the top by one port while the lower and side walls temperature is kept constant. The effect of inlet and outlet air ports positions in the cabinet is considered. In this work, ***Icepak4.2.8*** package is used in the numerical study. The steady of the 3D incompressible viscous flow problem is solved by using the Icepak package. Various air-cooling geometries are applied. Specific conditions for each case are defined, and the computational fluid dynamics is provided for three different groups containing six cases of local inlet and outlet ports. The results are performed for cooling effect factor (CEF).

Print ISSN 2735-4792
VOLUME 5, ISSUE 2, 2022, 49 – 62.
Online ISSN 2735-4806

KEYWORDS:
Air cooling, electronic circuit, CFD, Icepak.

INTRODUCTION

The design of cooling systems for electronic equipment is getting very involved and challenging due to an increase in demand for faster and more reliable electronic systems. Therefore, robust and more efficient design and optimization methodologies are required. Natural convection heat transfer is an important phenomenon in engineering systems due to its wide application in electronics cooling, heat exchangers, and double pane windows. Enhancement of heat transfer in these systems is essential from the industrial and energy saving perspectives. The low thermal conductivity of conventional heat transfer fluids, such as water puts a primary limitation on the performance and the compactness of thermal systems. As a result, different cooling technologies have been developed to efficiently remove the heat from these components. The use of a liquid coolant has become attractive due to the higher heat transfer coefficient achieved as compared to air-cooling. Coolants are used in both single phase and two-phase applications. A single-phase cooling loop consists of a pump, a heat exchanger (cold plate/mini- or microchannels), and a heat sink (radiator with a fan or a liquid-to-liquid heat exchanger with chilled water cooling). The heat source in the electronics system is attached to the heat exchanger. Liquid coolants are also used in two-phase systems, such as heat pipes, thermo-siphons, sub-cooled boiling, spray cooling, and direct immersion systems for cooling of electronics [1].

The rapid development in the design of electronic packages for modern high-speed computers has led to the demand for new and reliable methods of chip cooling. As stated by Mahalingam and Berg [2], the averaged dissipating heat flux can be up to 25 W/cm^2 for high-speed electronic components.

However, the conventional natural or forced convection cooling methods are only capable of removing small heat fluxes per unit temperature difference, about 0.001 W/cm^2.^{o}C by natural convection to air, 0.01 W/cm^2.^{o}C by forced convection to air, and 0.1 W/cm^2.^{o}C by forced convection to single-phase liquid [3].

Print ISSN 2735-4792
VOLUME 5, ISSUE 2, 2022, 49 – 62.
Online ISSN 2735-4806

In response to these demands, different highly effective cooling techniques have been used to achieve heat transfer enhancement with a minimum of frictional losses, including a variety of passive and active cooling techniques.
Po-Chuan Huang and Young [4] conducted a numerical investigation on the flow field and heat transfer characteristics of two successive porous-block mounted heat sources subjected to pulsating channel flow.

Time-dependent flow and temperature fields were calculated and averaged over a pulsating cycle in a periodic steady-state.
The basic interaction phenomena between the porous substrate and the fluid region, as well as the action of pulsation on the transport process were scrutinized within the study. Steady-state experiments were performed to study general convective heat transfer from an in-line four simulated electronic chips in a vertical rectangular channel using water as the working fluid [5]. Experiment has been performed to investigate the natural convection heat transfer coupled with the effect of thermal conduction from a steel plate with discrete heat sources. The behavior and heat transfer enhancement of a particular nanofluid, Al2O3 nanoparticle– water mixture, flowing inside a closed system that is destined for cooling of microprocessors or other electronic components was investigated by Cong Tam
Nguyen et al [6]. The development of a novel cooling strategy of directly injected cooling for electronic packages was studied by Wits et aL [7]. Two-dimensional forced convection heat transfer between two plates with flush-mounted discrete heat sources on one plate to simulate electronics cooling is studied numerically using a finite difference method in [8]. The effects of rotation on natural convection cooling from three rows of heat sources in a rectangular cavity were studied by Jin et al [9]. Conjugate convective–conductive heat transfer in a rectangular enclosure under the condition of mass transfer within cavity with local heat and contaminant sources is numerically investigated by Kuznetsov and Sheremet [10]. Mathematical model, describing a two dimensional and laminar natural convection in a cavity with heat-conducting walls, is formulated in terms of the dimensionless stream function, vorticity, temperature and solute concentration. The main attention is paid to the effects of Grashof number (Gr), Buoyancy ratio (Br) and transient factor on flow modes, heat and mass transfer.

Steady-state experiments are performed to study general convective heat transfer from an in-line four simulated electronic chips in a vertical rectangular channel using water as the working fluid [11]. Numerical simulation of conjugate, turbulent mixed convection heat transfer in a vertical channel with discrete heat sources was studied by Mathews and Balaji [12]. Investigation of mixed convection heat transfer in a horizontal channel with discrete heat sources at the top and at the bottom was studied by Dogan et al [13]. Laminar-mixed convection of a dielectric fluid contained in a two-dimensional enclosure is investigated [14].

Spray cooling is a very complex phenomena that is of increasing technological interest for electronic cooling and other high heat flux applications since much higher heat transfer rates compared to boiling can be achieved using relatively little fluid [15]. An experimental study of cooling an array of multiple heat sources simulating electronic equipment by a single row of slot air jets positioned above a critical row (row having maximum heat dissipation rate) of the array was conducted [16]. Kabeel et al [18] presented forced convection cooling of heat sources by use liquid (water and dielectrics oil). The study was carried out at different exit conditions, different flow rate, and different heat flux. Also, the study presented the effect of outlet ports positions for water and dielectrics oil to investigate the best position of exit flow. Although earlier works have studied the use of liquid to enhance heat transfer from heat sources, they present few cases of this use.

In the present study, IC's components "chips" is idealized as smooth rectangular blocks with uniform thermal conductivity and constant heat flux. The aim of the present work is to use air to cooling heat sources.
Turbulent heat transfer with three heat sources for air coolant is studied numerically.

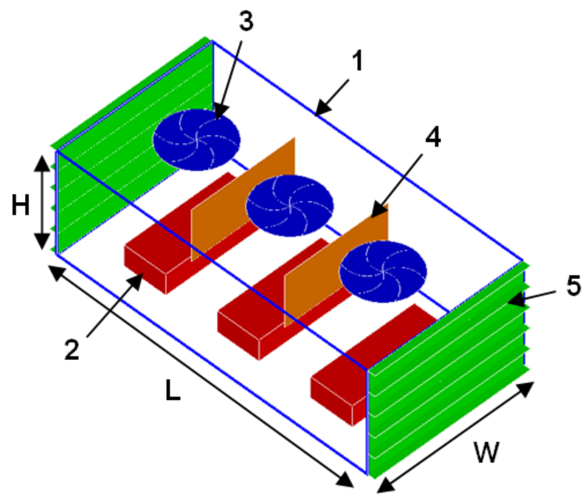
I. PHYSICAL PROBLEM

The physical model, that describes the case of studying, the schematic diagram of the physical model is shown in Fig. 1, the flow is approximated to be three-dimensional flow.

Print ISSN 2735-4792
VOLUME 5, ISSUE 2, 2022, 49 – 62.
Online ISSN 2735-4806

II. PROBLEM DESCRIPTION

A cabinet 3-D view is shown in Fig. 1, which describes the case study. The cabinet as cooling chamber (1) is made of Acrylic with of 10 mm thickness, width of W=150 mm, height of H=100 mm and total length of L=300 mm. The heat sources (chips) (2) are assumed to be of a constant heat flux and the enclosure having 100mm length, 20mm width and 20mm height. The supply air enters the cabinet through the three circular fans (3) in the top at a mass flow rate of 0.0001 kg/s for each to cool the chips. We put the Rectangular Plates (4) between the fans ports. The grilles (5) representing as the exhaust ports are located in the right and left sides.



1-Cabinet.
2-Heat Sources (chips).
3-Circular Fan.
4- Rectangular Plate.
5- Grill.

Fig.1. Schematic diagram of Case Study

Print ISSN 2735-4792
VOLUME 5, ISSUE 2, 2022, 49 – 62.
Online ISSN 2735-4806

III. Numerical solution

The three-dimensional steady-state, turbulent compressible flow in the cabinet is governed by continuity, momentum and energy equations together with applying the turbulence k-ε model. After setting the governing equations, a numerical method is used to convert it into a set of algebraic equations. Then, the computational fluid dynamics (CFD) software package used in this study is called ***Icepak4.2.8.*** It solves the governing mass, momentum and energy (Navier-Stokes) equations numerically using the finite volume method.

III.1 Governing Equations to be solved

The governing equations of the flow are modified according to the conditions of the simulated case. Since the problem is assumed to be steady state with low velocities. Therefore, time dependent parameters are dropped together with the viscous dissipation term, are dropped from the equations. The resulting equations are:

- Conservation of mass:

$$\nabla.(\rho\bar{V}) = 0$$

- x-momentum:

$$\nabla.(\rho u\bar{V}) = -\frac{\partial P}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x$$

- y-momentum:

$$\nabla.(\rho v\bar{V}) = -\frac{\partial P}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y$$

- z-momentum:

$$\nabla.(\rho w\bar{V}) = -\frac{\partial P}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z$$

- Thermal Energy:

$$\nabla.(\rho e\bar{V}) = -p\nabla.\bar{V} + \nabla.(k\nabla T) + \dot{q}$$

Where u, v and w are the components of velocity v in x, y and z directions; f and τ are body force and shear stress components; q is heat generation rate; and *P, T*, ρ and *K* are pressure, temperature, density and thermal conductivity of the fluid.

Print ISSN 2735-4792
VOLUME 5, ISSUE 2, 2022, 49 – 62.
Online ISSN 2735-4806

III.2 Boundary conditions

The boundary conditions of above governing equations for the considered problem as shown in Fig. 1 are determined as:

- On solid walls (Chips and enclose): No slip condition is applied for velocities, i.e., $u = v = 0$.
- The thermal condition on cooling walls of enclose (right, top and left) are $T = T_o$
- On adiabatic bottom walls (Chips and enclose) are $q = 0$
- On heat source walls (right, top and left)

III.3 Cooling Effectiveness Factor

The Cooling effectiveness factor (CEF) is defined according to as the outlet air temperature difference to the average cabinet temperature difference:

$$CEF = \frac{T_o - T_i}{T_{avg,Cabinet} - T_i}$$

IV. Results and Discussions

IV.1 Effect of inlet and outlet air ports:

The cooling effectiveness factor (CEF) is calculated for all cases at different ambient temperatures and listed in Table 1.

From the above cases in Fig. 2, the optimum case is which has the greatest cooling effectiveness factor (CEF). The best position for the input and output locations was found in case (1W) and shown in Fig. 3.

INTERNATIONAL JOURNAL OF
ARTIFICIAL INTELLIGENCE AND EMERGING TECHNOLOGY
Print ISSN 2735-4792
VOLUME 5, ISSUE 2, 2022, 49 – 62.
Online ISSN 2735-4806

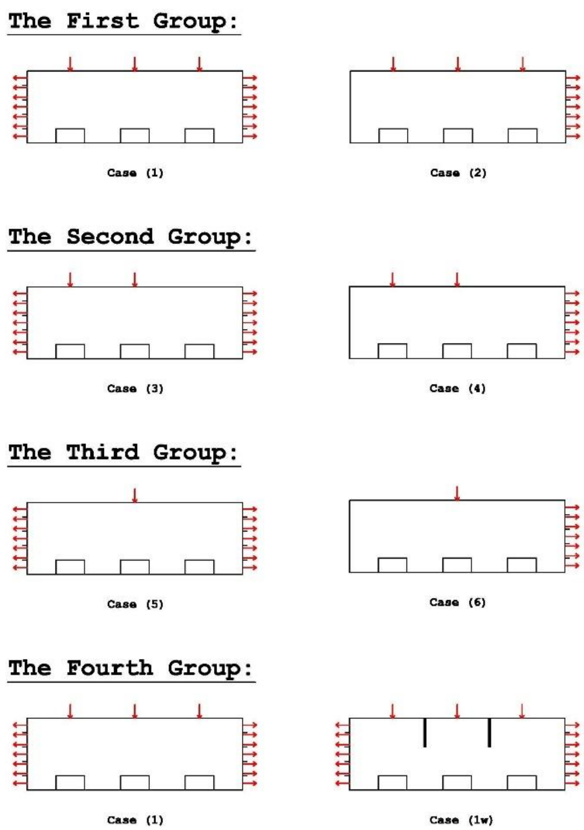


Fig.2. Inlet and outlet air ports location groups

The temperature and velocity contours for the best case are shown in Fig. 4a and b.

Table 1: CEF values of all cases

Case no.	CEF
1	1.25
2	0.83
3	0.85
4	0.82
5	0.53
6	0.4
1w	1.3

INTERNATIONAL JOURNAL OF
ARTIFICIAL INTELLIGENCE AND EMERGING TECHNOLOGY

Print ISSN 2735-4792 | VOLUME 5, ISSUE 2, 2022, 49 – 62. | Online ISSN 2735-4806

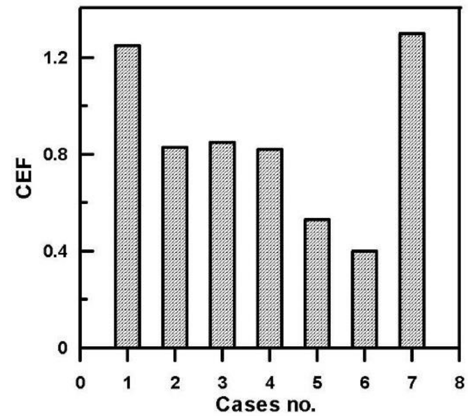
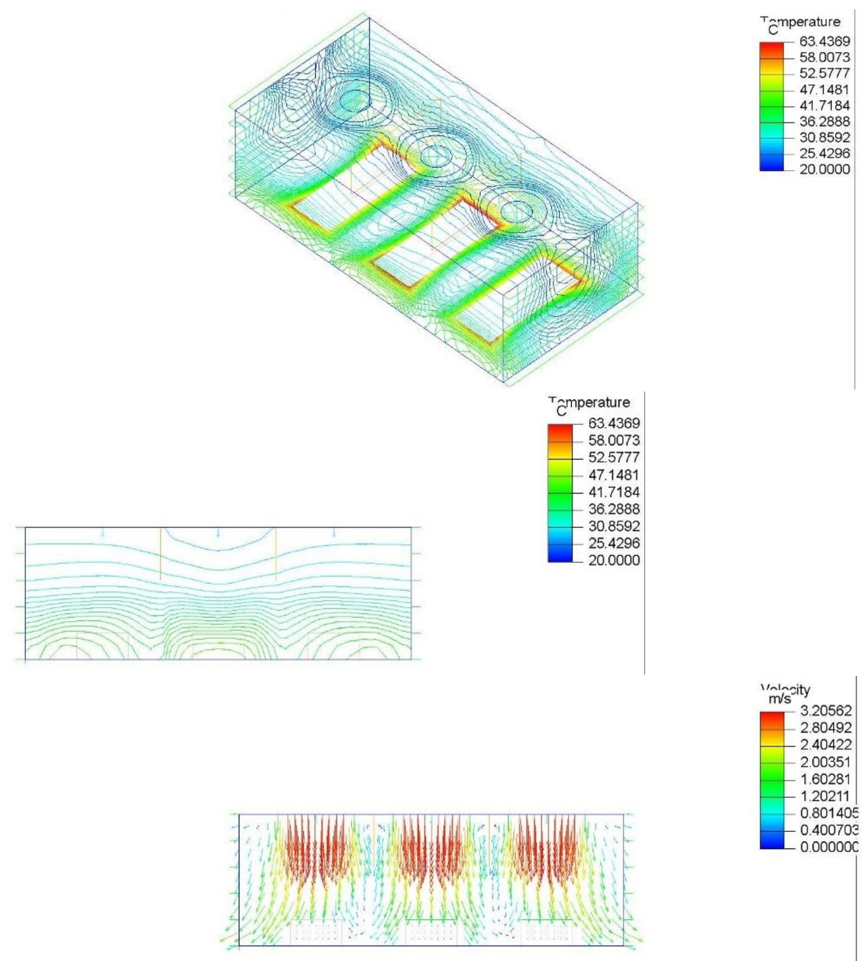


Fig.3. Effect of the inlet and outlet ports positions



Print ISSN 2735-4792
VOLUME 5, ISSUE 2, 2022, 49 – 62.
Online ISSN 2735-4806

IV.2 Relationship between the average surface nusselt number and heat flux

Fig.5 presents the relationship between the average surface nusselt number (Nu) and heat flux ($q^{"}$) for using water. The average surface nusselt number increases with the increase heat flux and average surface nusselt number for chip 2 is lower than that chip 1 and 3. The following empirical formal between Nu and $q^{"}$ can be obtained from the theoretical data:

$$Nu = 6.22 + 0.025q^{"} - 2.064q^{"2} \pm 4$$

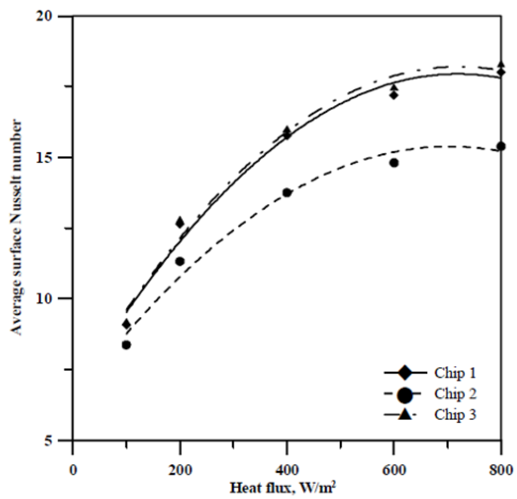


Fig.5.The relationship between the average surface Nusselt number and heat flux.

V. Conclusion

The temperature and velocity distributions in the cabinet with air cooling at different geometries are studied.

It has been established that, over the range of the variables studied and for the particular geometries:

1. Air cooling gives good cooling results at a low air velocity, a small temperature difference between the inlet and outlet levels.

2. The optimum case for best temperature gradient and velocity distribution is obtained when the outlet port at the right and the left walls and 3-Fans as inlet ports at plates between it.

Print ISSN 2735-4792
VOLUME 5, ISSUE 2, 2022, 49 – 62.
Online ISSN 2735-4806

3. The velocity of supply air must be increased (decrease the Archimedes number, Ar) or decreasing the inlet air temperature in order to enhance the air cooling performance.

4. Maximum heat transfer occurs at the chip leading and side edges.

5. The chip surface temperature is the highest at the top-row heaters.

6. The larger cell breaks into multiple smaller cells at higher heat fluxes.

Nomenclature

CFD	Computational fluid Dynamics , -
u	Velocity component in x-direction, m/s
$\overline{V}$	Velocity Vector, -
v	Velocity component in y-direction, m/s
w	Velocity component in z-direction, m/s
f	Body Force, -
P	Pressure, Pa
T	Temperature, K
T_o	Ambient Temperature, K
$T_{av,\ Cabinet}$	Average cabinet temperature, K
K	Thermal Conductivity, W/m.K
$\dot{q}$	Heat Generation rate (heat flux), W/m³
ρ	Density, kg/m³
τ	Shear Stress, N/m²
CEF	Cooling effectiveness factor , -
x, y, z	Position Coordinates

Gr	Grashof number
Br	Buoyancy ratio
Nu	Nusselt number
Ar	Archimedes number

Print ISSN 2735-4792
VOLUME 5, ISSUE 2, 2022, 49 – 62.
Online ISSN 2735-4806

DIMENSIONLESS GROUP

SUBSCRIPTS:

a	air
amb	ambient
avg	average
c	cold
cab	cabinet
h	hot
i	inlet
o	exit
t	turbulent

T	thermal
w	wall
ε	dissipation

SUBSCRIPTS:

-	average
'	fluctuating

ACKNOWLEDGMENTS

Thanks go first to ALLAH. I wish to express my appreciations and deep gratitude to my supervisors Prof. Dr. Gamal Sultan, Prof. Dr. Abd-Elnaby Kabeel and Dr. Alsaied Khalil for their continuous guidance and encouragement throughout the course of this work.

I also wish to express my thanks and appreciations for Mechanical Engineering (Energy Engineering) Laboratory staffs and workshop of the Faculty of Engineering, Tanta University, for their assistance.

I wish also to thanks my family who gave me more than support. They gave me the power to continue this research. I think that there are no words that can express my love and grateful for them. I express my gratitude to my family because this which remains at the end in the heart.

Print ISSN 2735-4792
VOLUME 5, ISSUE 2, 2022, 49 – 62.
Online ISSN 2735-4806

REFERENCES

[1] S. Mohapatra and D Loikits, Advances in Liquid Coolant Technologies for Electronics Cooling, Proceedings of the 21st IEEE Semiconductor Thermal Measurement and Management Symposium, San Jose, CA, 2005, pp. 354-360.

[2] M. Mahalingam, H. Berg, Thermal trend in component level packaging, Int. J. Hybrid Microelectron. 7 (1984) 1–9.

[3] R. Simon, Thermal management of electronic packages, Solid State Technol. 26(1983) 131–137.

[4] Po-C. Huang, C.-Fu Yang, Analysis of pulsating convection from two heat sources mounted with porous blocks, International Journal of Heat and Mass Transfer 51 (2008) 6294–6311

[5] C. Linhui , T. Huaizhang, L. Yanzhong, Z. Dongbin, Experimental study on natural convective heat transfer from a vertical plate with discrete heat sources mounted on the back, Energy Conversion and Management 47 (2006) 3447–3455

[6] C. T. Nguyen, G. Roy , C. Gauthier, N. Galanis, Heat transfer enhancement 8 using Al2O3 water nanofluid for an electronic liquid cooling system, Applied Thermal Engineering 27 (2007) 1501– 1506

[7] W.W. Wits, T.H.J. Vaneker, J.H. Mannak, R. Legtenberg, Novel cooling strategy for electronic packages: Directly injected cooling, CIRP Journal of Manufacturing Science and Technology (2008)

[8] G. P. XU, K. W. TOU and C. P. TSO, Numerical modeling of turbulent heat transfer from discrete heat sources in a liquid-cooled channel, lnt. J. Heat Mass Transfer. Vol. 41, No. 10, pp. 1157- 1166, 1998

[9] L.F. Jin , K.W. Tou , C.P. Tso , Effects of rotation on natural convection cooling from three rows of heat sources in a rectangular cavity, International Journal of Heat and Mass Transfer 48 (2005) 3982–3994

[10] G.V. Kuznetsov, M.A. Sheremet, Conjugate heat transfer in an enclosure under the condition of internal mass transfer and in the presence of the local heat source, International Journal of Heat and Mass Transfer 52 (2009) 1–8

[11] H. Bhowmik, C.P. Tso, K.W. Tou, F.L. Tan, Convection heat transfer from discrete heat sources in a liquid cooled rectangular channel, Applied Thermal
[12] R. N. Mathews, C. Balaji, Numerical simulation of conjugate, turbulent mixed convection heat transfer in a vertical channel with discrete heat sources, International Communications in Heat and Mass Transfer 33 (2006) 908–916
[13] A. Dogan , M. Sivrioglu , S. Baskaya, Investigation of mixed convection heat transfer in a horizontal channel with discrete heat sources at the top and at the bottom, International Journal of Heat and Mass Transfer 49 (2006) 2652–2662
[14] A. Rivas-Cardona, A. Hernandez- Guerrero, R. Romero-Méndez, R. Lesso- Arroyo, Liquid-mixed convection in a closed enclosure with highly-intensive heat fluxes, International Journal of Heat and Mass Transfer 47 (2004) 4089–4099
[15] J. Kim, Spray cooling heat transfer: The state of the art, International Journal of Heat and Fluid Flow 28 (2007) 753–767
[16] A.S. Huzayyin, S.A. Nada, M.A. Rady, A. Faris, Cooling an array of multiple heat sources by a row of slot air jets, International Journal of Heat and Mass Transfer 49 (2006) 2597–2609
[17] A. E. Kabeel, A. Khalil and F. Eldosary, Liquids cooling of electronic heat sources, Mansoura Engineering Journal, March 2009
[18] S. V Patankar, "Numerical Heat Transfer and Fluid Flow", Hemisphere Publishing Company, New York, 198.