Al-khwarizmi
Engineering
Journal

Al-Khwarizmi Engineering Journal, Vol.3, No.2, pp ١٧-٣١ (2007)

Investigation of heat transfer phenomena and flow behavior around electronic chip

Dr. Sattar j. Habeeb
Mechanical Engineering Dept.
Technology University

(Received 1 November 2006; accepted 24 April 2007)

Abstract:

Computational study of three-dimensional laminar and turbulent flows around electronic chip (heat source) located on a printed circuit board are presented. Computational field involves the solution of elliptic partial differential equations for conservation of mass, momentum, energy, turbulent energy, and its dissipation rate in finite volume form. The k-ε turbulent model was used with the wall function concept near the walls to treat of turbulence effects. The SIMPLE algorithm was selected in this work. The chip is cooled by an external flow of air. The goals of this investigation are to investigate the heat transfer phenomena of electronic chip located in enclosure and how we arrive to optimum level for cooling of this chip. These parameters, which will help enhance thermal performance of electronic chip and flow patterns, through the understanding of different factors on flow patterns. The results show the relation between the temperature rise, heat transfer parameters (Nu, Ra) with (Ar, Q) for two cases of laminar and turbulent flows.

Keywords:
Electronic equipment, Fluid flow, Convection heat transfer

Introduction:

Electrical and mechanical engineering have long been aware of the potential thermal related reliability problems associated with high powered, high density of electronic circuitry. Circuit designers have traditionally used prototype testing to monitor the thermal response of new circuit designs. But the prohibitive cost of prototype development both in terms of financial investment and in terms of time to deliver the finished product to market restrict the designer in this ability to developed on optimized thermal design. During the early stages of the design process the circuit concerned with determining the sensitivity of board temperatures to changes in basic design parameters, such as thermal conductivity of the board, flow velocity and package location (Peterson, and Ortega, 1990). Conjugate problems can be divided into two parts, part one specialized with heat transfer phenomena on solid (Printed Circuit Board PCB) and solve it using three-dimensional Possion heat conduction in homogenous solid. The second part is to solve the governing equations for flow field region mass, momentum, energy and k-ε turbulence model.

Cooling of electronic equipment has been studied in great detail in recent years. There are a many papers, which have been written to analyze the flow pattern and thermal performance of printed circuit board PCB. The literature review is presented in subjective manner, and not in a comprehensive one. It is divided into three parts (thermal performance of PCB, flow field characteristics in enclosure, and turbulence modeling). Icoz, T., Verma, N., and Jaluria, Y. (2006) studied the heat transfer phenomena from multiple heat sources simulating electronic components and located in a horizontal channel. Two experimental setups were fabricated for air and liquid cooling experiments to study the effects of different coolants. De-ionized water was used as the liquid coolant in one case and air in the other. The effects of separation distance and flow conditions on the heat transfer and on the fluid flow characteristics were investigated. The results from simulations and experiments were combined to create response surfaces and to find the optimal values of the design parameters. Wang, Q., and Jaluria, Y., (2002) found that inducing oscillations in the driving flow enhances the heat transfer rates from the heat sources and oscillatory flow is a common phenomena encountered in electronic cooling applications.

Icoz, T., and Jaluria, Y., (2005), used that concurrent simulation and experiment for design of cooling systems for electronic equipment, which consists of multiple heat sources in a channel. The conventional engineering design and optimization are based on sequential use of computer simulation and experiment. However, the conventional methods fail to use the advantages of using experiment and simulation concurrently in real time. Chen, K. N., (2006), studied of a PCB carrying a heavy CPU cooling fan and supported by six fastening screws was investigated by the modal testing experiment and was analyzed by the finite element method. After the finite element model was verified by the experimental results, the locations of the six supporting screws were optimized to achieve a maximum fundamental frequency for the loaded PCB.

Lee, Culham, and Yovanovich (1991) examined some of common design parameters in design of microelectronic circuity such as the thermal conductivity and surface emissivity of the circuit board under forced convection condition. The flow velocity of the cooling fluid and the positioning and power dissipation on the heat sources are studies to determine the relative merit of each as a means of controlling circuit board temperatures. Also they solved three-dimensional Laplace equation for heat flow in homogenous solids in PCB and they solved boundary layer equations over the PCB based on Blasius's solution and coupling their solution to find temperature field in solid and flow domains. Conjugate heat transfer problems in this problem can be extremely complex due to the interaction of the fluid and solid domains and the necessity for the governing equations in each domain to be satisfied simultaneously. Lee, Cuham, and Yovanovich (1991) found the parameters on which the operating temperature depends in natural convection heat transfer and they showed the effect of these parameters such as the thermophysical properties, package location, and the applied power level on localized temperature and average Nusselt number. Finally they showed the effect of radiation and they recommended that the radiation must be included also as radiation may account for a significant fraction of the total heat dissipation from heat sources as well as from entire circuit board surface.

Lee, Cuham, Lenczyk, and Yovanovich (1990) developed a conjugate model for airflow over flat plates with arbitrarily located heat sources based on the integral formulation of the boundary layer equations combined with a finite volume solution, which assumes two-dimensional heat flow within the plate. The fluid and solid solutions are coupled through an iterative procedure, allowing a unique temperature profile to be obtained at fluid-solid interface, which simultaneously satisfies the temperature field within each domain. Lee, Cuham, Jeakins, and Yovanovich (1992) used an analytical routine, META to predict wall temperatures along a flat rectangular duct with heated rectangular modules attached to one surface. The rotationally dominant inlet velocity, resulting from an axial fan located at the entrance to the duct is approximated using a simple linear velocity variation across the principle flow direction. Thermal simulation is compared to publish experimental data obtained over a range of channel Reynolds number from (600 to 1800). This study examined the effect of these non-uniform inlet conditions on the surface temperature of board modules within the system. META was used to simulate the thermal performance of populated circuit boards when inlet velocities were varied across the width of the circuit boards. Afrid and Zebib (1989) discussed a numerical study of natural convection air-cooling of single and multiple uniformly heated devices. A two-dimensional, conjugate, laminar flow model is used. They found that for the multi-component cooling, the effects of component thickness, the spacing between components, non powered components, and highly powered components are very important to arrive at qualitative suggestions that may improve the overall cooling of multi-component system. They showed the results for the case of a single heated component, the temperature rise varies linearly with the heat generation for cases of many heated component, it was found that increased spacing between components and increased component thickness reduce the temperature rise.

Mahaney, Ramadhyni, and Incropera (1989) used a vectorized finite-difference marching technique. The steady state continuity, momentum, and energy equations are solved numerically to evaluate the effects of buoyancy-induced secondary flow on forced flow in a horizontal rectangular duct with a four-row array of (12) heat sources flush mounted to the bottom wall. Also they showed that for a fixed Rayleigh number and decreasing Reynolds number, the row-average Nusselt number decrease, reach a minimum, and subsequently increase due to buoyancy effect. Thus, due to buoyancy-induced secondary flow, conditions exist for which heat transfer reducing the flow rate may enhance and hence the pump power requirement.

Mahaney, Icropera, and Ramadhyani (1990) investigated mixed convection heat transfer from a four-row, in-line array of (12) square heat sources that are flush mounted to the lower wall of a horizontal, rectangular channel. The experimental data encompass heat transfer regimes characterized by pure natural convection, mixed convection, laminar forced convection, where was water used as a coolant.

In most literature surveyed, the literature of cooling of electronic component located in an enclosure can be divided into three parts. **Part one** discussed analytical and numerical simulation of PCB and studies all the parameters that may effect the design of PCB. The aim of these studies is to maintain the operation temperature of the chips on PCB below the maximum allowable temperature as specified by various design constraints in order to ensure reliable performance of integrated circuits and to maintain minimum failure rates of components. The **second part**, covering with simulation of recirculating flow in enclosure and solving governing equations for mass, momentum and energy equations for flow field. In general for laminar flow and especially for turbulent flow, these procedures introduced under titles of CFD technique and its flexibility to solve these equations, with some studied for experimental analyses. The **third part**, covering the simulation of turbulent flow with k-ε turbulence model for solution of the governing equations. All these studies solved the problem of cooling of electronic component for two cases, solved the conduction heat transfer in PCB only, or solved the convection heat transfer in flow only, for laminar and turbulent flows.

In present work will be solve the conduction and radiation heat transfer in PCB with convection heat transfer in flow domain and coupled these two solutions in general form to present all domain, solid and fluid sides.

The general transport equation

The transport equations for continuity, momentum, energy, and the turbulence scales κ and ε, all have the general form (Awbi, 1998):

$$\frac{\partial}{\partial t}(\rho\,\varphi)+\frac{\partial}{\partial x}(\rho\,u\varphi)+\frac{\partial}{\partial y}(\rho\,v\varphi)+\frac{\partial}{\partial z}(\rho\,w\,\varphi)=\frac{\partial}{\partial x}\left(\Gamma_\varphi\frac{\partial\varphi}{\partial x}\right)+\frac{\partial}{\partial y}\left(\Gamma_\varphi\frac{\partial\varphi}{\partial y}\right)+\frac{\partial}{\partial z}\left(\Gamma_\varphi\frac{\partial\varphi}{\partial z}\right)+S_\varphi \quad \text{……(1)}$$

Where the terms on the left-hand side of equation (1) include the time-derivative and convective terms, and the terms on the right hand side include the diffusion and source terms. Also, φ is the dependent variable and S_φ is the source term that has different expression for different transport equations. The convection and diffusion terms for all the transport equations are identical with Γ_φ representing the diffusion coefficient for scalar variables and the effective viscosity μ_φ for vector variables, i.e. the velocities. This characteristic of the transport equations is extremely useful when the equations are discretized (reduced to algebraic equations) and solved numerically since only a solution of the general equation (1) is required. In fact equation (1) also represents the continuity equation when φ =1 and S_φ = 0. Table (1), gives the expressions for the source terms S_φ for each dependent variable that is likely to be needed in solving flow problems.

Table (1). Source terms in the transport equations for laminar and turbulent flows.

Equation	φ	Γ_φ	S_φ
Laminar Flow			
Continuity	١	٠	٠
Momentum	***U***	μ	$-P_x+\rho g_x$
Momentum	***V***	μ	$-P_y+\rho g_y$
Momentum	***W***	μ	$-P_z+\rho g_z$
Temperature	***T***	Γ	Q/Cp
Turbulent Flow			
Continuity	١	٠	٠
Momentum	***U***	μ_e	$-P_x+\nabla\cdot(\mu_e\nabla\cdot\vec{U})+\rho g_x$
Momentum	***V***	μ_e	$-P_y+\nabla\cdot(\mu_e\nabla\cdot\vec{U})+\rho g_y$

If we use the Boussinesq approximation, we get

$$\rho g_x=\rho g_z=0$$

$$\rho g_y=-g\left(1-\frac{\Delta T}{T}\right) \text{ where } \Delta T=T-T_r$$

Solution procedure:

Because of the non-linearity of the transport equations, an iterative method of solving the discretization equations to achieve a converged solution is the most plausible approach. An iteration solution starts from guessed values of the dependent variables for the whole field. In deriving the transport equations and their discretized forms there was no equation for pressure except that the pressure gradient was added to the source terms. However, to achieve a convergent solution it is obvious that for the velocity component u, v, and w obtained from a solution of the momentum equation to satisfy continuity, the correct pressure field must be used in the momentum equations. This link between velocity and pressure can be employed in the iterative solution without the necessity of solving a discretization equation for the pressure (Versteeg, and Malalasekera, 1995).

Boundary conditions:

Due to the viscous influences near wall, the local Reynolds number becomes very small, thus the turbulent model which is designed for high Reynolds number become inadequate. Both this fact and the steep variation of properties near wall necessitate special treatment for nodes close to the wall. One way of handling this problem is to use turbulence models in which modification has been introduced to take the viscous effects into account. The use of low-Reynolds number models is one way. However, these models, when used near wall regions where the dependent variables and their gradient vary steeply, require a very fine mesh for adequate numerical solution and may lead to very high computational costs. The approach adapted in this program is called " Wall Function " as suggested by Launder and Spalding (1972). This treatment is based on the fact that the logarithmic law of the wall applies to the velocity component parallel to the wall in the region close to the wall, corresponding to a $y^+ = y.u_t/v$ value in the region $30 \le y^+ \ge 200$.

In the following explanation of the treatment of turbulence quantities near the wall (Davidson (1995)), it is assumed that the region near the wall consists of two layers. The layer nearest the wall is designated the" viscous sublayer" in which the turbulent viscosity is much smaller than molecular viscosity, i.e. the turbulent shear stress is negligible. Ignoring the buffer layer, the second layer is designated the "inertial sublayer "in which the turbulent viscosity is much greater than molecular viscosity, making it a fully turbulent region. These two layers are the wall dominated regions and it is assumed that the total shear stress is constant, an assumption that is supported by experimental data.

The point $y^+ \le 11.63$ is defined to dispose the buffer (transition) layer, and it corresponds to the intersection point between the log-law and the near-wall linear law. Above this point the flow is assumed to be fully turbulent and below this point the flow is assumed to be purely viscous. Then, the particularly simple one-dimensional form of shear stress equation will become (Davidson, and Farhanieh, 1995):

$$\tau = (\mu + \mu_t) \cdot \frac{\partial u}{\partial y} \qquad \ldots\ldots(3)$$

The wall function implemented in the present study can be summarized in table (2).

Table (2) Wall Function for Vector and Scalar Transport and Conditions at Inlet and Exit Region.

Wall Function for Vectors Transport		
Equation	For $y^+ \ge 11.63$ where $\frac{\mu_t}{\mu} >> 1, \tau \approx \tau_w$ $y^+ = y.u_t/v$	For $y^+ \le 11.63$ where $\frac{\mu_t}{\mu} << 1, \tau \approx \tau_w$ $y^+ = y.u_p/v$
Shear Stress	$\tau_w = \mu_t \cdot \frac{\partial u}{\partial y} \approx \mu_t \cdot \frac{u_p}{y}$	$\tau_w = \mu \cdot \frac{\partial u}{\partial y} \approx \mu \cdot \frac{u_p}{y}$
Viscosity	$\mu_t = \rho u_t y \kappa_1 \big/ ln(Ey^+)$	-----
Turbulent Kinetic Energy	$k_P = C_\mu^{-0.5} \cdot u_t^2$	-----
Energy Dissipation	$\varepsilon_P = \frac{1}{\kappa_1 \cdot y} u_t^3$	-----
Wall Function for Scalar Transport		
	For $y^+ \ge 11.63$ where $\frac{\Gamma_t}{\Gamma} >> 1, q \approx q_w$	For $y^+ \le 11.63$ where $\frac{\Gamma_t}{\Gamma} << 1, q \approx q_w$
Heat Flux Parameter	$\frac{q_w}{C_p} = \frac{\rho u_t (T_w - T_p)}{\sigma_t \left[u^+ + P(\frac{\sigma}{\sigma_t}) \right]}$ where	$\frac{q_w}{C_p} = \frac{\mu}{\sigma y}(T_w - T_p)$

	$P(\frac{\sigma}{\sigma_t}) = 9.24\left[(\frac{\sigma}{\sigma_t})^{0.75} - 1\right] * \left[1 + 0.28e^{(-0.007/(\sigma/\sigma_t))}\right]$	
Condition at Inlet Region		
Velocities	U=Uin , V=W=0.	
Turbulent kinetic energy	$k_{in} = 1.5 I_u^{\,2} \cdot u_{in}^{\,2}$	
Energy dissipation rate	$\varepsilon = k^{1.5} / \Re \cdot H$	
Condition at Exit Region		
Normal velocity	$u_{out} = u_{in} \cdot \frac{(\rho A)_{in}}{(\rho A)_{out}}$	
All vectors and scales parameters	$\frac{\partial \varphi}{\partial x} = 0 \quad , \quad \varphi = u, v, w, p, k, \varepsilon$	

Thermal module:

Figure (1) demonstrated for 3D laminar and turbulent flow involving conjugate heat transfer. The flow is over a rectangular heat-generating electronic chip, which is mounted on a flat circuit board. The heat transfer involves the coupling of conduction in the chip with radiation and convection in the surrounding fluid. The physics of conjugate heat transfer such as this is common in much engineering application, including the design and cooling of electronic components. In this study the electronic chip generate heat dissipation from 2-10 W and have a bulk conductivity of 1.0 W/m^2.k, the circuit board conductivity is assumed to be of the order of magnitude lower than 0.1 W/m^2.k. The airflow enters the system at 25 C^o with Reynolds number changes for both laminar and turbulent cases.

RESULTS:

Velocity vector and temperature contour around electronic chip.

Figures (2, and 3) show the velocity vector distribution and isothermal contours for laminar and turbulent flow respectively. In figure (2) where the flow is laminar, the velocity vector is assumed to be uniform at inlet region and buoyant flow is generated by buoyancy force after the location of electronic chip. Then the buoyant flow arises and collides with the jet flow in enclosure. The jet flow in turbulent flow is strong compared with the same case in laminar flow, so the buoyant force becomes weak in this case. Isothermal contours represent the flow behavior around electronic chip where the max. temperature in enclosure in laminar and turbulent cases are 94, and 74 C^o respectively

Effect of Archimedes number on Temperature rise percentage in the enclosure

Figure (4) represents the relation between temp. rise in the duct and Ar for different cases, laminar and turbulent flow. In all curves the relation which represent a log scale for Ar and the slope of the curve which increase according to increasing value of Ar. This study applied for Re=600 and 2800 according to inlet velocity. The power of electronic chip is changed in each test so the Grashof number will be changed, therefore the Archimedes number will be changed too. The relation indicated that for the same temp. rise the value of Ar in turbulent flow is greater than in laminar flow.

Effect of heat dissipation from the electronic chip on Temperature rise percentage in the enclosure

Figure (5) declares the effect of heat generation in the electronic chip on temp. rise in the duct, where the relation has the same behavior on both laminar and turbulent flows. The results shows that in turbulent flow, temp. rise is greater than for the case in the laminar flow at the same heat dissipation from the electonic chip.

Effect of Archimedes number on average Nusselt number

Figure (6) shows the effect of Ar on average Nusselt number for Re=600, and 2800. For the same average Nusselt number, we could see that, the range of Ar between (0.6-2) in turbulent flow, but in laminar flow the range will become between (20-80). That lead to the fact that in turbulent flow the heat dissipation from the chip is greater.

Effect of heat dissipation from the electronic chip on average Nusselt number

Figure (7) shows the linear relation between average Nusselt number for and heat dissipation from the electronic chip, the relation can be represent as below equation: $\overline{Nu} = A1 + A2 * Q$, Where the constants A1, A2 are defend in below table:

A1	A2	
0.0533	2.0794	Laminar flow
0.0529	2.0338	Turbulent flow

Effect of heat dissipation from the electronic chip on Rayleigh number

Figure (8) represents the relation between Ra and Q where the results show linear behavior between them for both cases. For the same heat dissipation from the electronic chip, the value of Ra in turbulent flow is smallest than in laminar flow. The relation is represented by the equation: $Ra = A1 + A2 * Q$, Where the constant A1, A2 are defined in below table:

A1	A2	
2.523×10^6	2.5×10^6	Laminar flow
1.459×10^6	2.254×10^6	Turbulent flow

Conclusions:

The importance of computational modeling for flow cooling of the electronic chip was demonstrated. To simulate internal flow properly all factors have been clearly detailed. In recirculating flow with mixed convection heat transfer in duct, we could see proportional behavior between Ar and $\overline{Nu}$ for different cases but the slopes of the curve lines are different according to value of Ar. The flow patterns and isotherms do not show any significant difference between the cases of laminar or turbulent flow other than slight shift and changes in streamline and isotherm values.

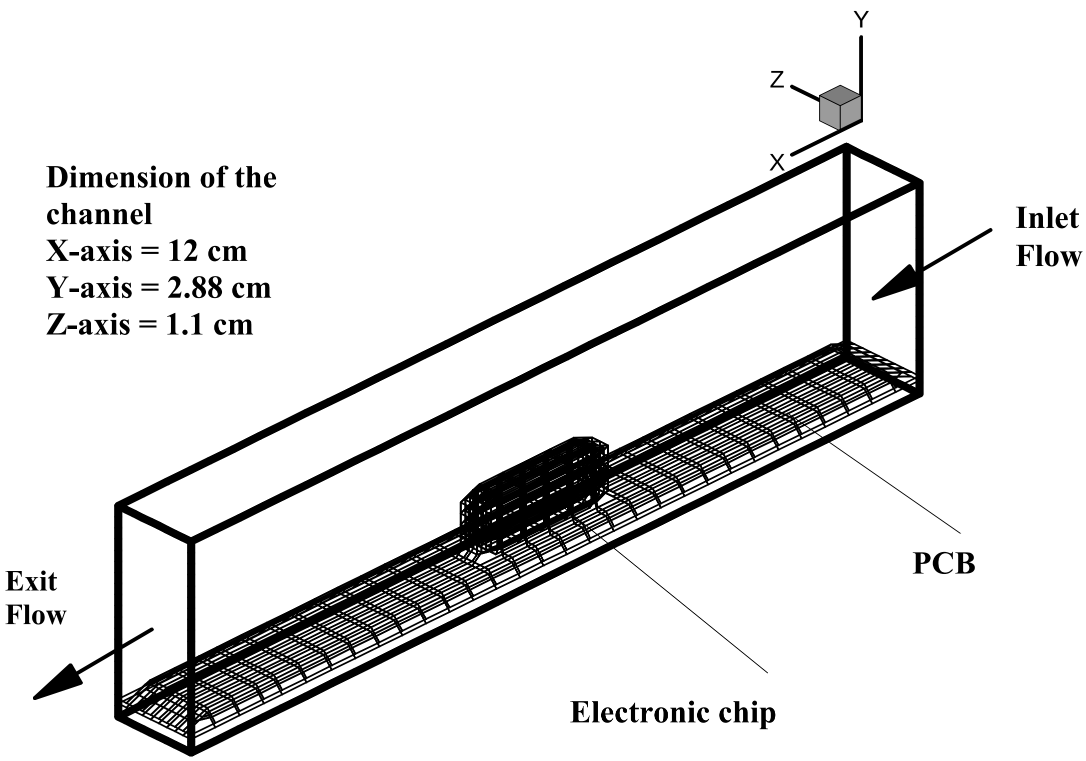

Fig. 1 Schematic diagram of the enclosure

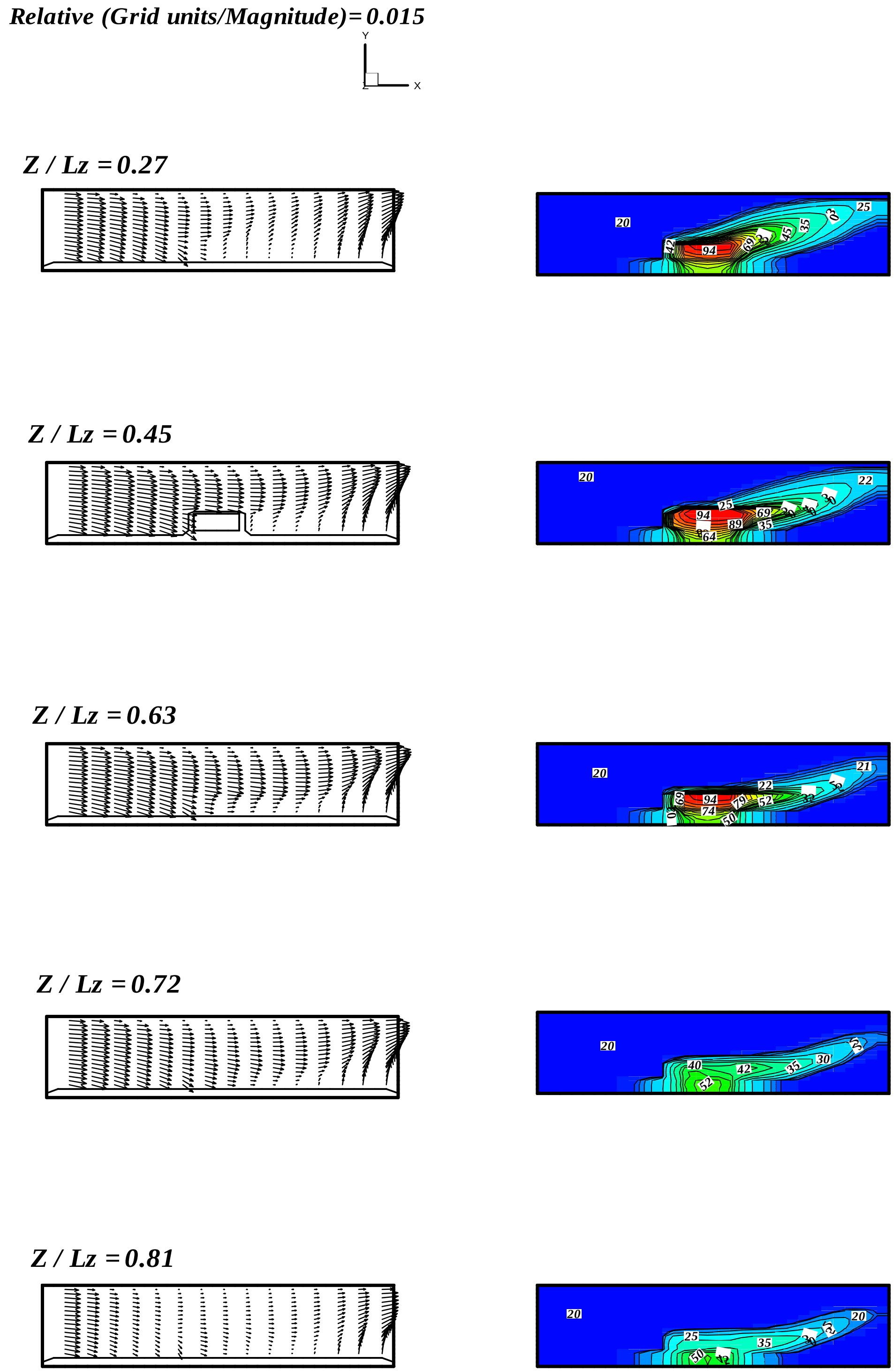

Fig. 2 Velocity Vector and Isothermal Contour for Laminar Flow Re=600.

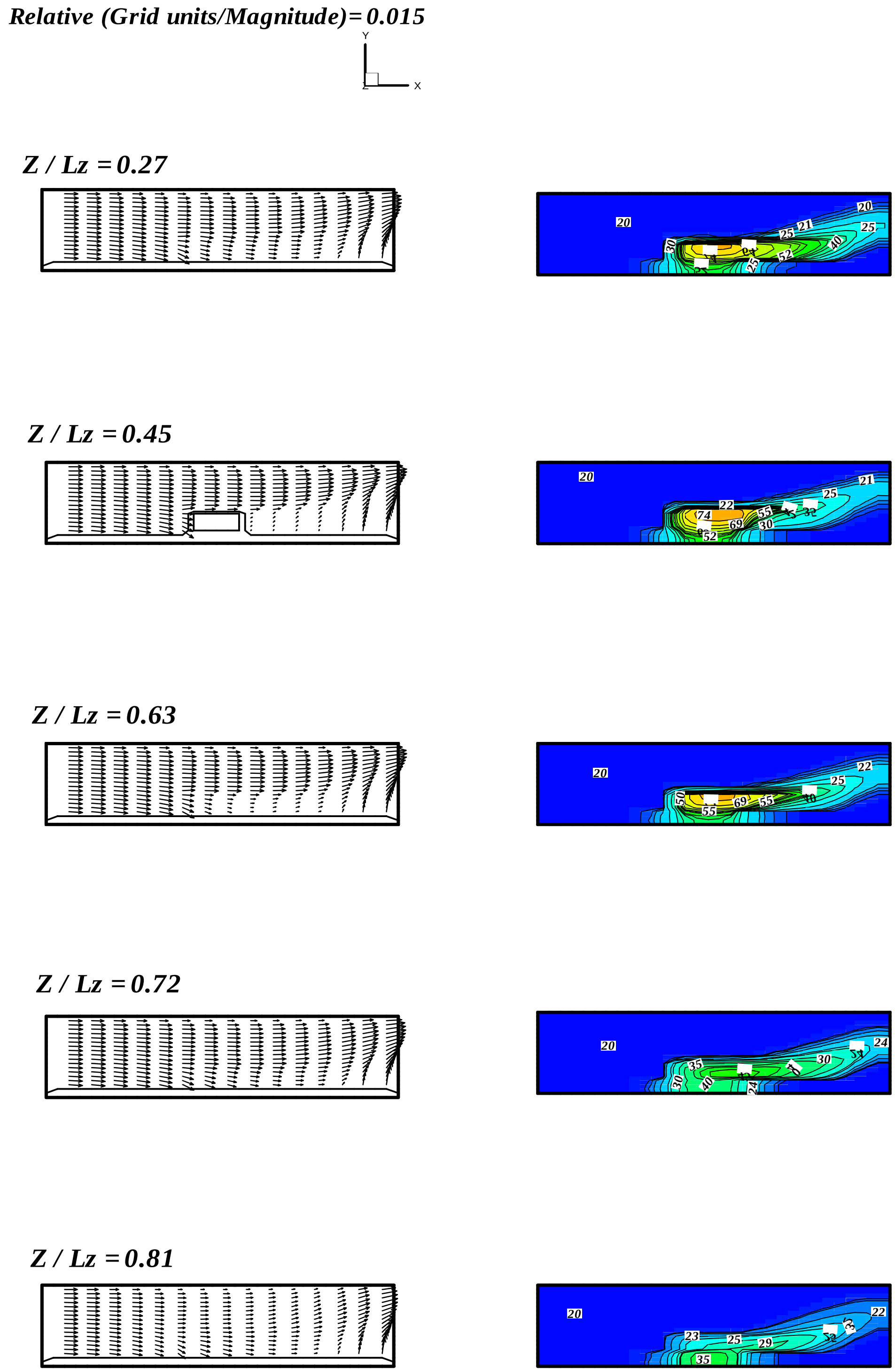

Fig. 3 Velocity Vector and Isothermal Contour for Turbulent Flow Re=2800.

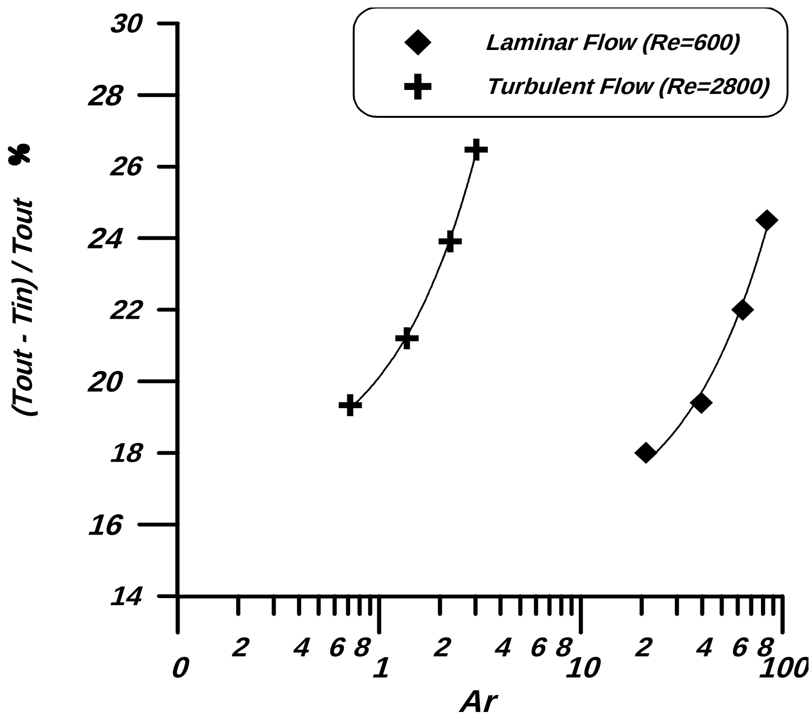

Fig. 4 Effect of Archimedes Number on Temperature Rise Percentage in the Enclosure

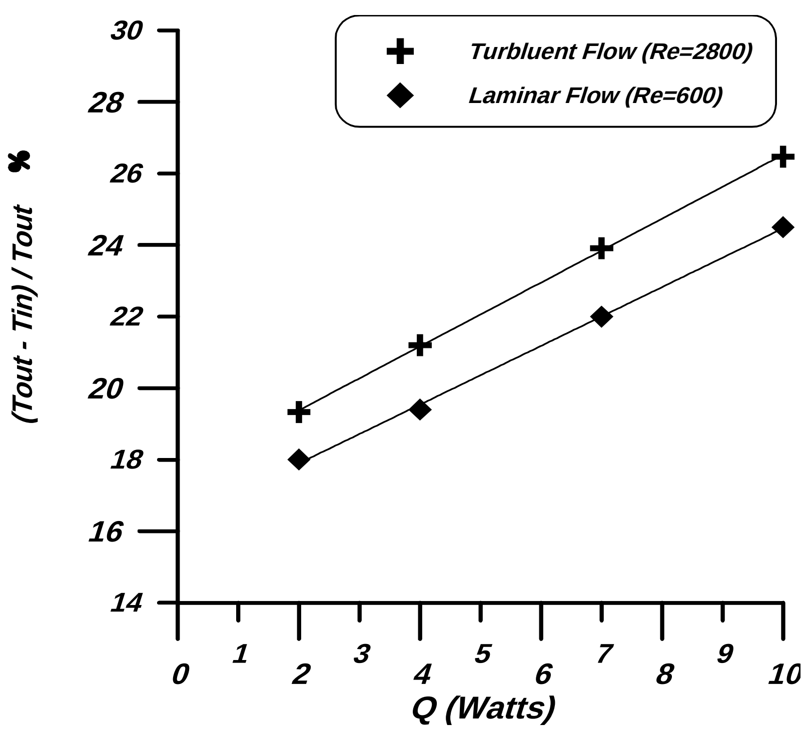

Fig. 5 Effect of Heat Dissipation from the Electronic Chip on Temperature Rise Percentage in the Enclosure

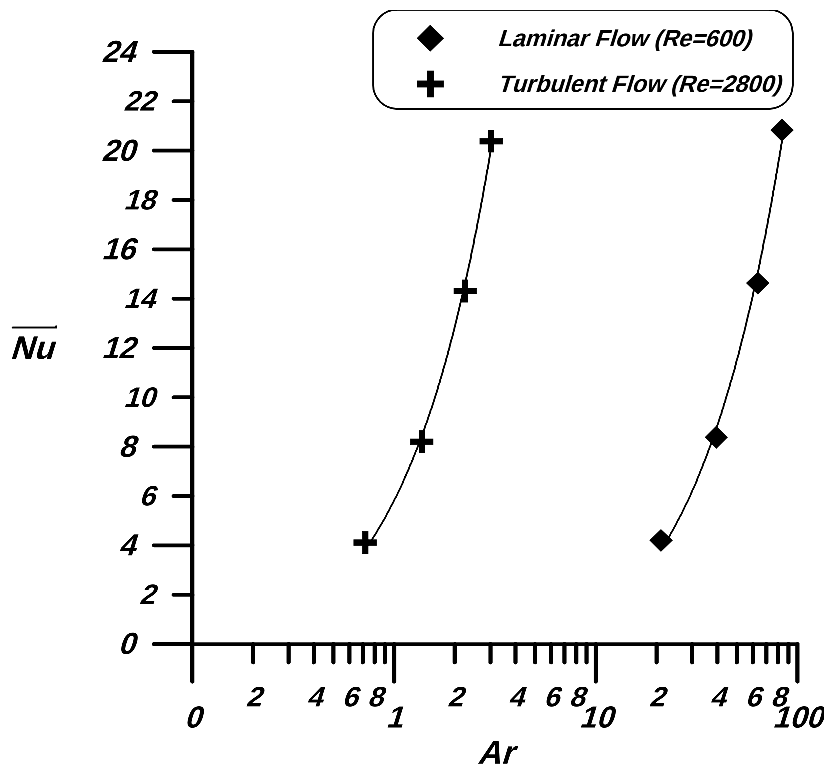

Fig. 6 Effect of Archimedes Number on Average Nusselt Number.

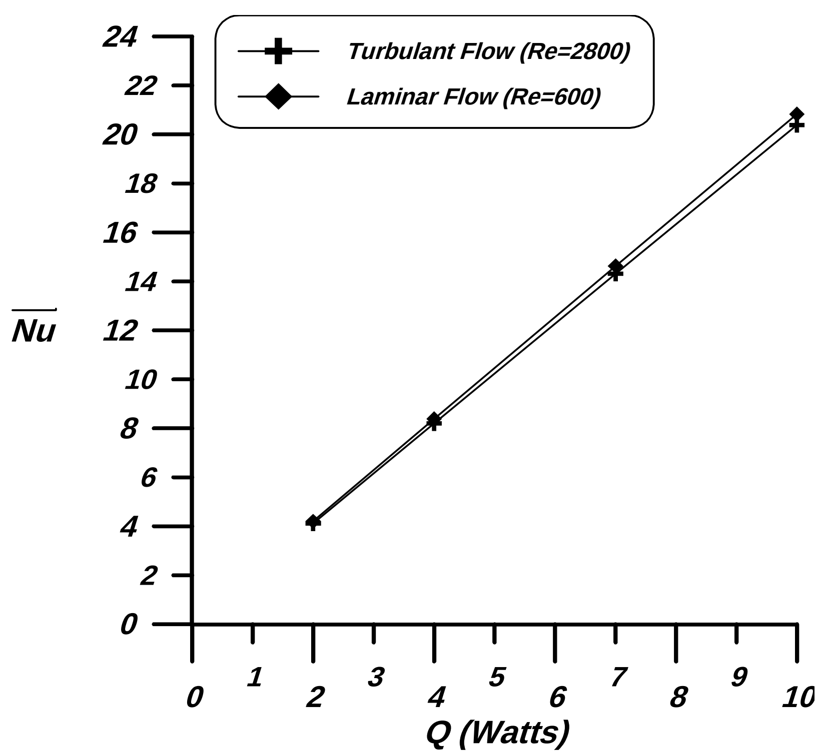

Fig. 7 Effect of Heat Dissipation from the Electronic Chip on Average Nusselt Number

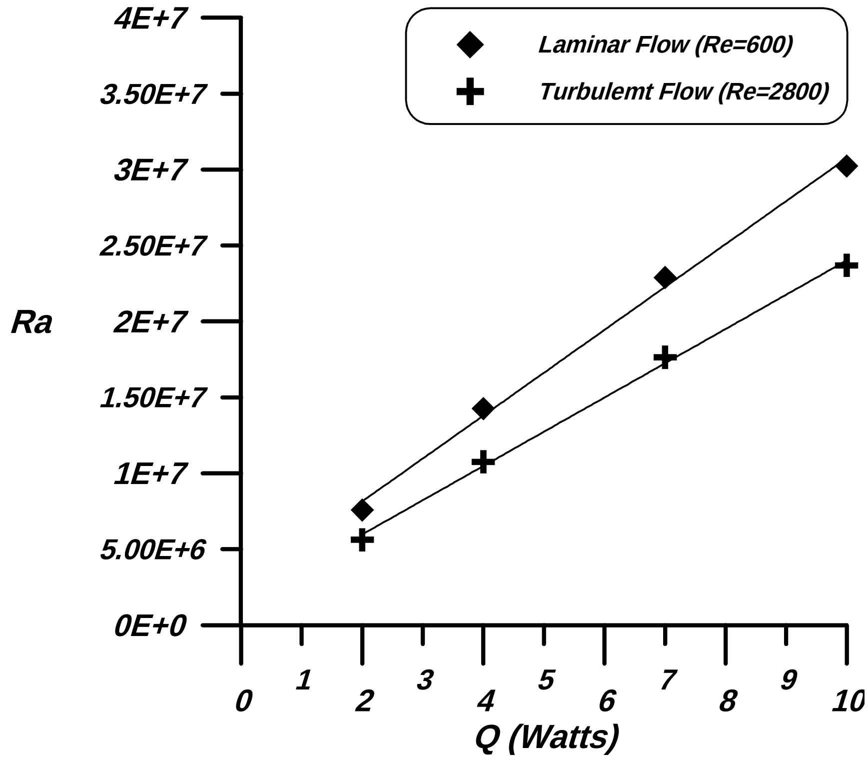

Fig. 8 Effect of Heat Dissipation from the Electronic Chip on Rayleigh Number.

References:

- **Peterson, G. P., and Ortega, A., 1990**, "Thermal Control of Electronic Equipment and Devices" Advance in heat transfer, Vol. 20, pp. 181-245.
- **Icoz, T., Verma, N., and Jaluria1, Y., 2006,** "Design of Air and Liquid Cooling Systems for Electronic Components Using Concurrent Simulation and Experiment," ASME J. Heat Transfer, Vol. 128, DECEMBER, pp. 466–478.
- **Wang, Q., and Jaluria, Y., 2002,** "Unsteady Mixed Convection in a Horizontal Channel With Protruding Heating Blocks and a Rectangular Vortex Promoter," Phys. Fluids, Vol. 14, pp. 2109–2112.
- **Icoz, T., and Jaluria, Y., 2005,** "Design of Cooling Systems for Electronic Equipment Using Both Experimental and Numerical Inputs," ASME J. Electronic Packaging, Vol. 126, No. 4, pp. 465–471.
- **Chen, K. N., 2006,** "Optimal Support Locations for a Printed Circuit Board Loaded With Heavy Components," ASME J. Electronic Packaging, Vol. 128, DECEMBER, pp. 449–455.
- **Lee, S., Culham, J. R., and Yovanovich, M. M., 1991,** "The Effect of Common Design Parameters on the Thermal Performance of Microelectronic Equipment: Part II – Forced Convection", ASME, Heat Transfer in Electronic Equipment, HTD-Vol. 171, pp. 55-62.
- **Lee, S., Culham, J. R., and Yovanovich, M. M., 1991,** "The Effect of Common Design Parameters on the Thermal Performance of Microelectronic

Equipment: Part I – Natural Convection", ASME, Heat Transfer in Electronic Equipment, HTD-Vol. 171, pp. 47-54.

- **Lee, S., Culham, J. R., Lemczyk, T. F., and Yovanovich, M. M., 1990,** " META- A Conjugate Heat Transfer Model for Air Cooling of Circuit Boards with Arbitrarily Located Heat Sources", 1990, ASME, HTD, Heat Transfer in Electronic Equipment, Vol. 171, pp.117-126.
- **Lee, S., Culham, J. R., Jeakins, W. D., and Yovanovich, M. M., 1992,** "Thermal Simulation of Electronic Systems with Non-Uniform Inlet Velocities", ASME, Computer Aided Design in Electronic Packaging, EEP-Vol. 3, pp. 33-40.
- **Afrid, M., and Zebib, A., 1989,** " Natural Convection Air Cooling of Heated Components Mounted on a Vertical Wall", J. Numerical Heat Transfer, part A, Vol. 15, pp. 243-259.
- **Mahaney, H. V., Ramadhyani, S., and Incropera, F. P., 1989,** " Numerical Simulation of Three-Dimensional Mixed Convection Heat Transfer from an Array of Discrete Heat Sources in a Horizontal Rectangular Duct ", J. Numerical Heat Transfer, part A, vol. 16, pp. 267-286.
- **Mahaney, H. V., Incropera, F. P., and Ramadhyani, S., 1990,** " Comparison of Predicted and Measured Mixed Convection Heat Transfer from an Array of Discrete Sources in a Horizontal Rectangular Channel "Int. J. Heat Mass Transfer, Vol. 33, No. 6, pp. 1233-1245.
- **Awbi, H. B., 1998** " Ventilation of Buildings ", by E & FN Spot.
- **Versteeg, H. K., and Malalasekera, W., 1995,** " An Introduction to Computational Fluid Dynamics: The Finite Volume Method ", Longman Scientific & Technical (Longman Group Limited).
- **Lundar, B. E., and Spalding, D. B., 1972,** "Mathematical Models of Turbulence ", Academic Press., London.
- **Davidson, L., and Farhanieh, B., 1995,** "A Finite-Volume Code Employing Collocated Variable Arrangement and Cartesian Velocity Components for Computation of Fluid Flow and Heat Transfer in Complex Three-Dimensional Geometries ", Dept. of Thermo- and Fluid Dynamics, Chalmers University of Technology, Sweden, November, Publ. No. 95/11.
- **Schlichting, H., 1995,** "Boundary Layer Theory ", seventh edition, McGraw-Hill.

Nomenclature

Ar	Archimedes number (Gr / Re^2)
C_p	Specific heat (*J/kg.K)*
C_1, C_2, C_D, C_μ	Coefficient in turbulence models
g	Gravitational acceleration *(m/sec^2)*
Gr	Grashof number
G_B	Buoyancy production
G_k	Generation rate of turbulence energy
H	Enclosure height (*m)*
h	Convective heat transfer coefficient (*W/m^2.K)*
I_u	Turbulence intensity
k	Turbulent kinetic energy *(m/sec^2)*
P	Pressure (*N//m^2)*
Q	Power dissipation from heat element (*W)*
q_w	Wall heat flux (*W/m^2)*
Re	Reynolds number

S_φ	General source term
T	Temperature ($^\circ C$)
T_r, T_w, T_{sk}	Reference , wall and surrounding temperature respectively ($^\circ C$)
U, V, W	Mean velocity components *(m/sec)*
u, v, w	Component of velocity vector in x, y, and z direction *(m/sec)*
u_τ	Friction velocity *(m/sec)*
x, y, z	Physical or Cartesian coordinates (*m*)

Greek symbol

Γ	Diffusion coefficient
ε	Energy dissipation *(m^2/sec^3)*
$\Re$	Constant (0.005)
κ_1	Von karmen constant (0.41)
μ	Dynamic viscosity *($N/sec.m^2$)*
ν	Kinematics viscosity *(m^2/sec)*
ρ	Fluid density (*kg/m^3*)
$\sigma 1$	Stefan-Boltzmann constant ($5.67x10^{-8}$) ($W/m^2.K^4$)
σ	Prandtl / Schmidt number
τ	Shear stress (N/m^2)
φ	Dependent variables

Subscript

e	Effective condition
in	Inlet condition
out	Outlet condition
r	Reference value
t	Turbulent
p	t. Point near the wall
w	Condition at wall
e	Effective condition

بحث ظاهرة انتقال الحرارة وتصرف الجريان حول جزء إلكتروني

د. ستار جابر حبيب
قسم هندسة المكائن والمعدات
الجامعة التكنولوجية

الخلاصة:

تمّ عرض دراسة نظرية لجريانات طباقية واضطرابية ثلاثية الأبعاد حول مصدر تسخين (جزء إلكتروني) موضوع فوق صفيحة إلكترونية PCB ، تضمنت الدراسة حل المعادلات التفاضلية الجزئية الأهليليجية المتمثلة بحفظ الكتلة، الزخم، الطاقة ، الطاقة المضطربة ومعدل ضياعها باستخدام الحجوم المحددة (Finite Volumes). لقد حلت هذه المعادلات باستخدام نموذج k-ε بالإضافة إلى استخدام مفهوم دالة الجدار بالقرب من الجدران لمعالجة تأثيرات الاضطراب. كما تمّ الاعتماد على الخوارزمية (SIMPLE algorithm) في بناء هيكلية البرنامج . الجزء إلكتروني يبرد بواسطة هواء خارجي يمر خلال الحيز. أهداف البحث تتمثل في بحث ظاهرة انتقال الحرارة للجزء الإلكتروني الموضوع داخل الحيز وكيفية الوصول إلى افضل مستوى تبريد له, هذه المتغيرات سوف تساعدنا على تصور الأداء الحراري للجزء الإلكتروني وشكل تيارات الجريان المتكون من خلال فهم تأثير معاملات مختلفة على شكل الجريان. النتائج وضحت العلاقة بين معدل ارتفاع درجة الحرارة داخل الحيز ومعاملات انتقال الحرارة (Nu, Ra) مع (Ar, Q)لحالتين من الجريان الطباقي والاضطرابي.