Convective Cooling of Volumetrically Heated Block in A Channel

Dr. Elhassen A. Omera[1] , Dr. Khalifa M. ALraqiq[2]
1 Dept. of Mechanical Engineering,
2 Dept. of Aeronautical Engineering
Zawia University

Abstract:

In the present study, a laminar forced convection cooling process of volumetrically heated block mounted in a lower wall on a parallel plate channel investigated numerically. A two dimensional conjugate heat transfer model with appropriate boundary conditions is used. The governing equations, that is, the conservation of mass, momentum and energy equations of the cooling medium are solved by vorticity-stream function approach coupled with the energy equation within the block via interfacing conditions. Calculations are made for distinct values of Reynold number, the ratio (soild/air) of thermal conductivities and other *geometrical parameters (variations in block height, width and size) in order to examine the influence of such variables on the convection cooling process of a heat generating block inside a channel.*

***Keywords**: Forced convection, Cooling, Electronic components, Channel flow.*

1. **Introduction**:

A problem of forced convection in a channel flow with heated blocks is of practical importance and widely considered in the design of devices such as heat exchangers, turbine blades, and electronic components. Therefore, there is a need for improving heat transfer performance of the heated blocks set in the channel. The thermal power density of compact electronic components is very high requiring efficient cooling system to maintain the temperature of these components in desired levels. There is currently a consensus that air cooling of electronics is a widely used technology [1,2]. There is a need of high confidence thermal models to accurately predict the temperature of components mounted on circuit cards. Models for the cooling of electronic components have been widely studied both numerically and experimentally. In most studies the problem is idealized as the fluid flow and thermal analysis of heat generation blocks within parallel plates.

Several studies [3–9] had investigated the heat transfer and flow characteristics in a channel with heated blocks. Ortega et al. [3], studied a wide variety of the cooling problems of electronic components. These include two- and three-dimensional systems in laminar or turbulent flow with natural, mixed, or forced convection. Davalath and Bayazitoglu [4], sloved numerically utilizing control volume formulation the two dimensional laminar flow over an array of blocks with uniform conductivity and heat generation between parallel plates. Nigen and Amon [5], employed the spectral element method to study the conjugate heat transport from a single block with uniform or local heat generation within parallel plates with different block conductivities. Young and Vafai [6, 7] investigated the effect of the fluid flow and heat transfer in a two-dimensional channel which contains a heated block or multiple heated blocks. The effects of block geometry and number on heat transfer studies extensively. They have shown that the more the blocks are smaller and widely spaced, the more the heat transfer is enhanced, and the narrow gaps between blocks allow upstream thermal transport. Furukawa and Yang [8] numerically studied a periodically fully developed flow in a ribbed channel with a special emphasis to the thermal contact resistance between the chip and the board which has a considerable impact on thermal enhancement.

Jubran et al. [9] reported in their experimental investigations the effect of cubical and non-cubical blocks with different lengths, widths and heights on the pressure drop and heat transfer enhancement of cooling process of various arrays configurations of electronic components. Gupta et al. [10] presents the heat transfer and fluid flow in a channel using an inclined block as an obstacle. The heat transfer augmentation with the inclined block is compared with a plane channel at same Reynolds number. The fluid flow is steady, laminar and incompressible. The governing equations are solved using finite volume method. By using inclined block between channels the heat transfer is augmented considerably.

In this study, a numerical analysis was carried out to investigate the flow field and heat transfer characteristics of the cooling process of a volumetrically heated block subjected to steady laminar channel flow. Detailed numerical results are obtained to describe the effects of various governing parameters defined in the problem.

2. Analysis

The schematic shape of the system under consideration is shown in Fig. (1). The governing equations for the problem under consideration are based on the balance laws of mass, linear momentum and thermal energy in two dimensions.

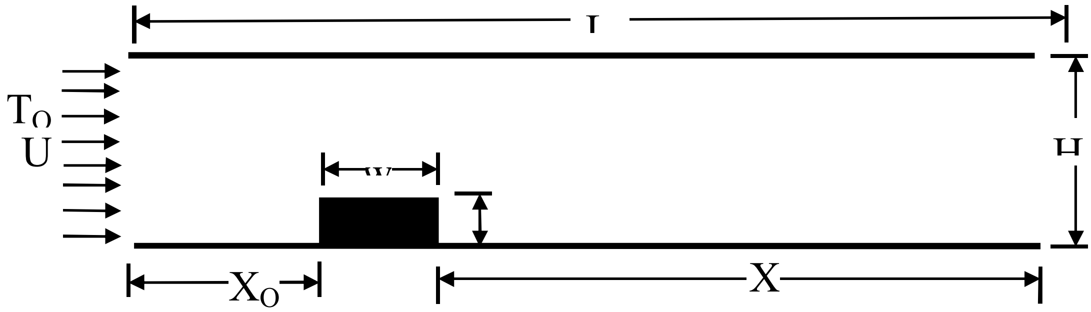


Fig. (1) The physical domain of the problem

Let the flow through the channel be two-dimensional, steady and laminar, It enters the channel, at y = 0, with uniform velocity V and temperature T_O, u and v are the velocity components in the x- and y-direction, respectively and T is the temperature at location(x,y), and an assumption is imposed that all thermal properties of flow and solidity, including density, viscosity, and conductivities, are constant with neglect of viscous dissipation and gravity force, the governing equations are[11]:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=\frac{-1}{\rho_f}\frac{dP}{dx}+\upsilon_f\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=\frac{-1}{\rho_f}\frac{dP}{dy}+\upsilon_f\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \tag{3}$$

$$u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=\alpha_f\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right) \tag{4}$$

The thermal energy release in an electronic component can be approximated as a constant surface heat flux or as volumetric generation. The energy equation for the solid phase, accounting for a volumetric source term, is

$$\frac{k_s}{k_f}\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right)+\ q'''\ =0 \tag{5}$$

Where : q''' is the volumetric heat generation

The flow equations are solved using vorticity stream function approach described by Roache [12]. By cross differentiation of the momentum equations for eliminating the pressure gradient terms and utilizing the stream function and vorticity definitions :

$$u=\frac{\partial\psi}{\partial y}\ ,v=-\frac{\partial\psi}{\partial x}\quad and\quad \omega=\frac{\partial v}{\partial x}-\frac{\partial u}{\partial y} \tag{6}$$

Then substituting of eq. (6) on the result of momentum differentiations, the continuity and momentum equations eqs. (1-3) replaced by :

$$\frac{\partial^2\psi}{\partial X^2}+\frac{\partial^2\psi}{\partial Y^2}=-\omega \tag{7}$$

$$\frac{\partial\psi}{\partial y}\frac{\partial\omega}{\partial x}+\frac{\partial\psi}{\partial x}\frac{\partial\omega}{\partial y}=\upsilon\left(\frac{\partial^2\omega}{\partial x}+\frac{\partial^2\omega}{\partial y^2}\right) \tag{8}$$

The governing equations are non-dimensionalized using the following dimensionless variables:

$$\left.\begin{array}{l} X=\frac{x}{H},\quad Y=\frac{y}{H},\quad U=\frac{u}{U_o},\quad V=\frac{v}{U_o} \\ R_k=\frac{k_s}{k_f}\quad Re_H=\frac{U_oH}{\nu}\ and\ \ Pr=\frac{\nu}{\alpha} \\ \Psi=\frac{\psi}{U_oH},\quad \Omega=\frac{\omega U_o}{H}\quad \theta=\frac{T-T_o}{q\,H/k_f} \end{array}\right\} \tag{9}$$

By employing eq. (9), the resulting dimensionless equations can be written as

$$\frac{\partial^2\Psi}{\partial X^2}+\frac{\partial^2\Psi}{\partial Y^2}=-\Omega \tag{10}$$

$$\frac{\partial\Psi}{\partial Y}\frac{\partial\Omega}{\partial X}+\frac{\partial\Psi}{\partial X}\frac{\partial\Omega}{\partial Y}=\frac{1}{Re_H}\left(\frac{\partial^2\Omega}{\partial X^2}+\frac{\partial^2\Omega}{\partial Y^2}\right) \tag{11}$$

$$\frac{\partial\Psi}{\partial Y}\frac{\partial\theta_f}{\partial X}+\frac{\partial\Psi}{\partial X}\frac{\partial\theta_f}{\partial Y}=\frac{1}{Re_H\,Pr}\left(\frac{\partial^2\theta_f}{\partial X^2}+\frac{\partial^2\theta_f}{\partial Y^2}\right) \tag{12}$$

$$\frac{\partial^2\theta_s}{\partial X^2}+\frac{\partial^2\theta_s}{\partial Y^2}=-\frac{1}{R_{k\,h}} \tag{13}$$

The heat transfer coefficient is determined as the local Nusselt number along the surface of the block which is defined as

$$Nu_x=-\frac{1}{\theta_w}\frac{\partial\theta_f}{\partial n} \tag{14}$$

It should be pointed out that the mean temperature of the fluid does not appear because it was assumed to be too small in comparison with the wall temperature, due to heat generation in the block [4]. The mean Nusselt number for a block is evaluated by:

$$\overline{Nu}_{surface}=\frac{1}{X_S}\int_{surface}Nu_n\,dn \tag{15}$$

where X_s is length of heated surface for each face or the total surface exposed to the flow.

3. Boundary conditions:

Boundary conditions along the entire solution domain must be specified for all field variables due to the elliptic nature of the governing conservation equations. The fluid, at the entrance, is assumed to be at the ambient temperature, and entering the channel with a uniform velocity profile. At the outlet the stream wise gradients of temperature and velocity components are assumed to be zero. The channel walls are impermeable and no slip conditions must be satisfied, also adiabatic walls condition is adopted except at the block location. At the fluid-solid interfaces, the no-slip condition and the continuities of temperature and heat flux are taken into account. In summary, the boundary conditions are described in the following dimensionless form:

At inlet : for X = 0 , 0 ≤ Y ≤ 1 :

$$U = 1.0\,,\ V = 0.0\,,\ \Omega = 0 \ \ and \ \ \theta_f = 0.0 \qquad (16)$$

At outlet : for X = L , 0 ≤ Y ≤ 1 :

$$\frac{\partial \theta_f}{\partial X} = \frac{\partial \Omega}{\partial X} = \frac{\partial \Psi}{\partial X} = 0.0 \qquad (17)$$

At the walls : Y = 0 and 1 , and 0 ≤ X ≤ L :

$$U = 0.0\,,\ V = 0.0\,,\ \Omega = -2\frac{\partial^2 \Psi}{\partial n^2} \ \ and \ \ \frac{\partial \theta_f}{\partial Y} = 0.0 \qquad (18)$$

At the block walls :

$$\theta_f = \theta_s \ \ \text{and} \ \ k_f\left(\frac{\partial \theta_f}{\partial n}\right)_{Fluid\ side} = k_s\left(\frac{\partial \theta_s}{\partial n}\right)_{Solid\ side} \qquad (19)$$

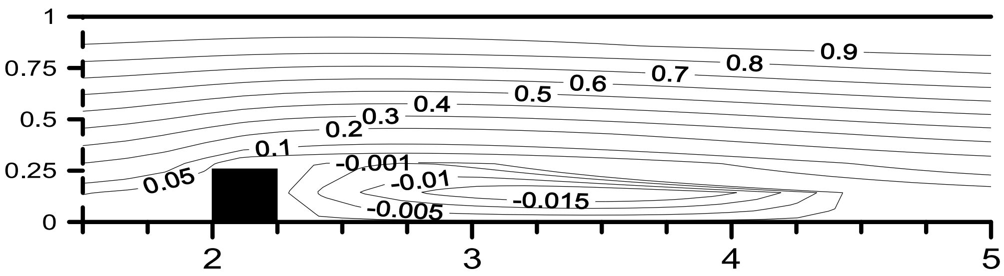
4. Numerical solution :

The governing equations along with the boundary conditions are solved numerically employing finite difference techniques where the diffusive terms are discretized by using central differencing while the use of upwind differencing is preferred for convective terms for numerical stability. The vorticity transport and energy equation are solved using the alternating direction implicit (ADI) method with tri-diagonal matrix algorithm (TDMA) [12]. The stream function equation is solved by SOR "successive over relaxation method. The relaxation parameters are chosen to be 1.2 and 0.5 for stream function and vorticity equations to avoid divergence and instability in the solution [13]. The convergence criterion was satisfied when

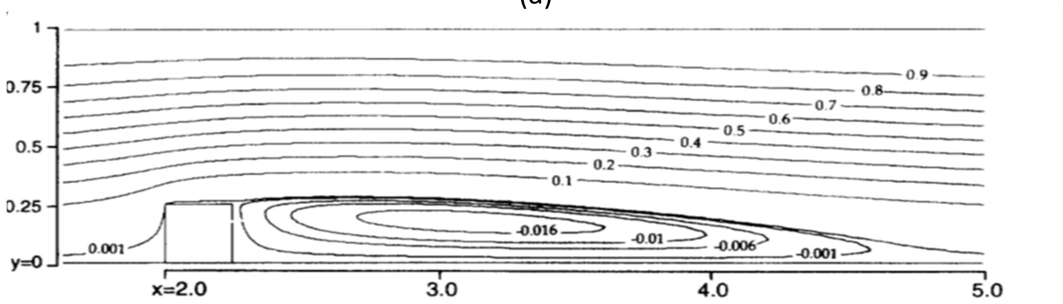
$$R = \sum_{\substack{i=1,..,M \\ j=1,..,N}} \frac{\left|p_{i,j}^{k+1} - p_{i,j}^{k}\right|}{\left|p_{i,j}^{k}\right|} \le \varepsilon \qquad (20)$$

Where: p represents the property Ω, ψ *and* θ, the superscript k refers to the iteration number and the subscripts i and j refer to the space coordinates. The value chosen for ε was 10^{-5}, for all calculations.

A grid independence study is conducted using different grid sizes of 21x21, 51x51, 51x101 and 101x101 for aspect ratio = 1.0 and it is observed that a further refinement of grids from 51x101 to 101x101 does not have a significant effect on the results in terms of overall mean Nusselt number value. Based on this observation, a uniform grid of 51x101 points is used for all of the calculations. A validation test carried out by comparing the results of cooling process of a heated block of the current study and Young and Vafai [7], with the same tested data where the block geometry is (W = h = 0.25) and Re_{Dh} = 1000 as shown in Fig. (2).



(a)



(b)

Fig. (2) Streamlines of present study Young and Vafai [7]

5. Results:

The dimensionless parameters that specify this system include the hydraulic diameter **(Dh = 2H)** based Reynolds number, block thermal conductivity ratio $R_k=k_s/k_f$, and block height and width. The fixed input parameters utilized in this work are q"= 1 , H= 1, X_o = 4.0 it is recommended to avoid entrance effects [6], X_t = 10 chosen long enough to assure outlet conditions are satisfied and Pr=0.72 for air as a working medium. The range of Reynolds numbers in this investigation, $100 < Re_{Dh} < 1500$, was chosen such that laminar conditions were maintained. This range of values is typical of laminar forced convective cooling of electronic systems, where the inlet velocity may range from 0.3 to 5 ms^{-1} [14]. The thermal conductivity was varied from R_k = 10, 100 and 1000, values typical of materials utilized in electronic packaging, such as epoxy glass, ceramics and heat spreaders [2].

The block geometries were parametrically varied to evaluate the effects of systematic changes. Relative to the unity channel spacing, the block geometric variations are as follows : W = 0.125 - 0..5 and ***h*** = 0.125 - 0.25. In order to illustrate the results of the flow and temperature fields only near the block region and its vicinity is focused upon. However, it should be noted that the computational domain included a much larger region than what is displayed.

5.1. Effects of Reynolds number:

The Reynolds number is an important parameter in determining the different states of flow. For such analysis, results are obtained for the case with ***Pr = 0.7,*** R_k = 10, and Re_{Dh}, = 100 to 1500, with block dimension as h=0.125 and W=0.5. Fig. (3) shows the streamline plots for different values of the Reynolds number. The presence of the block causes the flow to turn upwards and accelerate into the bypass region

(vena contracta). The stream lines close to the solid body reveal three different circulation zones. These are the stagnation flow (leading surface of the block), the flow spilling from the front top corner of the solid body, and the wake formation (trailing surface of the block) regions. These vortices strength slightly increases and occupies slightly more area as Re_{Dh} increases. The downstream recirculation zone beyond the block is more pronounced and expands axially and gains strength as Re_{Dh} increases.

Fig. (4) shows isotherms for different values of Re number. It is clear that the temperature field is strongly affected by the variation of Re. The temperature attains higher values close to the solid body. The high-temperature attainment is mainly due to the boundary layer development and circulation formation in these regions. Because, the solid body is bluff, the stagnation flow is generated in front of the body which in turn decelerates the convective cooling of the object. Moreover, the circulation generated at the rear side of the solid body also reduces the convective heat transfer from the solid body. The temperature contours in the region close to the top surface of the solid body extend further into the fluid as the Re decreases, this indicates that the size of the thermal boundary layer is considerable and the temperature gradient is less in this region. Fig. (5) shows the isotherms contours within the block for different Reynolds numbers, the surface temperature distributions and the maximum temperature within the block significantly lowered as the Reynolds number increased.

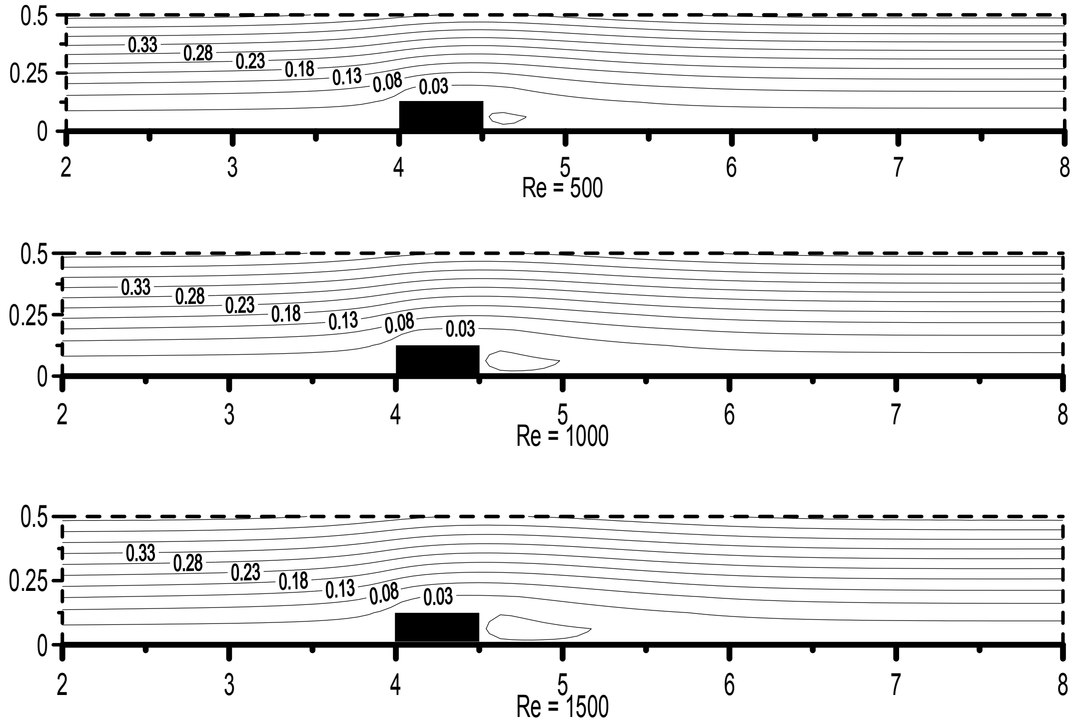


Fig. (3) Streamlines at different Re numbers

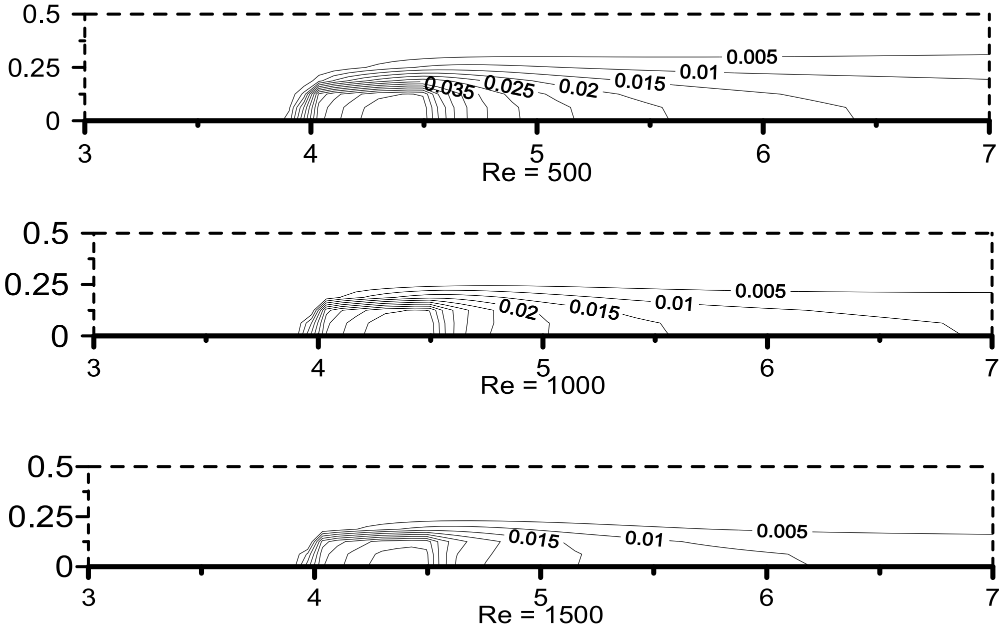


Fig. (4) Isotherms at different Re numbers

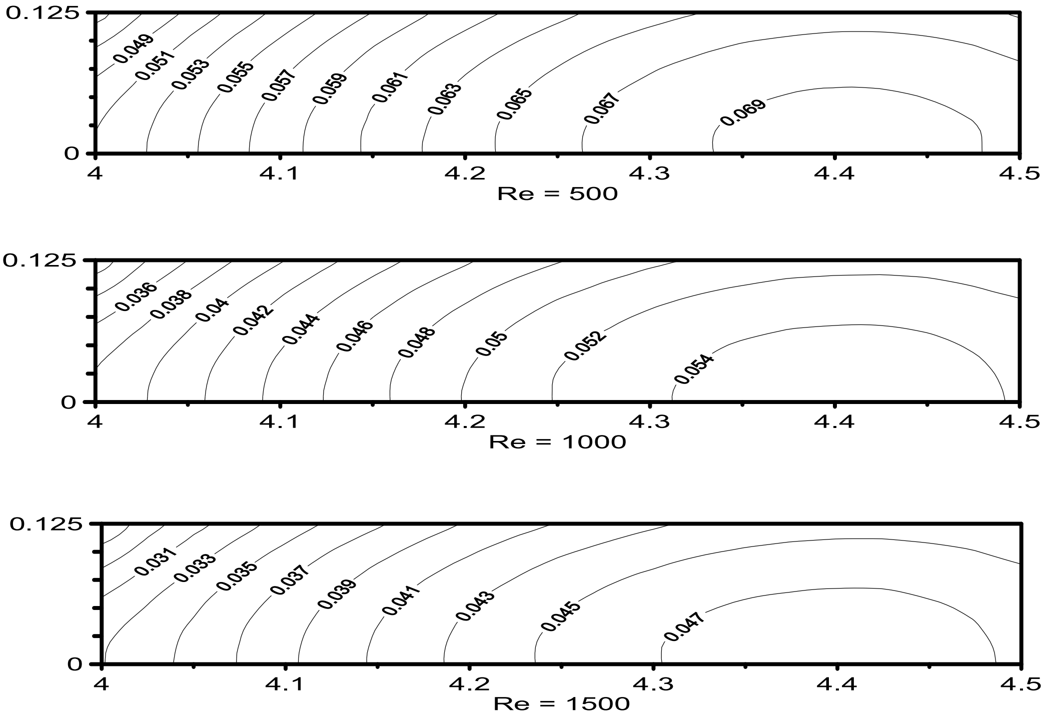


Fig. (5) Isotherms within the block at different Re numbers

Fig. (6) shows the local Nusselt number distributions around the exposed faces of the heated block. The heat exchange is strongly influenced by Re. By increasing Re number the flow becomes less stable and intense recirculation zones appear around the block. From this figure, we note that for the block along the front face (0 to 0.125) the Nusselt number increases rapidly and reaches a local maximum at the upper corner and this can be explained as due to the increase in upstream momentum causes the heat transfer coefficient to increase rapidly towards the upper corner. Along the top surface (0.125 to 0.625) the Nusselt number drops as the thermal boundary layer grows. Also, the magnitude of temperature gradient increases at the top face as the Re number increases since greater flow rates reduce the thickness of the thermal boundary layer. The convection from the back surface (0.625 to 0.75) is somewhat unaffected compared to the other two surfaces due to the circulation zone formed behind the back surface, which acts as an insulation layer.

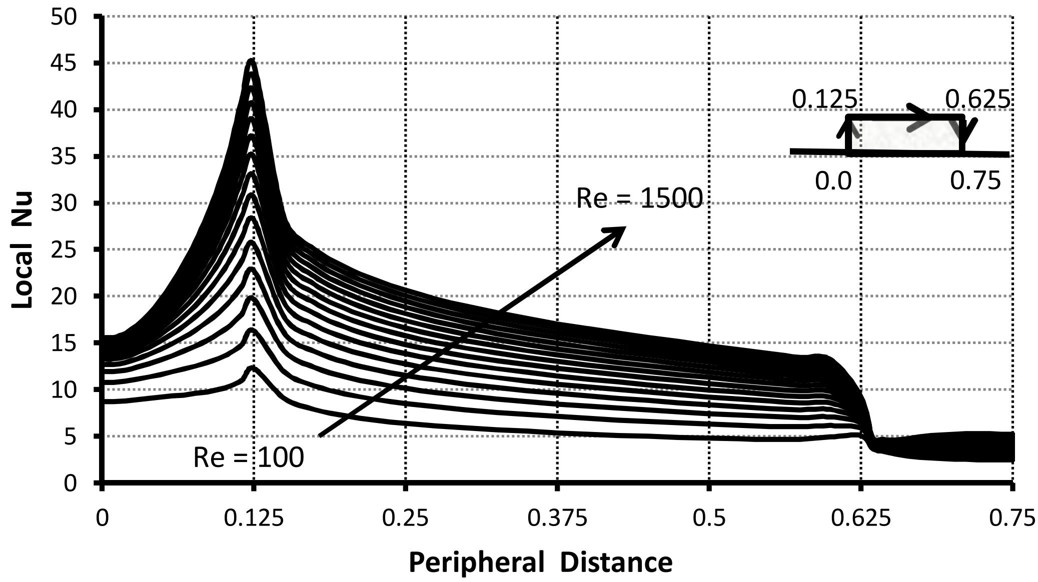


Fig. (6) Local Nu along the periphery of the block faces exposed to the flow

5.2. Effects of block geometry:

The geometries of the block investigated can be organized into three groups based upon block width, height and size as detailed in table (1).

Table (1) Cases classification based on geometry parameters

Case no.	h	W	Different . W Constant h	Different h Constant W	Different Size h/W =1
1	0.125	0.125	X	X	X
2	0.125	0.25	X		
3	0.125	0.5	X		
4	0.2	0.125		X	
5	0.25	0.125		X	
6	0.2	0.2			X
7	0.25	0.25			X

a) Block height:

The present study investigated the effect of the block height on the heat transfer around the block. The height, *h*/*H* varied between 0.125 - 0.25 cases (1, 4 and 5). The Reynolds number varied between 100 and 1500. The increase of h implies an increase of the exchange surfaces and an acceleration of the fluid since the flow domain is reduced and consequently, the heat transfer is more pronounced around the block. The recirculation zones formed behind the block increase in size whereas the block height increases as shown in Fig. (7) where the stream lines at Re = 1000 are plotted for different cases.

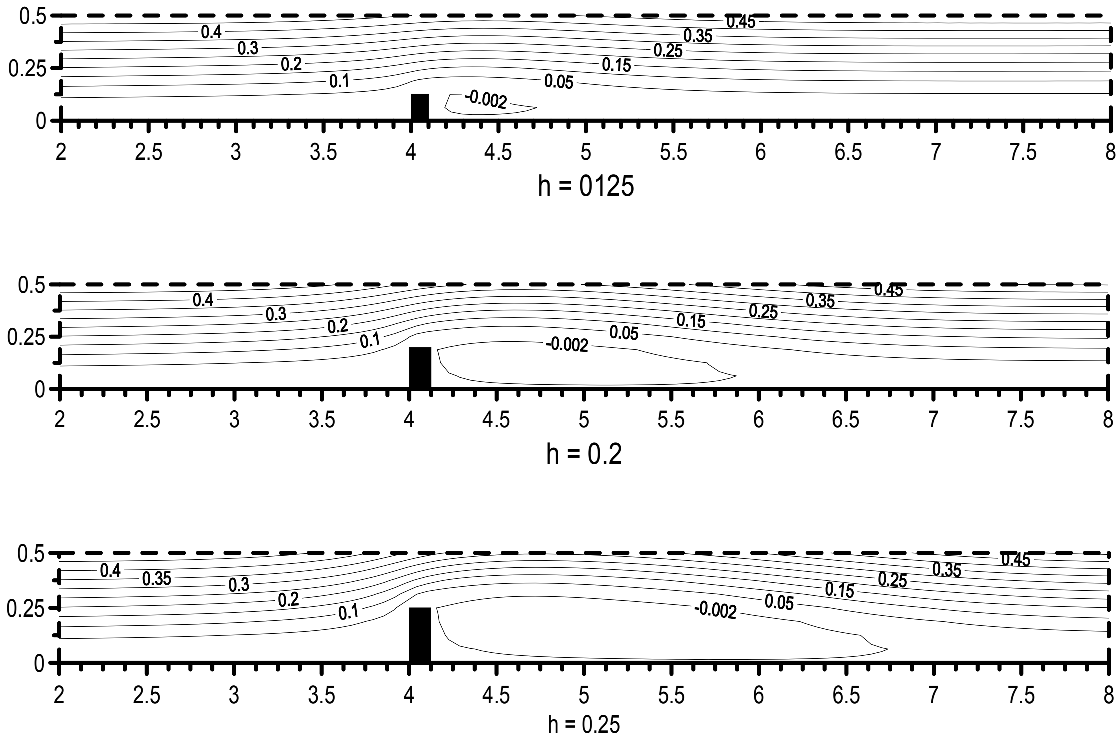


Fig. (7) Streamlines at different block heights and Re = 1000

Varying the block height changes the clearance height above the block which reported to be one of those geometric parameters which had a substantial effect on the Nusselt number [4 , 6]. The effect of this geometrical parameter on Nu_m is presented in Fig. (8). The shorter block results in a larger Nu_m, as the Reynolds number increases where, the influence of block height on Nu_m is amplified. In general, the shorter block height does allow better thermal transport into the cooler core flow of the cooling medium.

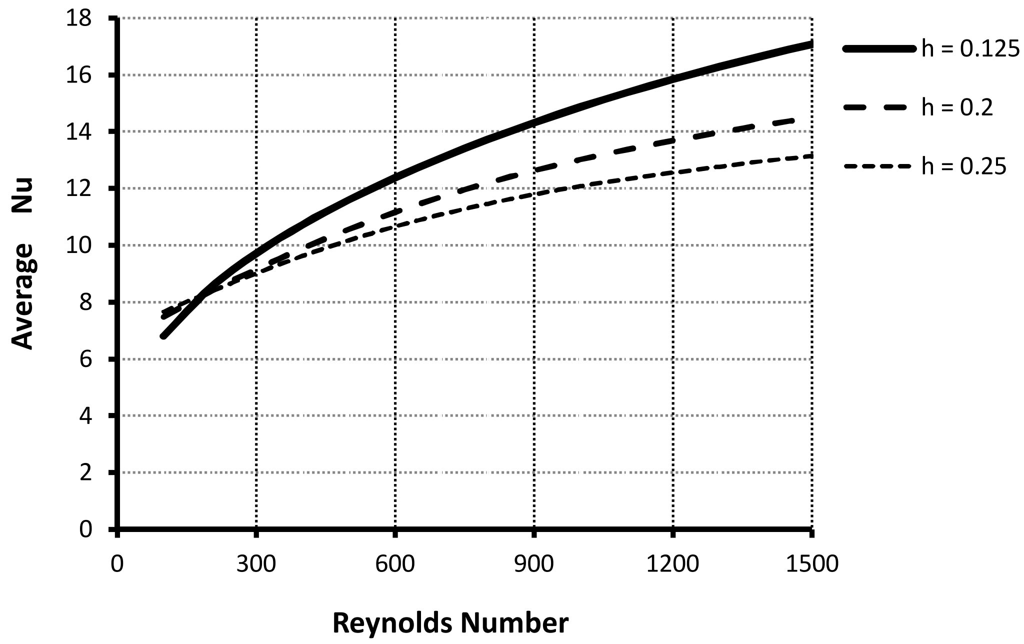


Fig. (8) The overall mean Nu at different heights

b) Block width:

The effect of varying block height is demonstrated by comparing cases (1, 2 and 3) with fixed height. Fig. (9) shows the temperature contours within the block with different block widths and Re_{Dh} = 1000, and it is clear that the block with smaller width cooled faster than the larger widths blocks.

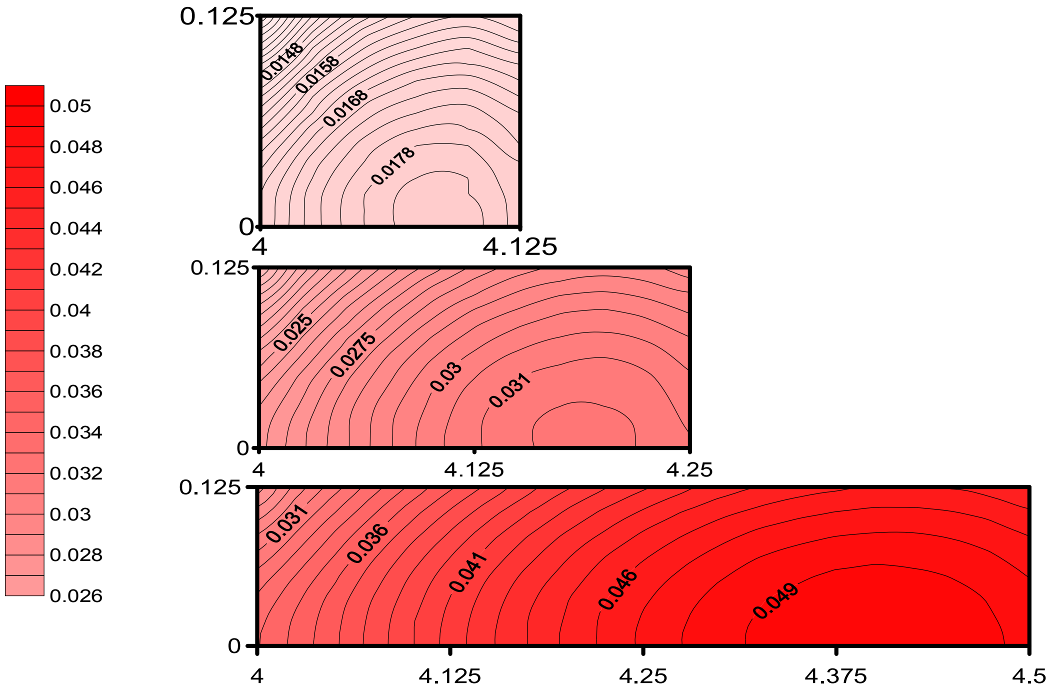


Fig. (9) Isotherms within the block at different widths and Re=1000

Fig. (10) shows the greater effect is in the top face mean Nu_T as the larger portion of the face is away from the entrance and experience lower values of local Nu. The value of the Nu_L (left face mean Nu) remains unaffected as the block width increases. The value of Nu_R decreases slightly with increasing width, but their small magnitudes produce small effects on the overall heat transfer. This decrease is due to the larger amount of thermal energy released by the wider block further heating the fluid and reducing the heat transfer. Since the Nu_m is an area weighted average of the three exposed faces of the flowing fluid the overall mean Nu changes slightly with increasing width as shown in Fig.(11).

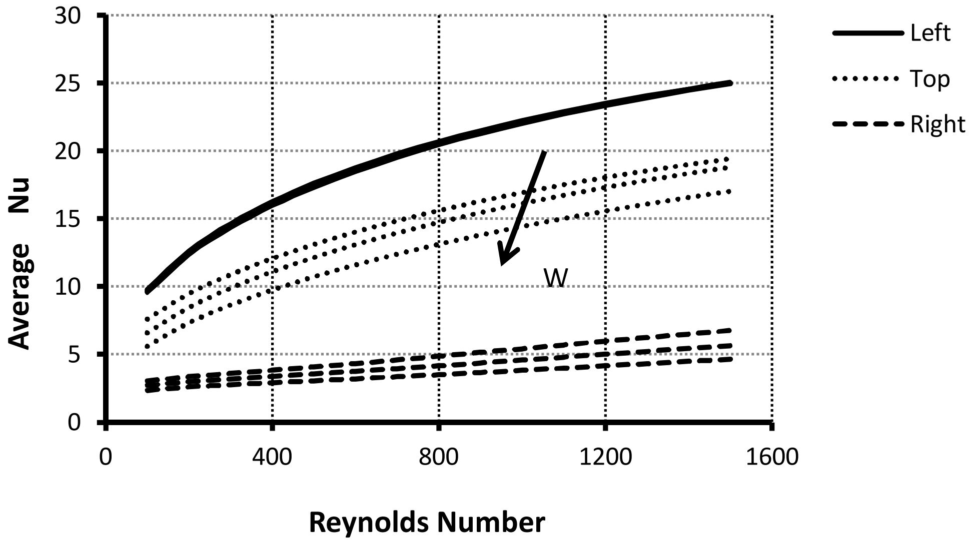


Fig. (10) Mean Nu for the block faces at different widths

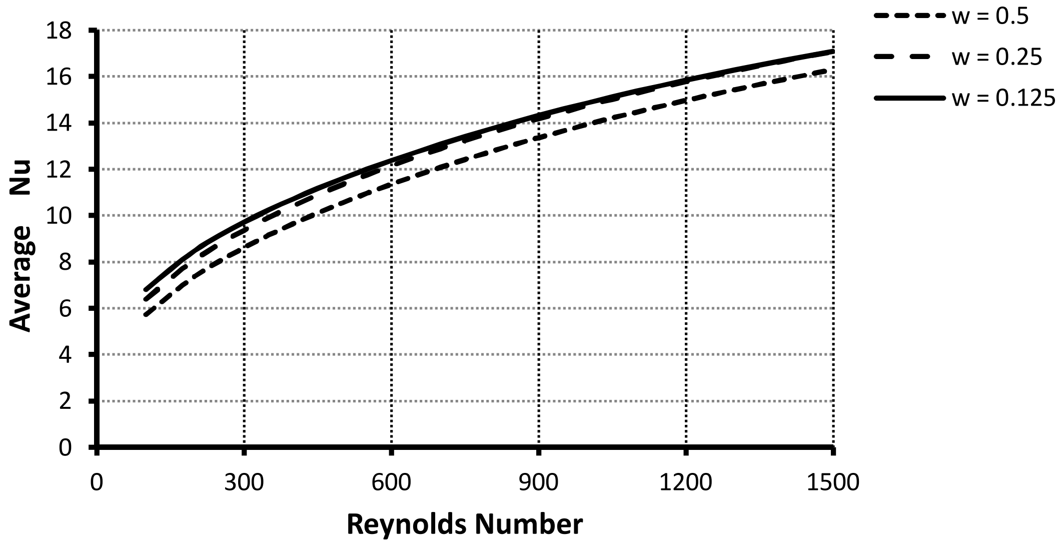


Fig. (11) The overall mean Nu at different widths

c) Block size:

Cases (1, 6 and 7) were compared to investigate the effects of an increase in block size. The block volume for Case 7 is greatest even though the aspect ratio for the three cases is ***w/h*** = 1. Fig. (12) reveals that the mean Nusselt numbers decreased with increased obstacle size. The smaller block introduce less thermal energy into the fluid, due to their smaller surface receiving the heat flux. The fluid remains cooler and thus, able to transport more thermal energy away from the block, as indicated by the greater Nusselt numbers. Also, temperature contours within the block for different block sizes at Re = 1000 shown in Fig. (13) confirm this result.

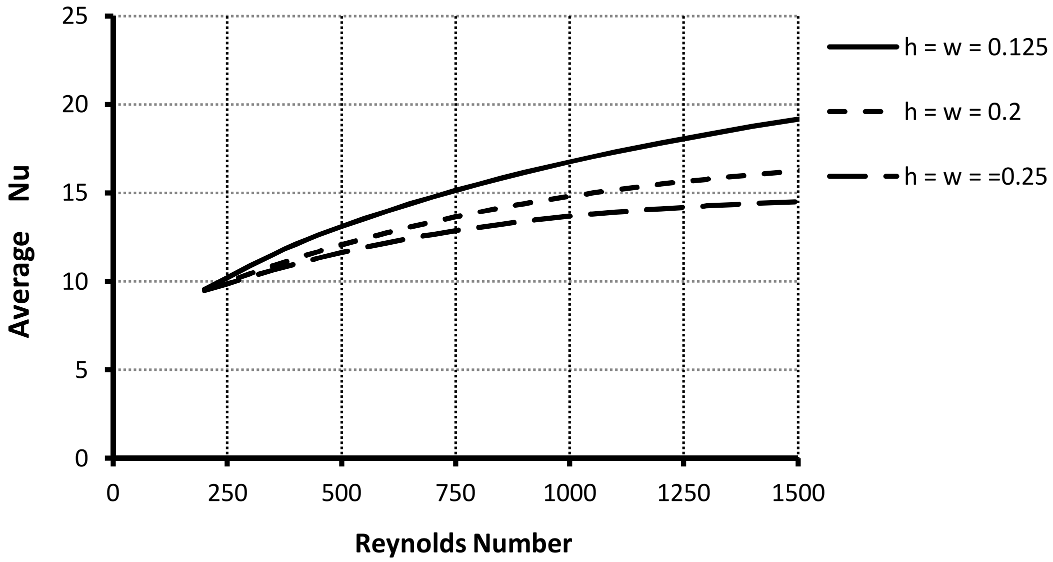


Fig. (12) The overall mean Nu at different blocks sizes

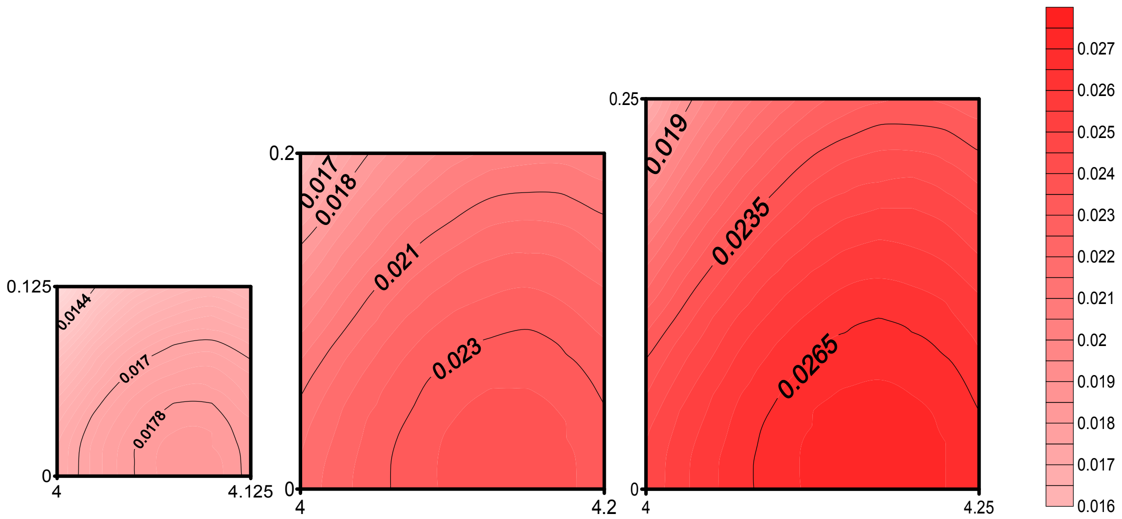


Fig. (13) Isotherms within the block at different block sizes and Re=1000

5.3. Effect of thermal conductivity ratio:

The solid thermal conductivity has a significant effect on the thermal transport within the block. Increasing the thermal conductivity reduces the temperatures and thermal gradients within the block by reducing the internal resistance to heat flow. When the thermal conductivity is larger than 10, the block become essentially isothermal and the Nu distributions become nearly identical. The temperatures along the exposed block surfaces for the case (h = w = 0.125) case, with Re_{Dh} = 1000 and R_k = 1, 10, and 100 are shown in Fig. (14). The block surfaces are nearly isothermal when R_k = 100, the wall temperatures for R_k = 10 are lower than for R_k = 1.0 at the lower corners (A and D), but near the block higher corners (B and C) the surface temperature for R_k = 10 is higher. The benefits of increasing block thermal conductivity are reducing the internal resistance and temperature gradient within the block and increasing surface isothermality.

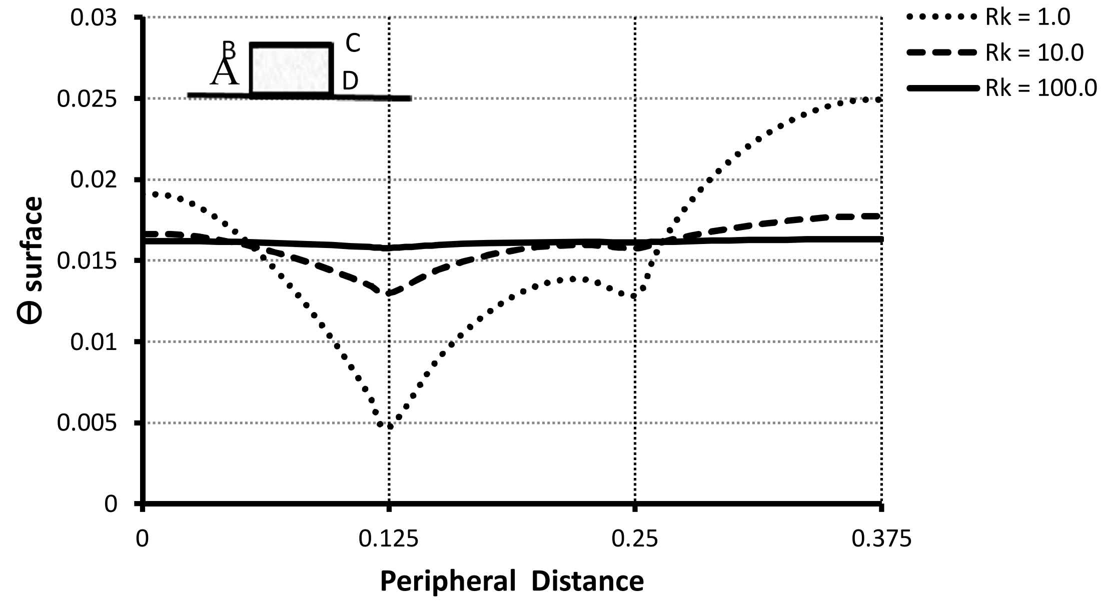


Fig. (14) Surface temperature distribution along the periphery of the block at different thermal conductivity ratios

6. Conclusions:

A two dimensional numerical simulation of the convective cooling of a heated block mounted on the lower wall of a horizontal channel with adiabatic walls was studied. The effect of in block geometry, Reynolds number and thermal conductivity ratio were investigated and documented. The results were expressed in terms of streamlines, isotherms and Nusselt number curves and reveal clearly the effect of the these parameters on the cooling process of a heated block. The following conclusions can be drawn from this work:

1. The flow structure is affected by the Reynolds number, where the zones of recirculations are generated around the block and intensified as the Re increases involving an important role of heat transfer.

2. The heat transfer rate is increased when the Reynolds number increases.
3. The heat transfer is clearly influenced by changing dimensions of the block particularly the height, whereas the height and width getting smaller the heat transfer process will be enhanced.
4. The more the blocks are smaller, the more the heat transfer is enhanced.
5. The results showed that the block material has a significant effect on heat transfer where large values of the solid thermal conductivity effectively isothermalize the block regardless of geometry.

References:

1. *Incropera FP, 1988 "Convection heat transfer in electronic equipment cooling", J. Heat Transfer 110 (1988) 1097– 1111.*
2. *Yeh LT 1995, "Review of heat transfer technologies in electronic equipment", Journal of Electronic Packaging 117, pp 333-339.*
3. *Ortega, A, Sugavanam R, and Choi, CY, 1995, "A Numerical Investigation of Conjugate Heat Transfer from a Flush Heat Source on a Conductive Board in Laminar Channel Flow", International Journal of Heat and Mass Transfer, Vol. 38, pp. 2969-2984.*
4. *Davalath J, Bayaszitoglu Y, 1987, "Forced convection cooling across rectangular blocks", J Heat Transfer;109:321-328.*
5. *Nigen JS, and Amon CH, 1995, "Effect of material composition and localized heat generation time dependent conjugate heat transfer", Int. J. of Heat and Mass Transfer, Vol. 38, pp. 1565-1576.*
6. *Young TJ, Vafai K., 1998, "Convective flow and heat transfer in a channel containing multiple heated obstacles", Int J Heat Mass Transfer vol. 41, pp 3279-3298.*

7. *Young TJ, Vafai K., 1998, "Convective cooling of a heated obstacle in a channel ", Int J Heat Mass Transfer vol. 41, pp 3131-3148.*
8. *Furukawa T and Yang WJ, 2003 "Thermal-Fluid Flow in Parallel Boards with Heat Generating Blocks", Int. J. Heat Mass Transfer, vol. 46, pp. 5005–5015.*
9. *Jubran BA., Swiety SA., and Hamdan MA., 1996 "Convective Heat Transfer and pressure drop characteristics of various array configurations to simulate the cooling of electronic modulesl", Int. J. Heat Mass Transfer, vol. 39, pp. 3519 –3529.*
10. *Gupta1, M. , Khodm U. and Kumar, S., 2011 "Heat Transfer Augmentation Using an Inclined Block in Laminar Channel Flow", Current Trends in Engineering Research Vol.1, No.1, pp. 105-108.*
11. *Kays, W. M. and Crawford, M.E., 1993, "Convective Heat and Mass Transfer", 3rd ed., McGraw-Hill, New York, USA.*
12. *Roache, P. J., 1976, "Computational Fluid Dynamics", Hermosa publishers, Albuquerque, New Mexico.*
13. *Omer, E. A. A., Ph D thesis, 2009, "Simulation of Drying Process of porous materials", Mansura University, Egypt.*
14. *Anderson, AM and Moffat RJ, 1992, "The adiabatic heat transfer coefficient and the superposition kernel function : Part l- Data for arrays of flat packs for different flow conditions", J Electronic Packaging 1992; 114: 14-21.*