International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 01, January-2017

Numerical Simulations of Mixed Convection from a Horizontal Channel with Radiating Heat Sources

Rahul. K. C
Department of Mechanical Engineering,
Kannur university
Vimal jyothi Engineering college, chemperi, Kannur, India

***Abstract* - The fluid flow in a channel containing heat sources has been of interest for several decades as the canonical model for electronic chip cooling. Both experimental and numerical methods have been employed to study a wide variety of problems. The powered components are generally idealized as quadrilateral obstacles mounted individually or in arrays to a channel wall with thermal energy transfer to the surroundings. Improved thermal design of electronic components is necessary to reduce hot spots, increase energy throughput, and reduce the failure rate, which is related to the device junction temperature. The convective and radiation heat transfer from the surfaces of the obstacles varies substantially with the geometrical configurations of the chips. The current study involves 2D simulation of mixed convection with surface radiation heat transfer in a channel containing heated obstacles using FLUENT. This study will detail the effects of variations in the obstacle height, width, spacing, and number, along with the Reynolds number, to illustrate important fundamental and practical results. The study would be useful in design of chips for commercial electronics and electrical systems which has efficient heat transfer characteristics.**

Keywords: Mixed convection; Surface radiation; Horizontal channel; Protruding heat sources

Nomenclature

1. Heat source height H
2. Heat source width W
3. Spacing between the heat sources, m d
4. modified Grashof number, based on volumetric heat generation Gr
5. thermal conductivity, W/mK k
6. width and height of the protruding heat source, respectively, m L_h,t_h
7. Peclet number based on S Pe_S
8. Prandtl number Pr
9. volumetric heat generation from the protruding heat sources ,W/m^3 q_v
10. Reynolds number Re
11. temperature at any location in the computational domain, K T
12. non-dimensional horizontal velocity U
13. non-dimensional vertical velocity V
14. non-dimensional horizontal and non-dimensional vertical distances X,Y

1. INTRODUCTION

Heat generation by virtue of resistance to electricity is a common phenomenon, encountered in many day to day cases such as electrical and electronic appliances. Though beneficial applications of this phenomenon are present, heat generation in electrical/electronic devices proves to be harmful to the efficiency as well as life of the component. In these cases, cooling of the heat generating component is a significant necessity. Appropriate cooling of the component can yield better performance for a longer period of time. For feasibility concerns over a wide range of applications, cooling by convection and radiation process will prove to yield better results in terms of investment, installation and maintenance. Improved thermal design of electronic components is necessary to reduce hot spots, increase energy throughput, and reduce the failure rate, which is related to the device junction temperature. The convective and radiation heat transfer from the surfaces of the obstacles varies substantially with the geometrical configurations. So this study would be useful in design of chips for commercial electronics and electrical systems which has efficient heat transfer characteristics.

The numerical simulation of forced convective, incompressible flow in a channel with an array of heated obstacles attached to one wall was numerically studied by Young'et al. [1]. This study details the effects of variations in the obstacle height, width, spacing, and number, along with the obstacle thermal conductivity, fluid flow rate, and heating method, to illustrate important fundamental and practical results. The periodicity of the mean Nusselt number is established, relative to the ninth obstacle, at the 5% and 10% difference levels (eighth and seventh obstacles, respectively). The periodic behavior of the velocity components and temperature distributions are also explicitly demonstrated for the array. Extensive presentation and evaluation of the mean Nusselt numbers for all obstacles within the array is fully documented.

In another study Young et al.[2] conducted an extensive investigation of the fluid flow and heat transfer in a parallel plate channel with a solid, conducting obstacle. Parametric numerical simulations have been performed to capture the fundamental and practical results. The rectangular obstacle changes the parabolic velocity field significantly, resulting in recirculation zones both up- and downstream and a thermal boundary layer along the top face. The dependence of flow and temperature fields on parametric changes in the governing parameters, Reynolds number, solid thermal conductivity, heating method, and two geometric parameters, is documented. The results of this investigation show that the shape and material of the obstacle has a significant effect on the fluid flow and heat transfer.

An experimental investigation of flow and heat transfer characteristics over blocked surfaces in laminar and turbulent flows were carried out by Yemenici, et al.[3].This researches showed that heat transfer enhancement on the blocked surface increased with block heights and become more pronounced in laminar than that of turbulent flows.

Mahmood Yaghoubi et al.[4] investigated numerically the Conjugate heat transfer for three-dimensional developing turbulent flows over an array of cubes in cross-stream direction, representing finite heat elements mounted over a surface. Temperature fields in the blocks and on their outer surfaces were obtained solving heat conduction equation. Finite volume procedure with appropriate boundary conditions is used to solve the coupling between the solid and fluid region. The heat transfer characteristics resulting from recirculating zone around the blocks are presented for a wide range of Reynolds numbers from 4.2 × 103 to 1 × 105 (for Pr = 0.7) and blockage ratios from 10 to 50%. This research results the overall convective heat transfer increases with increasing flow Reynolds number and blockage ratio. Average convection on the front surface is higher for all Reynolds numbers. However, convective heat transfer is low over the back surface for all Reynolds numbers.

Kim and Anand [5] carried out a numerical investigation of forced convection conjugate heat transfer from protruding heat sources, considering a uniform profile at the inlet. A periodic boundary condition was used at the outer surface of the substrate to simulate a large number of circuit boards arranged in the transverse direction.

Furukawa et al. [6] numerically investigate thermal-fluid flow behavior in a bundle of parallel boards with heat producing blocks. The system simulates cooling passages in a stack of electronic circuit boards with heat generating chips. At a low Reynolds number flow, a developing flow may achieve a fully developed flow state at certain block number from the entrance. Thermal conductivity of the board and thermal contact resistance between the chip and board has a considerable impact on thermal performance. The fluid flow and heat transfer performance in this channel flow is similar to that in ribbed channel flow

Davalath and Bayazitolgu [7] presented results of a numerical study of conjugate forced convection air-cooling of three protruding heat sources mounted on a horizontal channel. To study the effect of conduction in the substrate, both finite thick wall and adiabatic wall were considered as substrate. The effect of Reynolds number on the fluid flow and heat transfer was analyzed and the results were also presented for the effect of Prandtl number.

Liu and Phan-thien [8], numerically, investigated conjugate natural convection with surface radiation heat transfer in a differentially heated square cavity and a volumetric heat generating heat source placed at the middle of the cavity. They included the effect of shadowing while calculating the radiation heat transfer.

Dehghan and Behnia [9], numerically, investigated combined conduction and natural convection with surface radiation from a top open cavity with a discrete constant flux source mounted on the left sidewall of finite thickness. Flow visualization studies were also carried out. A parametric study was conducted for a fixed set of geometric parameters and a fixed thermal conductivity ratio.

Shiang-Wuu Perng a et al.[10] have conducted Numerical investigation of mixed convective heat transfer for unsteady turbulent flow over heated blocks in a horizontal channel. A rectangular turbulator was mounted in the channel to enhance heat transfer by means of an internal flow modification induced by vortex shedding in this study. The width-to height ratio of turbulator was changed (0.25, 0.5 and 1.0) with a constant Reynolds number (5000) under various Grashof numbers (0–5.0E+8) for the purpose of investigating the heat transfer performance. The results indicate that the turbulator mounted in cross-flow above an upstream block can effectively enhance the heat transfer performance of mixed convection in the horizontal channel.

Hamouche et al. [11], investigated numerically the laminar mixed-convection heat transfer to air from two identical protruding heat sources, which simulate electronic components, located in a two-dimensional horizontal channel. The finite volume method and the SIMPLER algorithm are used to solve the conservation equations of mass, momentum, and energy for mixed convection. Results show that the heat transfer increases remarkably for Pr=0.71 and 5≤Re≤30. It was also found that the increase of separation distance, the height and the width of the components has a considerable enhancement of the heat removal rate from the components, and therefore, on the improvement of the heat transfer inside the channel.

Premachandran et al. [12], have conducted a numerical investigation of conjugate convection with surface radiation from horizontal channels with protruding heat sources. The flow is assumed to be steady, laminar, incompressible, hydrodynamically and thermally developing. Air is considered as the working fluid. The geometric parameters such as spacing between the channel walls, size of the protruding heat sources, thickness of the substrate and the spacing between the heat sources are fixed. Results are presented to show the effect of parameters such as Re_S, Gr_S , k_p/k_f, k_s/k_f, e_p and e_s on the fluid flow and heat transfer. A correlation for the non-dimensional maximum temperature is also developed using the method of asymptotic expansions.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 01, January-2017

In most of the studies, forced convection air-cooling is used if the heat flux from the component is more than 1000 W/m^2. Even if the flow is a forced one, the effect of buoyancy is not negligible when the heat flux from the electronic chip is very high. Hence, the flow may be in the mixed convection regime. It is important to consider the effects of conjugateness and surface radiation when analyzing a mixed convection problem, to accurately predict the fluid flow and heat transfer characteristics, as for example the cooling of electronic components, when air is considered as a cooling medium. Based on the survey, it is clear that the analysis of mixed convection with surface radiation from a horizontal channel with protruding heat sources by varying the geometrical parameters of the component has not been conducted effectively.

The schematic view of the geometry considered in the present study is given in Fig. 2.1. Heat sources with a volumetric heat generation q_v are mounted on the bottom wall of the channel. The length and thickness of the top wall of the channel are same as that of the bottom wall. Here the material for channel and chip wall is considered as Aluminium and copper respectively. The thermal conductivity (k_p) and the emissivity (e_p) of the heat sources and those of the channel wall thermal conductivity (k_s) and emissivity (e_s) are different from each other. The inlet, exit faces and the outer surfaces of the channel walls are assumed to be adiabatic. The principal objective of the present work is to study the effect of geometric parameters. Hence, the thermal parameters are fixed throughout the parametric study. The dimensions considered in the present study are shown in Fig. Air is considered as a working fluid and is assumed to be a non-participating medium.

2.MATHEMATICAL MODEL

The analysis is made for steady, laminar, viscous, incompressible, hydro dynamically and thermally developing, Newtonian fluid flow through a two dimensional channel with a solid obstacle on the wall. The medium is assumed to have constant properties, outside of density, for which the Boussinesq approximation is assumed to hold good. The thermo physical properties of the fluid and solid phases are constant. Viscous heat dissipation in the fluid is assumed to be negligible compared with conduction and convection.

Based on the above mentioned assumptions, the non-dimensional form of the governing equations for the fluid side can be written as follows:

a. Baseline

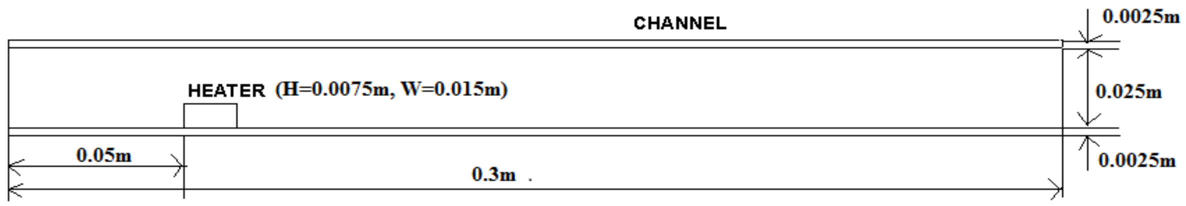


Fig.2.1.schematic of the computational domain (a)Baseline

Continuity

$$\frac{\partial U}{\partial X}+\frac{\partial V}{\partial Y}=0$$

X-momentum

$$\frac{\partial U}{\partial X}+U\frac{\partial U}{\partial X}+V\frac{\partial U}{\partial Y}=-\frac{\partial P}{\partial X}+\frac{1}{Re_s}\left(\frac{\partial^2 U}{\partial X^2}+\frac{\partial^2 U}{\partial Y^2}\right)$$

Y-momentum

$$\frac{\partial V}{\partial \tau}+U\frac{\partial V}{\partial X}+V\frac{\partial V}{\partial Y}=-\frac{\partial P}{\partial Y}+\frac{1}{Re_s}\left(\frac{\partial^2 V}{\partial X^2}+\frac{\partial^2 V}{\partial Y^2}\right)+\frac{{Gr_s}^*}{Re^2}\theta$$

Energy

$$\frac{\partial \theta}{\partial \tau}+U\frac{\partial \theta}{\partial X}+V\frac{\partial \theta}{\partial Y}=\frac{1}{Re_s Pr}\left(\frac{\partial^2 \theta}{\partial X^2}+\frac{\partial^2 \theta}{\partial Y^2}\right)$$

Energy equation for the solid region

Channel walls

$$Pe_s\left(\frac{\rho_s c_s}{\rho_f c_f}\right)\frac{\partial \theta}{\partial \tau}=\frac{k_s}{k_f}\left(\frac{\partial^2 \theta}{\partial X^2}+\frac{\partial^2 \theta}{\partial Y^2}\right)$$

Heat sources

$$Pe_s\left(\frac{\rho_p c_p}{\rho_f c_f}\right)\frac{\partial \theta}{\partial \tau}=\frac{k_p}{k_f}\left(\frac{\partial^2 \theta}{\partial X^2}+\frac{\partial^2 \theta}{\partial Y^2}\right)+\frac{S^2}{L_h t_h}$$

2.1 Boundary conditions

The computational domain along with the boundary conditions is shown in Fig.2.2. The inlet is specified as a velocity inlet .The outlet is a pressure outlet at zero gage pressure. Adiabatic boundary condition is imposed on the outer surfaces of the channel walls:

Inlet boundary conditions

The fluid is assumed to enter the channel with uniform velocity and temperature profiles, and the appropriate boundary conditions are

U = 1

V = 0

$\theta = 0$

Outlet boundary conditions

For the outlet of the computational domain, the following boundary conditions are used:

$$\frac{\partial^2 U}{\partial X^2} = 0 \; ; \; \frac{\partial^2 V}{\partial X^2} = 0 \; ; \; \frac{\partial^2 \theta}{\partial X^2} = 0$$

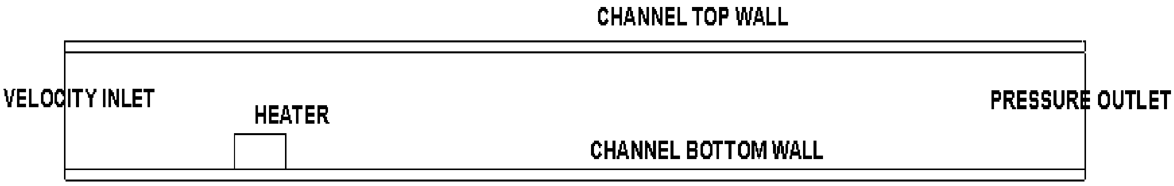


Fig.2.2. Computational domain and the boundary conditions

2.2 Validation

The numerical code was validated with the numerical results of B.Premachandran and C.Balaji. This was achieved through adjustments to the model to match the geometric, hydrodynamic, and thermal conditions of the previous study. Results were compared for the maximum temperature, T_{max} for Grashof number Gr=8.65×10^5, heat flux q_v=1×10^{5}w/m^3 Width w=0.015m, Height H=0.0075m, spacing d=0.010m, and Number of heat source N=4 for a wide range of Reynolds number. As shown in fig.2.3 it is clear that the numerical results of the present work are in a good agreement with B.Premachandran and C.Balaji predictions.

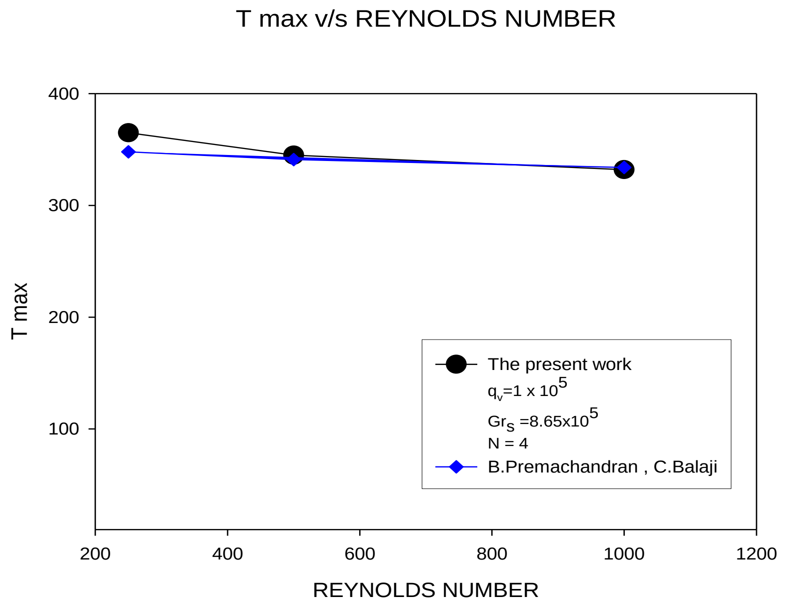


Fig.2.3. Comparison between the present predictions and B.Premachandran and C.Balaji predictions for Gr=8.65×10^5, q_v=1×10^{5}w/m^3, W=0.015m, H=0.0075m, d=0.010m, and N=4

3. RESULTS AND DISCUSSION

A detailed numerical study has been carried out on mixed convection with surface radiation heat transfer enhancement by using the commercial package ANSYS FLUENT based on finite volume method. The study deals with the effects of Reynolds number Re, parametric variations in the heat source height (H=0.0075m,0.015m), and width (w=0.015m,0.03m, 0.06m), separation distance between two heat sources (d=0.01m,0.02m, 0.03m) and number of heat sources on the flow structure and heat removal rate from the heat sources. In most practical applications of electronics systems cooling by air, the buoyancy strength is not very high and the maximum of the Grashof number is kept in the order of 8.65×10^5. The corresponding heat flux q_v=1×10^{5}w/m^3, Reynolds number, Re=500 for geometrical analysis, and channel dimensions (L=0.3m, H=0.03m, t=0.0025m) are taken as fixed baseline values in this study. Here the result outputs are discussed. Different temperature contours and graphs are presented in this chapter.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 01, January-2017

3.1 Grid independence study

In order to obtain grid independent solutions, a grid independence study was carried out. A non-uniform grid is used for throughout the domain. Very fine grids are used near the walls.

Table 3.1

Results of the grid independence study for Re=500, $Gr=8.65\times10^5$, $q_v=1\times10^5 w/m^3$, $W=15\times10^{-3}m$, $H=7.5\times10^{-3}m$

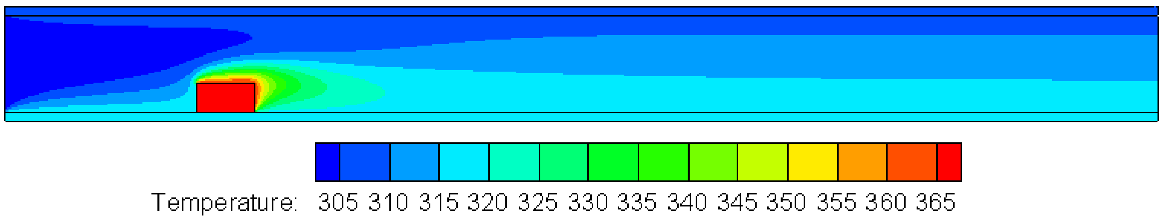
No.of nodal points	Average Nusselt number	Percentage change
183864	370.66	---
258330	369	0.45

The details of the results of the grid independence study are given in table 3.1. When the total number of grid points used for the computations increased from 183864 to 258330, a change of only 0.45% was observed on the average surface Nusselt number. Hence, a grid size of 183864 is used for the range of parameters considered in the present study.

3.2 Effect of Radiation

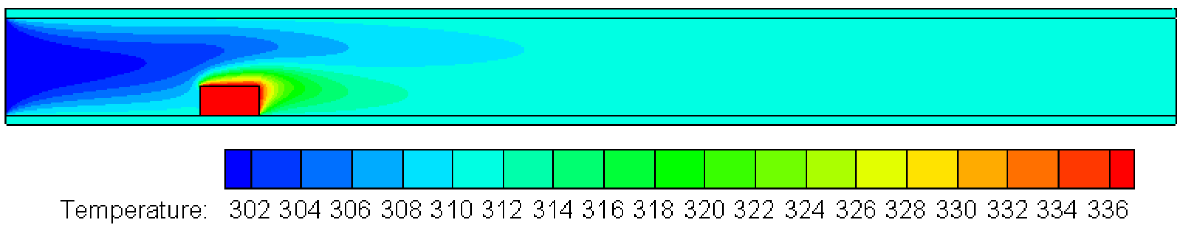
The effect of radiation on heat transfer rate is discussed here. Fig.3.1 shows the temperatur contours of two different cases. In fig.3.1(a) the temperature distribution without including the radiation effect is shown and fig.3.1(b) shows the temperature distribution by including the radiation effect also. The maximum temperature decreases from 365K to 336K when considering the radiation heat transfer. There is about 8.6% change in Tmax is obtained. So it is important to consider the effect of surface radiation when analysing a mixed convection problem to accurately predict the fluid flow and heat transfer characteristics, as for example the cooling of electronic components, when air is considered as a cooling medium.

T_{max}=365K



(a)

T_{max}=336K



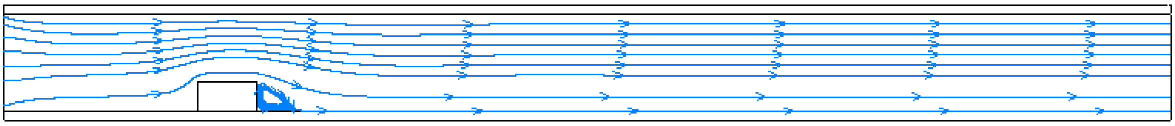
(b)

Fig.3.1.Temperature contours for Re=500, $Gr=8.65\times10^5$, $q_v=1\times10^5 w/m^3$, $W=15\times10^{-3}m$ $H=7.5\times10^{-3}m$ (a) without radiation (b) with radiation

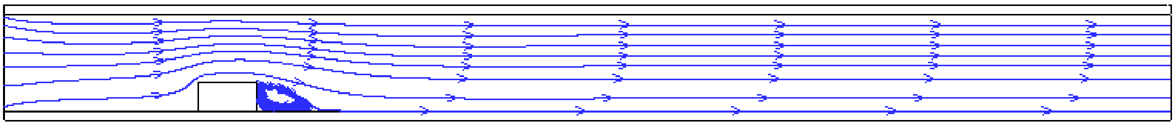
3.3 Effect of the Reynolds number

The effect of the Reynolds number on the heat transfer characteristics has also been investigated. Fig.3.2 shows the stream line pattern for Re=250,500,and 1000. Comparison of the stream lines show that, as reynolds number increases , the length and the relative strength of the downstream recirculation zone increases. The increased axial momentum of the fluid, caused by the constriction of the bypass region, inhibits its expansion into the full channel downstream of the heat source. The weak recirculation zone ahead of the heat source also increases in size and strength with increasing Reynolds number. The velocity magnitudes within this recirculation remain two to three orders of magnitude less than that within the core flow.

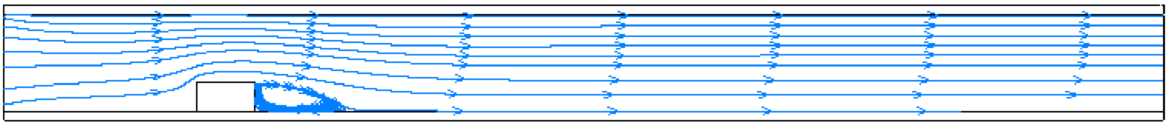
International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 01, January-2017



(a)



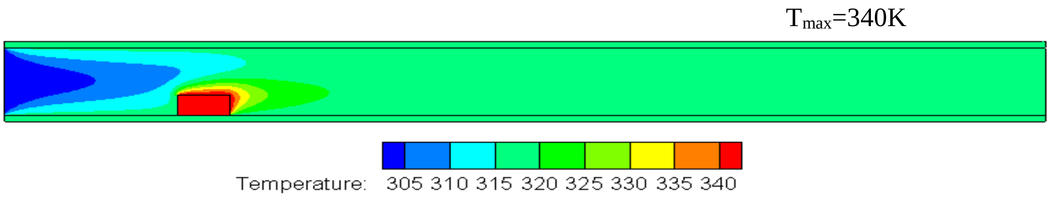
(b)



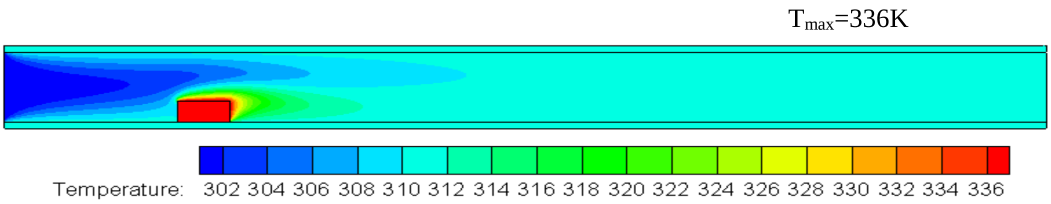
(c)

Fig.3.2. Streamline plots for various Reynolds number (a)Re=250, (b)Re=500, (c)Re=1000.

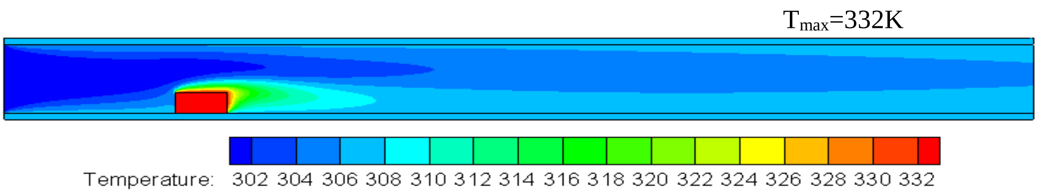
The temperature distribution is shown in fig.3.3 for Re=250,500,and 1000 while all the other parameters are at the baseline values. Convective heat transfer takes place between fresh air and heated obstacle when the air comes in contact with the obstacle. The temperature of the heat source decreases with increasing the reynolds number. Because the size and strength of the downstream recirculation zone increases with increasing reynolds number. The large recirculation carries away heat from the heat source to the core flow. Apart from convection, the heat source is exposed to the open atmosphere and the radiation heat transfer from the heat source is high. Because of radiation interaction, a radiation induced thermal boundary layer forms at the top channel wall. As the reynolds number increases, the effect of radiation decreases and the thermal boundary layer becomes less pronounced. As expected, the maximum temperature decreases as the reynolds number increases.



(a) Re=250



(a) Re=500



(b) Re=1000

Fig.3.3. Effect of Reynolds number on the temperature distribution at $Gr=8.65\times10^5$, $q_v=1\times10^5 w/m^3$, $W=15\times10^{-3}m$, $H=7.5\times10^{-3}m$

Fig.3.4 shows the variation of overall mean Nusselt number with the Reynolds number. This analysis also shows that the mean Nusselt number increases with increasing the Reynolds number. So heat transfer rate increases with increased Reynolds number.

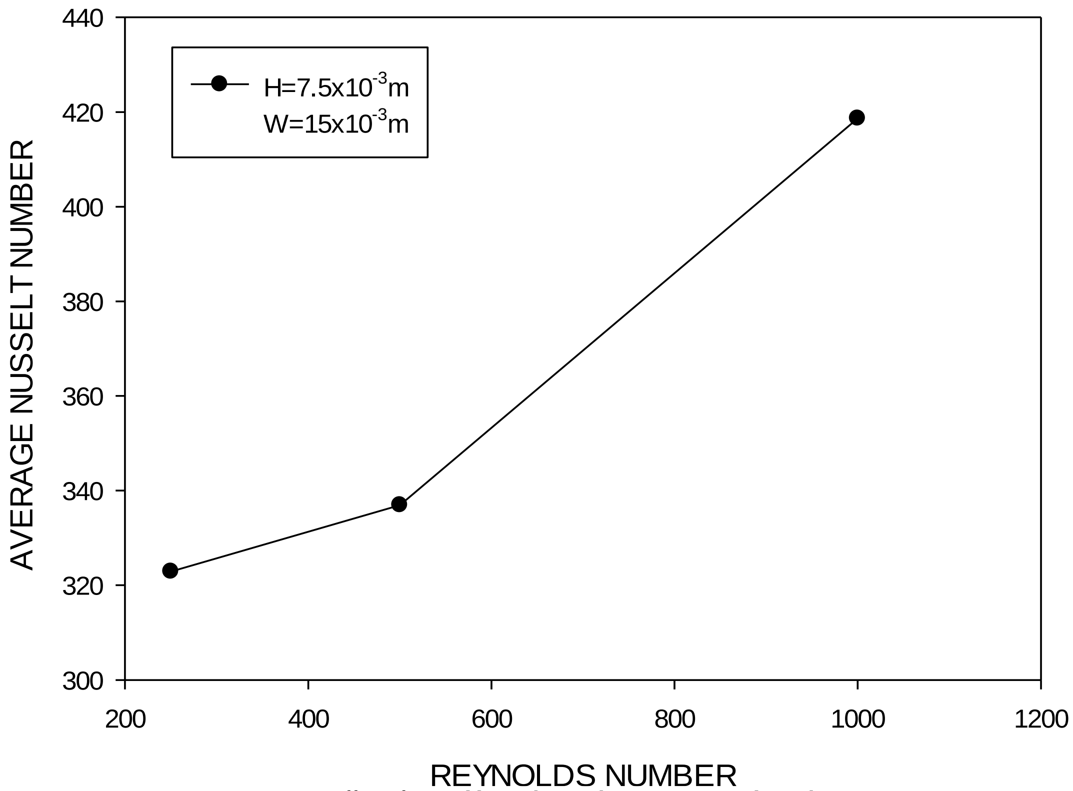


Fig. 3.4. Effect of Reynolds number on the Average Nusselt number

3.4 Effect of Heat source Height

The effect of varying the heat source height on heat transfer is demonstrated by comparing cases (case1 and case4) with a fixed width(w=0.015m). Fig.3.5 presents the changes in surface nusselt number along the peripheral distance of the heat source. From this analysis, it is clear that the convective heat transfer is greater in case1 where H=0.0075m than case4 where H=0.015m. Average Nusselt number at different height is shown in fig.3.6. The value of average Nusselt number is 322.97 in case1 and 126.27 in case 4. There is about 60.9% reduction in value is seen when height increases from 0.0075m to 0.015m. Thus reductions in the overall heat source mean Nusselt number are found as the height is increased. If the thermal energy flux within a heat source, such as the waste heat generated within an electronic component, can be held constant, division into smaller units will yield higher mean Nusselt number. This is analogous to transforming a large heat source into a set of fins composed of smaller heat sources.

Fig 3.7 shows the effect of the height of the heat source on the temperature distribution, when other parameters are fixed at the baseline value. As the height of the heat source increases from 0.0075m to 0.015m and the other values are at the baseline values, the maximum temperature increases from 336K to 340K, because of a decrease in radiation heat transfer. As the radiation heat transfer from heat source increases, the interaction between top wall and the heat source also increases.

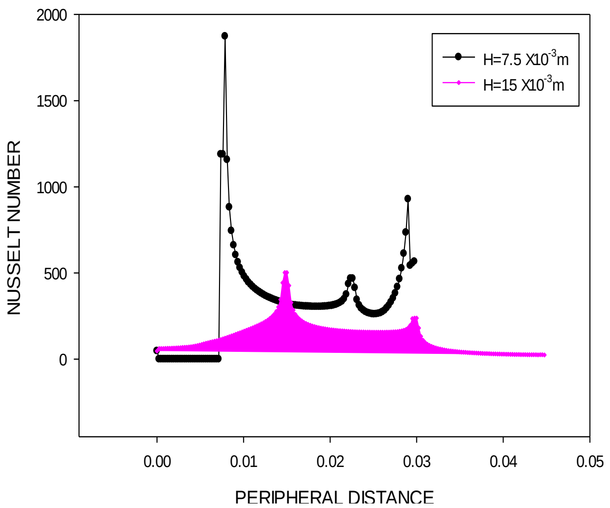


Fig. 3.5. Effect of Height variation on Nusselt number distribution around the heat source periphery for $Gr=8.65\times10^5$, $q_v=1\times10^5 w/m^3$, $W=15\times10^{-3}m$, Re=500

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 01, January-2017

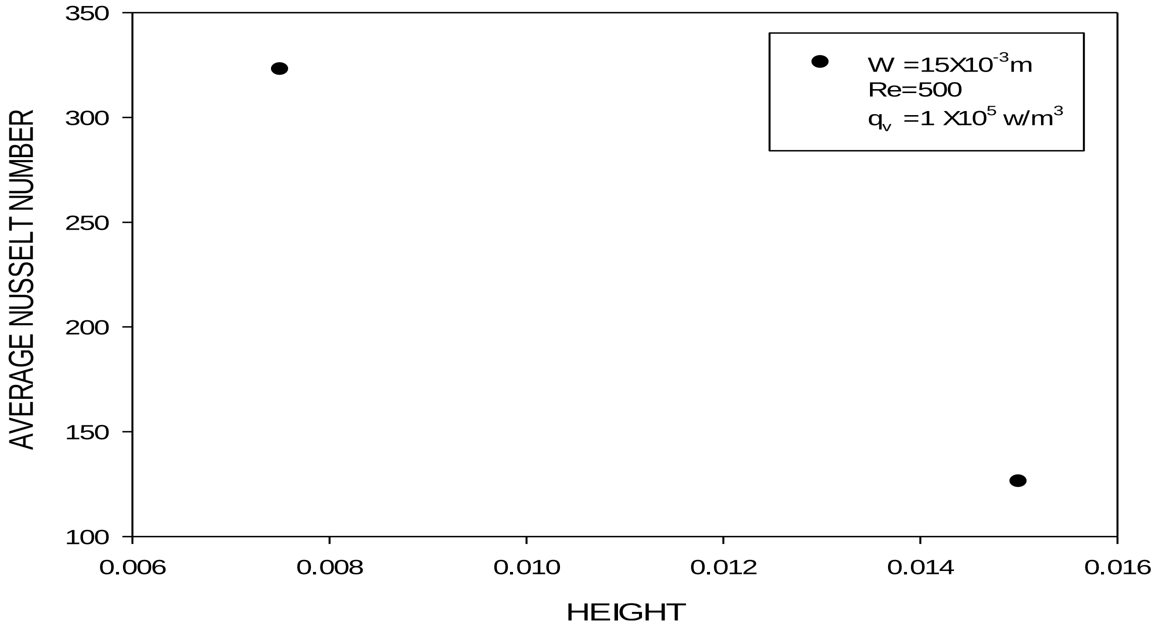


Fig.3.6. Average Nusselt number at Gr=8.65×105, qv=1×105w/m3, W=15×10-3m, Re=500 and for different height.

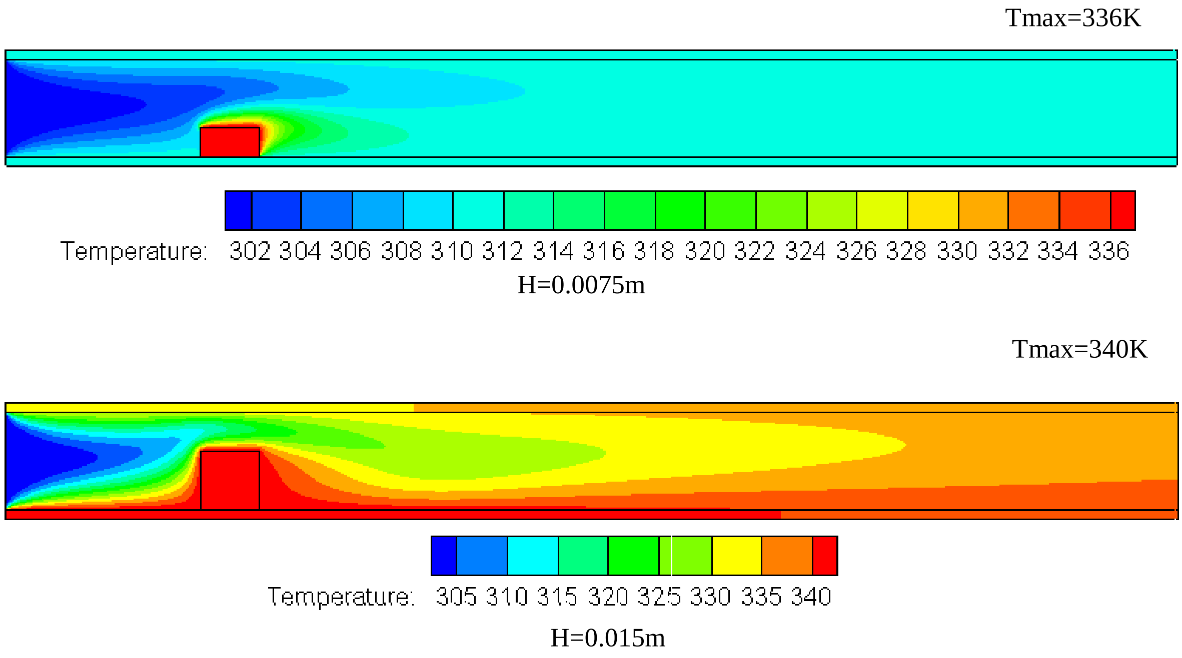


Fig.3.7. Effect of heat source height variations on the temperature distribution at Gr=8.65×105, qv=1×105w/m3, W=15×10-3m, Re=500

4. CONCLUSION

The results of a numerical investigation of mixed convection with surface radiation from a horizontal channel with discrete heat sources mounted on the bottom channel wall have been presented. The finite volume method has been used to solve the governing equations. The effect of the Reynolds number and height of the heat sources on the flow structure and heat transfer inside the channel have been examined.

Based on the parametric study, the following conclusions are arrived at:

Average Nusselt number increases with increasing the Reynolds number

Tmax decreases as the Reynolds number increases. The effect of radiation interaction also decreases as the Reynolds number increases.

Heat transfer rate increases with increasing reynolds number

Average Nusselt number decreases and Tmax increases as the height increases.

Overall heat transfer rate decreases with increasing the height of the heat source

REFERENCES

[1] T.J. Young, K.Vafai, Convective flow and heat transfer in a channel containing multiple heated obstacles, Int. J. Heat and Mass Transfer 41 (1998) 3279–3298.

[2] Timothy J. Young and Kambiz Vafai, Convective cooling of a heated obstacle in a channel, International Journal of Heat and Mass Transfer 41 (1998) 3131-3148

[3] O.Yemenici, Z.A. Firatoglu, H. Umur, experimental investigation of flow and heat transfer characteristics over blocked surfaces in laminar and turbulent flows.

[4] Mahmood Yaghoubi a, Ehsan Velayati b, Conjugate heat transfer for three-dimensional developing turbulent flows over an array of cubes in cross-stream direction.

[5] S.H.Kim, A.K. Anand, Laminar developing flow and heat transfer between a series of parallel plates with surface mounted discrete heat sources, Int. J. Heat and Mass Transfer 37 (1994) 2231–2244

[6] T. Furukawa, W. Yang, Thermal fluid-flow in parallel boards with heat generating blocks, Int. J. Heat and Mass Transfer 46 (2003) 5005–5015

[7] J. Davalath, Y. Bayazitoglu, Forced convection cooling across rectangular blocks, ASME J. Heat Transfer 109 (1987) 321–328.

[8] Y. Liu, N. Phan-Thien, A complete conjugate convection and radiation problem for a heated block in a vertically heated square enclosure, Comput. Mech. 24 (1999) 175–186.

[9] A.A.Dehghan, M. Behnia, Numerical investigation of natural convection in a vertical slot with two heat source elements, Int. J. Heat and Fluid Flow 17 (1996) 474–482

[10] Shiang-Wuu Perng a, Horng-Wen Wu, mixed convective heat transfer for unsteady turbulent flow over heated blocks in a horizontal channel.

[11] Adel Hamouche, Rachid Bessaïh, Mixed convection air cooling of protruding heat sources mounted in a horizontal channel, International Communications in Heat and Mass Transfer 36 (2009) 841–849

[12] B. Premachandran, C. Balaji, Conjugate mixed convection with surface radiation from a horizontal channel with protruding heat sources, International Journal of Heat and Mass Transfer 49 (2006) 3568–3582