International Journal of Advanced Trends in Computer Applications
www.ijatca.com

Effect of Length Ratio on Heat Transfer through Discrete Heaters in a Vertical Channel

[1]Karthik K Y, [2]Kishan Naik, [3]Banjara Kotresha, [4]N Gnanasekaran
[1,2]Department of Studies in Mechanical Engineering,
University BDT College of Engineering,
Davanagere -577004, Karnataka, India.
[3,4]Department of Mechanical Engineering,
National Institute of Technology,
Surathkal – 5750525, Karnataka, India.
[1]*karthikyogi01@gmail.com*, [2]*kishennaik@gmail.com* [3]*bkotresha@gmail.com*, [4]*gnanasekaran@nitk.edu.in*

Abstract: *This paper discusses about the effect of length ratio on mixed convection heat transfer through discrete heat sources placed inside a vertical channel. The distinct heat source assembly is the combination of alternative Bakelite and aluminum strips. The aim of the study is to investigate the effect of length ratio of the Bakelite strips on heat transfer and hot spot appearing on the heater surfaces. Four different length ratios of the Bakelite strips are considered in the present numerical exploration. The problem is solved as conjugate heat transfer as it involves aluminum and Bakelite solids along with fluid flowing in the vertical channel. The result shows that the hot spot reduces on the heater surfaces as Bakelite length ratio increases. It is also observed that excess temperature on all the heaters reduces as inlet fluid velocity increases. The results of excess temperature of all the heaters along with temperature contours are presented and discussed.*

Keywords: Vertical channel, Length ratio, Discrete Heat source, Conjugate heat transfer.

I. INTRODUCTION

In electronic devices, the recent trend is to reduce the size of the components which in turn reduces the space available for heat transfer. It is necessary to increase the heat dissipation rate by reducing the temperature of the component and provide efficient cooling to the electric devices like printed circuit board (PCB). The set of discrete heat sources having individual lengths mounted on the PCB in the vertical channel are investigated by many researchers for increasing the heat transfer rate. Natural convection is most useful for cooling the electronic device by considering air as the working fluid which is more economical due to less maintenance and easy cooling, but in some cases due to more heat flux, the natural convection is not useful to cool the electronic device. Hence for these cases forced convection is useful to increase the heat transfer but it requires additional pumping power. The mixed convection also increases the heat transfer rate in the electronic device which was observed by many authors.

There are many studies found in the literature for convection heat transfer in a horizontal, vertical, inclined channels and parallel plates. Desrayaud and Fichera (2003) studied natural convection through single heating component placed inside a vertical channel numerically. The effect of different protrusion of the heating component is considered for the analysis of hydrodynamic and thermal characteristics. Browmik and Tou (2005) performed experiments on four heater placed on the wall of the channel by natural convection. The four heaters resemble the in-line electronic chips placed in electronic component. The study reveals that the heat transfer rate is affected strongly by the number of chips used in the electronic boards. Gavara (2012) examined natural convection through discrete heat source mounted on opposite walls in a vertical channel numerically. They reported that the spacing between the heaters plays an important role in the heat transfer characteristics. The surface temperature of the heaters reduces by increasing the distance between the heaters.

Dogan et al. (2005) carried out experiments on flush mounted distinct heat sources in the bottom and top wall of the horizontal channel. All the heaters are assigned with a constant heat input. The research concludes that the highest heat dissipating components are to be placed at inlet and outlet of the channel. The heat transfer from the last row of the chips is due to secondary flow that is because of the buoyancy effect. Kumar and Balaji (2011) performed mixed convection numerical analysis on protruding heaters in a vertical channel. The artificial neural network is used for the inverse estimation of heat generated by the heat generating elements. Ahamad and Balaji (2015) studied laminar conjugate mixed convection through discrete heat sources by applying simple thermal model concept. They revealed that the geometric complexity can be reduced by applying simple thermal model concept instead of discrete heat sources which results in uniformly distributed heat in the domain. Sarper et al. (2018) analysed both natural and mixed convection through distinct heating elements in a vertical channel both numerically and experimentally. They studied the effect of Grashoff number, Reynolds number and length ratio on the heat transfer and fluid flow. It is given that the variation of heaters spacing affects the position of the hot spot in the channel. Kotresha and Gnanasekaran (2019) numerically predicted the isothermal condition on all the heaters of the discrete heat source system placed in the vertical channel with the use of metal foam porous medium. Premachandran and Balaji (2006) numerically investigated heat transfer through protruding heat sources in a channel by keeping the size of channel, thickness of heat sources/substrate and spacing between the heat sources constant. They found that maximum temperature and effect of radiation decreases with increase in Reynolds number.

In the aforementioned literature, the study on effect of length ratio of the spacing between the heaters by the researchers on distinct heating elements placed on the walls seems to be interesting in order to enhance heat transfer so as to maintain the isothermal condition. Hence, this study explores the variation of length ratio (L_r) of the substrate on fluid flow and heat transfer characteristics in the channel. The length of the Bakelite substrate is varied and the result is reported in terms of temperature difference. The discrete heat source considered in the present study resembles the printed circuit board used in any electronic equipment.

II. PROBLEM STATEMENT

Figure 1 shows a vertical channel with centrally placed distinct heaters which is considered for the present numerical investigation. The distinct heaters assembly is prepared by placing Bakelite and aluminium strips alternatively along the fluid flow and the assembly is placed at the centre of the channel. The size of the Bakelite strips and aluminium strips are 150 x 250 x 22.5 and 150 x 250 x 20 (all dimension in mm), respectively. The heater is placed in between two aluminium strips hence the total heat supplied is divided equally on both side of the aluminium plate. The physical geometry considered in the present study is similar to the experimental setup of Kamath et al. [10].

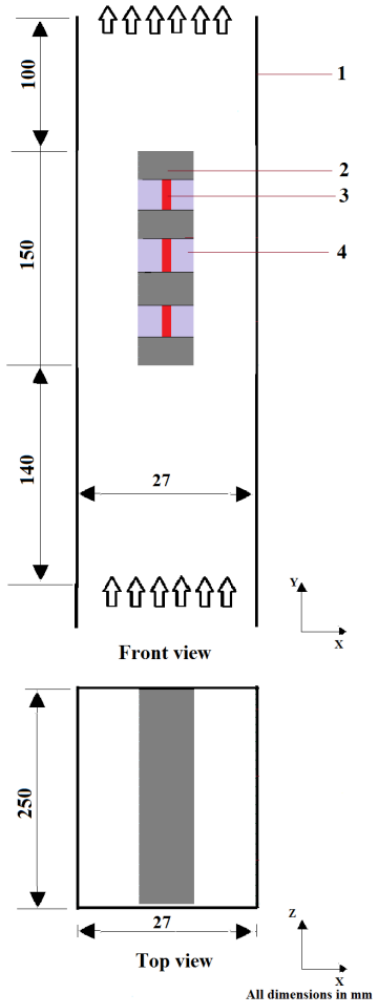

Figure 1: Vertical channel with distinct heat sources (1) side wall (2) Bakelite (3) Heater (4) Aluminium

III. BOUNDARY CONDITIONS ON COMPUTATIONAL DOMAIN

The numerical computation is performed for one half of the geometry due to its symmetry about vertical axis. Subsequently, one side of the channel with half of the aluminium discrete heater assembly is considered for the further analysis. In boundary conditions, a uniform velocity and zero pressure is assigned at inlet and outlet of the channel, respectively. A known heat flux is specified for the heater and the side wall is kept adiabatic. The computational domain along with boundary conditions is shown in Fig. 2. The numerical simulation is performed for four different length ratios (L_r) of the Bakelite substrate and is defined as (refer Eq. (1) – (4)).

$$L_{r_1} = \frac{L_1}{L_2} = \frac{L_2}{L_3} = \frac{L_3}{L_4} = 1.0 \quad (1)$$

$$L_{r_2} = \frac{L_1}{L_2} = \frac{L_2}{L_3} = \frac{L_3}{L_4} = 1.1 \quad (2)$$

$$L_{r_3} = \frac{L_1}{L_2} = \frac{L_2}{L_3} = \frac{L_3}{L_4} = 1.2 \quad (3)$$

$$L_{r_4} = \frac{L_1}{L_2} = \frac{L_2}{L_3} = \frac{L_3}{L_4} = 1.3 \quad (4)$$

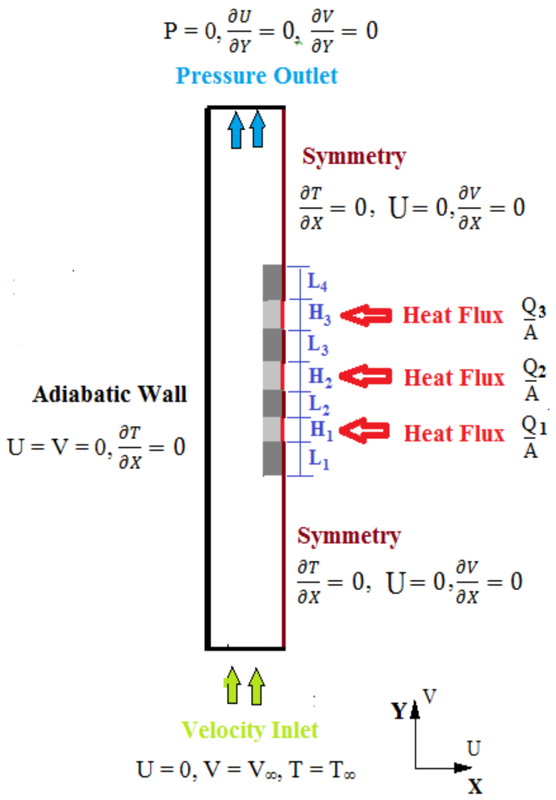

Figure 2: Boundary conditions used for simulation

IV. SIMULATION RESULTS

The numerical simulations are performed on the selected computational domain using commercially available ANSYS FLUENT. The governing equations used in the open region of the channel are similar to pipe flow. The air flows through the vertical channel with an inlet temperature of 30^0 C. The air velocity at the inlet varies from 0.42 to 3.5 m/s and the hydraulic diameter based Reynolds number varies from 2000 to 17000. The turbulent flow characteristic in the channel is captured using k-ω turbulence model. A conjugate heat transfer study is carried out for the computational domain since it involves both fluid as well as solid domains. The pressure and velocity are coupled using coupled scheme with pseudo transient in time. A second order upwind scheme is used for pressure, velocity, energy and for turbulence parameters. The convergence criteria for continuity are set below$1e^{-5}$, momentum is $1e^{-5}$, energy is $1e^{-10}$ and for turbulence parameters it is $1e^{-3}$.

V. RESULTS AND DISCUSSIONS

5.1 Grid independence study

Grid sensitivity analysis is carried out on the computational domain by selecting three different number of grid sizes. Table 1 shows the grid independency results carried out in the present study, based on the results the mesh size of 78,614 is selected as optimum for further numerical predictions because it gives less pressure deviation.

Table 1: Grid Independence Study

Grid Size	Maximum pressure (Pa)	Maximum temperature K)	% Deviation	
			Pressure	Temperature
41,506	0.129	370	11.0344	0
78,614	0.140	370	3.448	0
1,12,668	0.145	370	Baseline	

5.2 Validation of Numerical results

The numerical results of excess temperature obtained for heater 1 (bottom heater) is compared with experimental values of Kamath et al. (2014) and is shown in Fig. 3 for the purpose of validating the numerical methodology. The excess temperature results obtained numerically matches fairly well with the experimental results with a maximum deviation of 8%. This confirms that the methodology adopted for the solution of flow and heat transfer through the channel in the present study is correct.

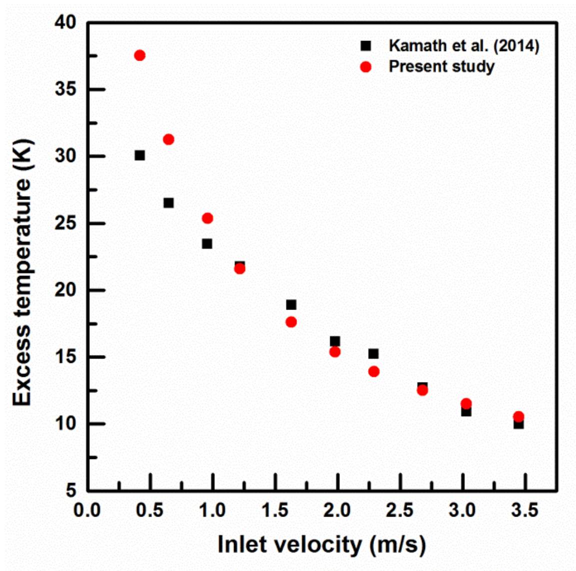

Figure 3: Variation of excess temperature with inlet velocity

5.3 Thermal results

The temperature distribution in the channel for different inlet velocities of the fluid for length ratio of 1.1 is shown in Fig. 4. All the heaters in the assembly are defined with equal input of 9.2W. The maximum temperature shows a general decrease with increase in fluid inlet velocity. The temperature of the substrate placed between middle heater and top heater shows almost same temperature as middle and top heaters. The distinct heating components can be seen in the channel at higher fluid inlet velocity. The top heater receives more heat from the secondary flow induced because of the buoyancy effect from the bottom and middle heaters.

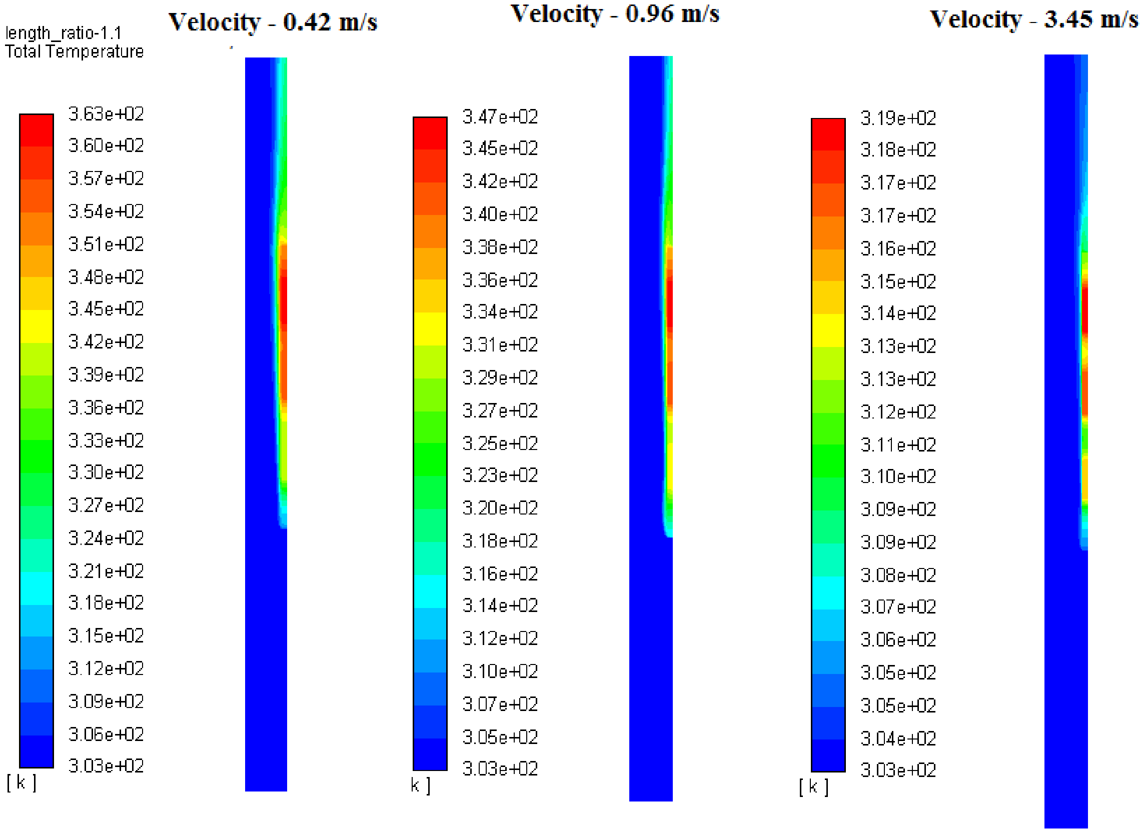

Figure 4: Variation of temperature in the channel for different velocities

Figure 5 shows the effect of length ratio on excess temperature for all the three heaters in the channel. It is observed that the temperature difference decreases as Reynolds number increases for all heaters. The top heater takes more heat from both middle and bottom heaters and hence top heater shows highest temperature in the channel. At particular Reynolds number the effect of length ratio on excess temperature is very less for all the heater surfaces. The slope of the curves changes at Reynolds number greater than 4000 for all the heating surfaces. At lower velocities, large difference in excess temperature exists between the middle and top heaters but this difference reduces as velocity increases.

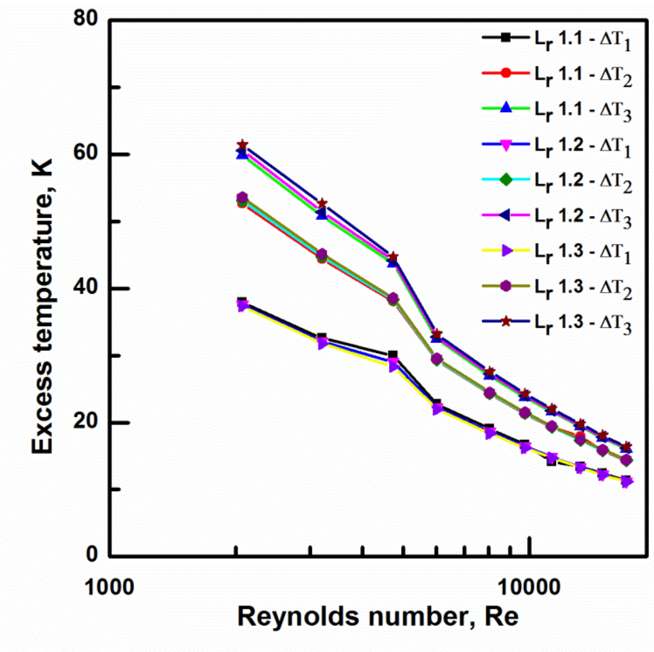

Figure 5: Effect of length ratio on excess temperature of heaters

VI. CONCLUSION

In this work the effect of substrate length ratio (L_r) on heat transfer through distinct heaters is carried out numerically. The numerical model consists of a vertical channel with distinct heaters assembly placed at the centre of the channel and is the combination of alternative Bakelite and aluminium strips. Four different substrate length ratios are considered in the study to investigate the thermal characteristics in the channel. The following are the major conclusions of the study.

- It is observed that the excess temperature obtained on all the heaters decreases with increase in the fluid inlet velocity.
- The top heater takes more heat because of the buoyancy effect caused by bottom and middle heaters.
- At lower velocities, the difference in temperature between middle and top heaters is higher but it reduces as velocity increases.
- It is noticed that the length ratio of substrate used in the study does not show significant improvement in heat transfer.

Conflict of Interest

The authors confirm that there is no conflict of interest to declare for this publication.

Acknowledgement

The authors would like to express their sincere thanks to the referee and for their valuable suggestions towards the improvement of the paper and also the CMFDP-2019 conference coordinators.

References

[1] Ahamad, S. I., & Balaji, C. (2015). A Simple Thermal Model for Mixed Convection From Protruding Heat Sources, *Heat Transfer Engineering*, *36:4*, 396-407.

[2] Browmik, H., & Tou, K. W. (2005). Experimental Study of Transient Natural Convection Heat Transfer From Simulated Electronic Chips, *Experimental Thermal Fluid Science*, *29(4)*, 485–492.

[3] Desrayaud, G., & Fichera, A. (2003). On Natural Convective Heat Transfer in Vertical Channels With a Single Surface Mounted Heat-Flux Module, *ASME Journal of Heat Transfer*, *125(4)*, 734–739.

[4] Dogan, A., Sivioglu, M., & Baskaya, S. (2005). Experimental Investigation of Mixed Convection Heat Transfer in a Rectangular Channel with Discrete Heat Sources at the Top and at the Bottom, *International Journal of Heat and Mass Transfer*, *32*, 1244–1252.

[5] Gavara, M. (2012). Natural Convection in a Vertical Channel With Arrays of Flush-Mounted Heaters on opposite Conductive Walls, *Numerical Heat Transfer Part A-Applications*, *62(11)*, 111–135.

[6] Kamath, P. M., Balaji, C., & Venkateshan, S. P. (2014). Heat transfer enhancement with discrete heat sources in a metal foam filled vertical channel, *International Communications in Heat and Mass Transfer*, *53*, 180–184.

[7] Kotresha, B., & Gnanasekaran, N. (2019). A Synergistic Combination of Thermal Models for Optimal Temperature Distribution of Discrete Sources Through Metal Foams in a Vertical Channel, *ASME Journal of Heat Transfer*, *141(2)*, 022004 (1-8).

[8] Kumar, A., & Balaji, C. (2011). ANN Based Estimation of Heat Generation from Multiple Protruding Heat Sources on a Vertical Plate under Conjugate Mixed Convection, *International Journal of Thermal sciences*, *50*, 532–543.

[9] Premachandran, B., & Balaji, C. (2006). Conjugate Mixed Convection with Surface Radiation from a Horizontal Channel with Protruding Heat Sources, *International Journal of Heat and Mass Transfer*, *49*, 3568-3582.

[10] Sarper, B., Saglam, M., & Aydin, O. (2018). Constructal Placement of Discrete Heat Sources with Different Lengths in Vertical Ducts Under Natural and Mixed Convection, *ASME Journal of Heat Transfer*, *140*, 121401 – 13.