Proceedings of the National Conference on Computational Modeling of Fluid Dynamics Problems (CMFDP-2019)
NIT Warangal, India – Jan 18-20, 2019
Paper No. XXX (The number assigned when the papaer is accepted)

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

Karthik K Y[a], Kishan Naik[a], Banjara Kotresha[b*], N Gnanasekaran[b*]
[a]*Department of Studies in Mechanical Engineering, University BDT College of Engineering, Davanagere -577004, Karnataka, India.*
[b]*Department of Mechanical Engineering, National Institute of Technology, Surathkal – 5750525, Karnataka, India.*

*Corresponding author Email: bkotresha@gmail.com, gnanasekaran@nitk.edu.in
Received: Date? Accepted: Date?

This paper discusses about the effect of length ratio on mixed convection through discrete heaters placed inside a vertical channel numerically. The discrete heat sources considered resembles the printed circuit board used in any electronic equipment. A vertical channel is considered in which distinct heaters is placed at the center. The distinct heaters are assembly of continuous Bakelite and aluminum strips. The aim of the study is to investigate the effect of length ratio of Bakelite on the heat transfer in a channel and also on the hot spot appearing on the heater surfaces. Four different length ratios of the Bakelite strips is considered in the present numerical exploration. Air is taken as a working fluid and flows in the channel with velocity varying between 0.4 to 3.5 m/s. The problem is solved as conjugate heat transfer as it involves aluminum and Bakelite solids along with fluid flowing inside the vertical channel. The result shows that the hot spot reduces on the heater surfaces as Bakelite length ratio increases. The results of temperature difference of each heaters and temperature contours are presented and discussed.

***Keywords*:** Vertical channel, Length ratio, Discrete Heat source.

1. Introduction

In electronic devices, the recent trend is to reduce the electronic component size and 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 varying lengths mounted on the PCB device 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 reduce the heat transfer but it requires additional pumping power. The study of mixed convection is used to increase the heat transfer rate in the electronic device so as to improve the life of the components by many authors.

There are number of studies are found for convection heat transfer in a horizontal, vertical and inclined channels and parallel plates in the literature. Desrayaud and Fichera [1] 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 [2] performed experiments on four heater placed on the wall of the channel by natural convection. The four heaters resembles 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 [3] 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. [4] carried out experiments on distinct heat sources flush mounted 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 [5] 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 [6] 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. [7] analysed both natural and mixed convection through distinct heating elements in a vertical channel both numerically and experimentally. The 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.

Aforementioned literature gives the number of studies on distinct heating elements placed on the walls of channels by varying shape of the heaters and length of the heaters. It is found that the variation of length ratio of the spacing between the heaters is considered very less by the researchers. Hence this study explores the effect of length ratio (L_r) of the fluid flow and heat transfer characteristics in the channel. The length of the Bakelite substrate used between the distinct heating elements is varied and presented the results of temperature difference and discussed.

2. Problem statement

Figure (1) shows a vertical channel with centrally placed distinct heaters 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 placed at the centre of the channel. The size of the Bakelite strips and aluminium strips is 150 x 250 x 22.5 and 150 x 250 x 20 (all dimension in mm). The heaters are placed in between two aluminium strips so that 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. [8].

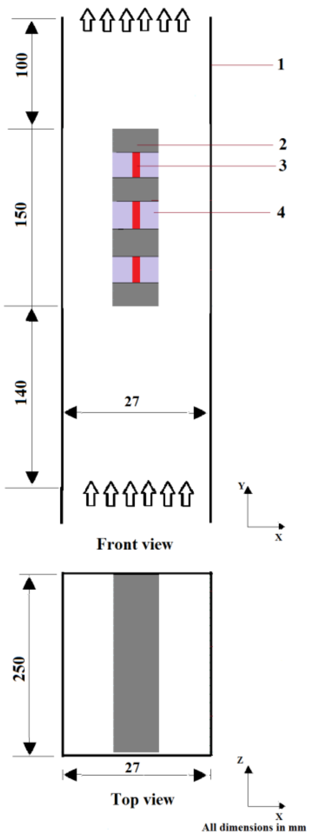

Fig. 1. Vertical channel with distinct heat sources (1) side wall (2) Bakelite (3) Heater (4) Aluminium

3. 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 plate are 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 in Eq. (1) – (4).

$$L_{r_1} = \frac{L_1}{L_2} = \frac{L_2}{L_3} = \frac{L_3}{L_4} = 1.0 \tag{1}$$

$$L_{r_2} = \frac{L_1}{L_2} = \frac{L_2}{L_3} = \frac{L_3}{L_4} = 1.1 \tag{2}$$

$$L_{r_3} = \frac{L_1}{L_2} = \frac{L_2}{L_3} = \frac{L_3}{L_4} = 1.2 \tag{3}$$

$$L_{r_4} = \frac{L_1}{L_2} = \frac{L_2}{L_3} = \frac{L_3}{L_4} = 1.3 \tag{4}$$

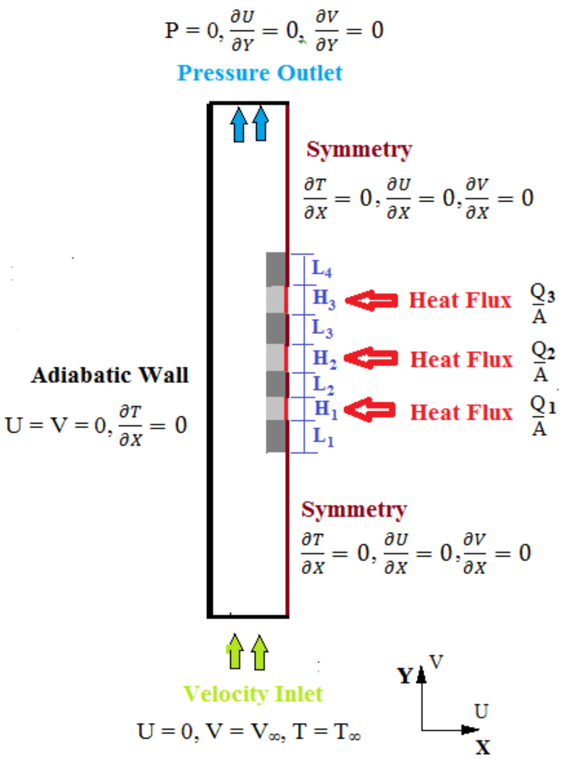

Fig. 2. Boundary conditions used for simulation

4. Simulation details

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 is 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 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 is 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 turbulence parameters it is $1e^{-3}$.

5. Results and discussions

5.1 Grid independence study

Grid sensitivity analysis is carried out on the computational domain by selecting three different number 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 present numerical result of excess temperature obtained for the heater 1 (bottom heater) is compared with excess temperature obtained experimentally by Kamath et al. [8] 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 experimentally obtained results. This confirms that the methodology adopted in the present numerical study is correct.

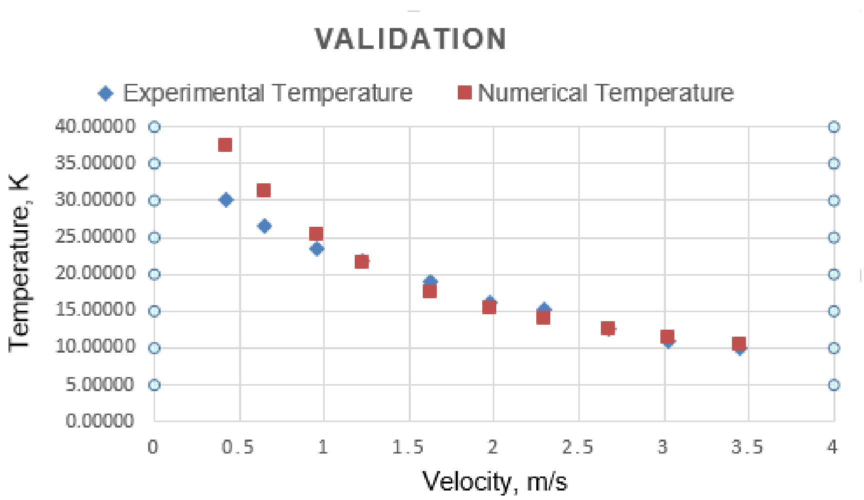

Fig. 3. Variation of excess temperature with inlet velocity

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 surface temperature of the substrate placed between middle heater and top heater shows almost same temperature as middle and top heater. The distinct heating components in the channel are seen at higher fluid inlet velocity. The top heater receives more heat from the secondary flow induced because of the buoyancy effect due to bottom and middle heaters.

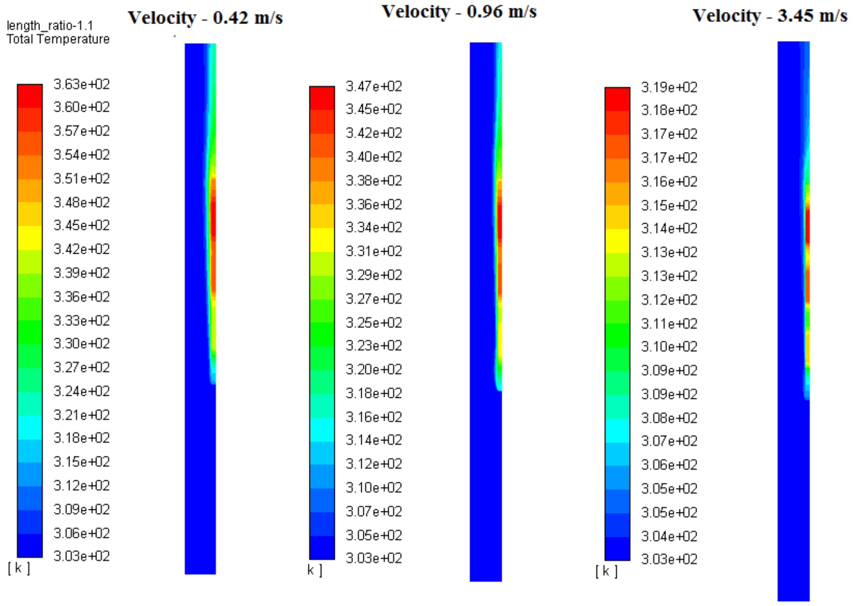

Fig. 4. Variation of temperature in the channel for different velocities

Figure 5 shows the effect of length ratio on excess temperature of 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 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 there large difference in excess temperature exists between the middle and top heaters but this difference reduces as velocity increases.

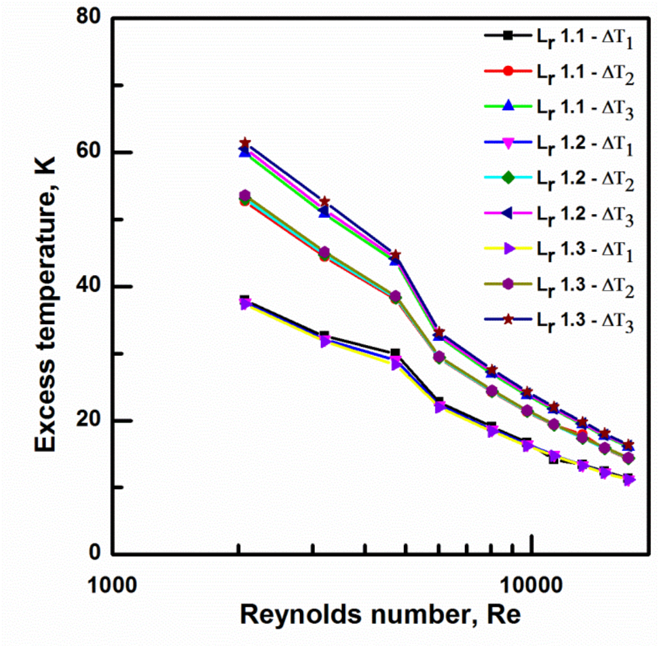

Fig. 4. Effect of length ratio on excess temperature of heaters

6. Conclusions

In this work the effect of substrate length ratio (L_r) on the heat transfer of 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. The assembly is made by placing the Bakelite and aluminium strips alternatively. Three different substrate length ratios are considered in the study to investigation thermal characteristics in the channel. It is observed that the excess temperature calculated for 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 for middle and top heaters shows higher deviation but it reduces as velocity increases. The length ratio of substrate used in the study does not significant improvement in the heat transfer is noticed.

REFERENCES

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

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

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

[4] A. Dogan, M. Sivioglu and S. Baskaya, 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, vol. 32, pp. 1244–1252 (2005)

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

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

[7] B. Sarper, M. Saglam and O.Aydin, Constructal Placement of Discrete Heat Sources with Different Lengths in Vertical Ducts Under Natural and Mixed Convection, vol. 140, pp. 121401 – 13 (2018).

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