

CFD Analysis of Heat Transfer Enhancement of an Electronic Chip in a Cavity

Harshit Verma¹, Vimal Kumar², Anuj kumar³

U.G. Student, Department of Mechanical Engineering, KIET Group of Institution Ghaziabad (U.P.), India¹

U.G. Student, Department of Mechanical Engineering, KIET Group of Institution Ghaziabad (U.P.), India²

U.G. Student, Department of Mechanical Engineering, KIET Group of Institution Ghaziabad (U.P.), India³

ABSTRACT: Due to the problem of heating of electronic chip in any electric instrument the possibility to reduce the working life of electric instrument are increase therefore it is required to reduce the heating of electronic chip and for this purpose we need to improve the heat transfer through the chip. For this purpose I have analysed the effect of chip position in an enclosure on heat transfer rate at a defined boundary condition. In this study I am considering the two chip in an enclosure and for different gap between the heaters i am analysing the effect on average Nusselt number. The heaters are placed at bottom wall and top wall are considered adiabatic.

KEYWORDS: Ansys 14.5(Fluent), Nusselt number, Temperature contour, stream function contour

I. INTRODUCTION

In this CFD analysis of electronics chip cooling, we are analysing the flow through a square obstruction with given boundary conditions. By the process we are solving the Continuity equation and Momentum equation and Energy equation by Marker and Cell algorithm. The method is finite difference method. Here our aim is to predict the temperature distribution, velocity distribution contours and Nusselt number variation at different orientation of heat source and different positions of it. We are trying to understand the heat flow for a natural convection heat transfer condition.

The objective of this project is to finding out the position of heat source for maximum heat transfer rate. To perform CFD I used Ansys 14.5 (Fluent). There are two electronic chip are present at different gap. I used Non-dimensional parameter to solve this problem. I consider the value of Rayleigh number is 10^4 . The gap between heaters are in range of 0.15, 0.2, 0.25, 0.3.

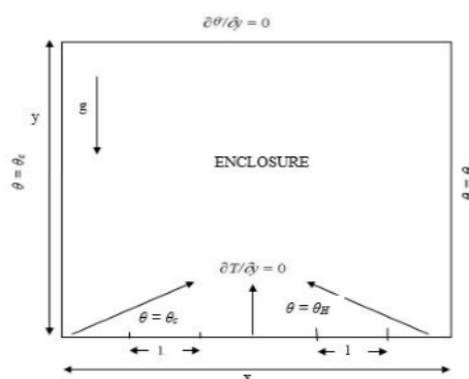


Fig 1 Project sketch

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 5, May 2017

As shown in figure 1, a simple sketch of our project is shown. There are two side wall are taken isothermally cooled and upper wall are assumed adiabatic and there are two heat source are present at different gap at bottom wall. The total length of heat wall is taken 0.2 times of total length of bottom wall.

II. RELATED WORK

All electronic components generate heat when it's used. Surrounding temperature of the electronic components have a direct impact on the performance of the device. In order to achieve the optimal performance of the device, the temperature should be within the prescribed limit. Electronic components mounted on the vertical boards, these form channels or cavities and the heat generated by the components is removed by a naturally induced flow of air. Thermally induced buoyancy forces for the fluid motion and transport processes generated in an enclosure is getting much importance because of its practical significance in science and technology. The topic of natural convection in enclosures is one of the most active areas in heat transfer research today. The current study is the prototype of many industrial and engineering applications such as cooling of electronic equipment, meteorology, geophysics, operations and safety of nuclear reactors, energy storage, fire control, studies of air movement in attics and greenhouses, solar distillers, growth of crystals in liquids etc. Buoyancy driven flows are complex because of essential coupling between the flow and thermal fields.

In the literature, investigation on natural convection heat transfer reported that the heat transfer occurs in an enclosure due to the temperature differences across the walls, a benchmark solution to natural convection of air flow in a square cavity with differentially heated walls was given by *Davis et al.* [1] where $10^3 \leq Ra \leq 10^6$ was considered. A comparison exercise was also performed by *Davis and Jones* [2] to confirm the accuracy of the benchmark solution. Natural convection of air in enclosures heating from below and symmetrically cooling from the sides has been numerically investigated using stream function-vorticity formulation by *Ganzarolli and Milanez* [3]. Natural convection of air in enclosures heating from one side and cooling from the ceiling has been numerically investigated using stream function -vorticity formulation by *Aydin et al.* [4]. Natural convection of rectangular enclosures with localized heating from bottom and cooling from the sides has been numerically investigated in which air is used as a working fluid has been studied by *Aydin and Yang* [5]. The heat transfer of a rectangular cavity with adiabatic side walls and aspect ratios of 1, 2, 4 and 8 is considered, along with the inclination angle of the enclosure was varied from 0° to 180° and the effect of inclination on flow field and heat transfer is studied in the paper by *Chang* [6]. Natural convection in enclosure with discrete isothermal heating from below is studied numerically using finite element method with aspect ratios of 0.5 to 1 and inclination angle of 0° to 30° was considered, where the pressure-velocity form of the Navier–Stokes equations and energy equation are used to represent the mass, momentum, and energy conservations of the fluid medium in the enclosure by *Saha et al.* [7]. *Radhwan et al.* [8] have studied the natural convection in a square enclosure with discrete heating of one vertical wall with isothermal heating strip, the remainder of the wall and the opposing one are isothermally maintained at a lower temperature where they have found that the variation of the local Nusselt number is influenced by this flow pattern and the average Nusselt number is higher than that of a discrete heating strip mounted on an adiabatic wall

III. STEPS TO PERFORM CFD ANALYSIS

We are using Ansys 14.5 to perform CFD analysis. These are the some steps are involve to perform the CFD analysis.

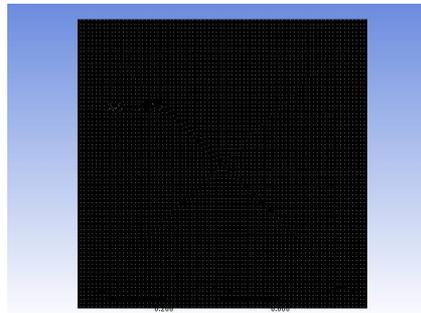
- **Sketching-** Firstly we open the FLUENT in Ansys 14.5. Then open the workbench to draw a 2D sketch. The dimension of sketch are according to the non-dimensional parameter.
- **Meshing-** In this step we generate a mesh with inflation at all the boundaries of enclosure. Generated number of node are 25,390 and no. of element is 25060

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 5, May 2017



No. of element -25060
No. of nodes-25390

Fig. 2 Mesh Generation

- **Apply Boundary Condition-** After the completion of mesh generation we have gone to put all boundary condition. In this we are using Pressure based solver, Power law method to solve the problem. We give all the boundary condition on setup. And then solve to find out the solution.
- **Solution-** After the solve the problem we got the result in form of temperature contour, stream function contour and avg, Nusselt number.

IV. EXPERIMENTAL RESULTS

In this analysis we are study about the heat transfer for different distance between heaters. The results are taken in the form of temperature and stream function contour and to compare the heat transfer we also find Avg. Nusselt number at the wall of heater.

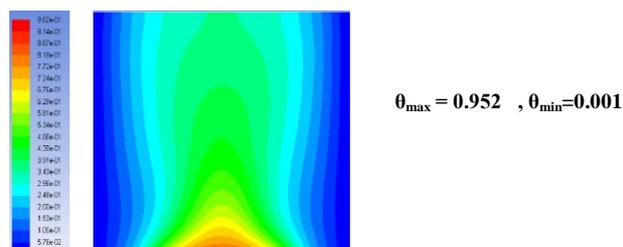


Fig. 2 temperature contour at Heater distance 0.15

As shown above fig.2 there are a temperature and stream function contour are shown in these contour we find that the temperature in case of heater distance 0.15 is maximum temperature are near heater and the temperature at near the both heater contain a common contour of isotherms.

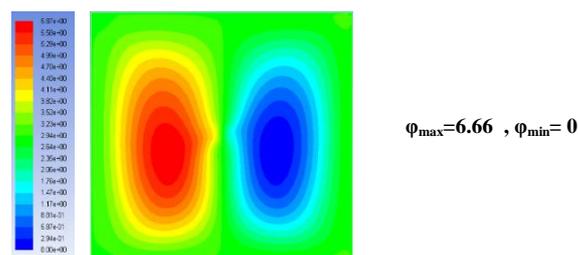


Fig. 3 Stream function contour at Heater distance 0.15

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 5, May 2017

In fig.3, the stream function contour are shown at heater distance of 0.15. we got that there are two different directional contour shown in figure. Every contour are occupy the half space of complete contour.

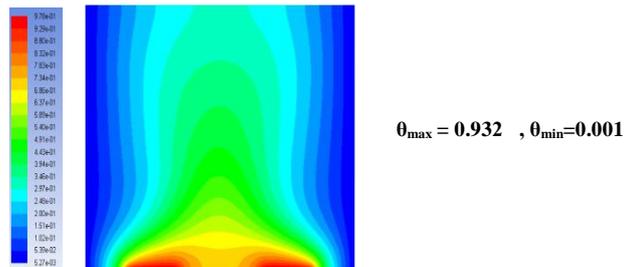


Fig. 4 temperature contour at Heater distance 0.2

As shown in fig.4 there are temperature contour at heater gap 0.2 are present. In this we got that the isotherm near the heater are goes to separate and the temperature distribution are present as similar to 0.15 heater gap. But the maximum temperature are reduce which state that the flow of heat are higher than above.

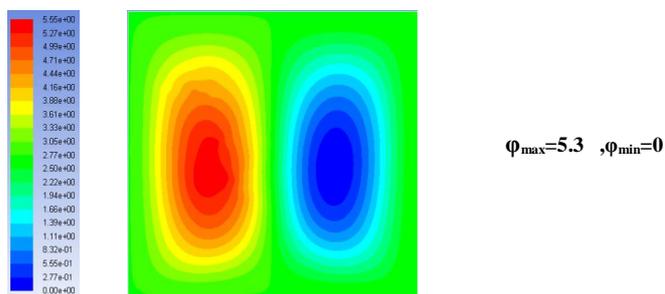


Fig.5 Stream function contour contour at Heater distance 0.20

As shown in fig.5 a stream function contour are present which state that the value of max. stream function are low than above its mean the flow of heat are at rotation is low. There are two different directional contour are present inside the enclosure which occupy the half space of complete enclosure.

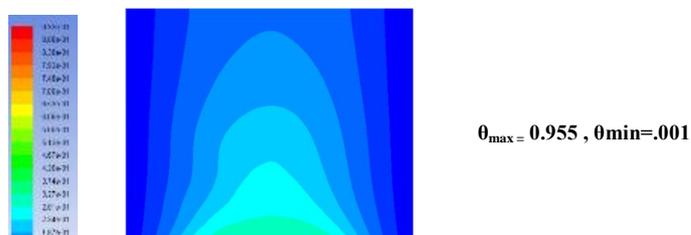


Fig.6 Temperature contour at Heater distance 0.25

In figure 6, a temperature contour are shown which show that the isotherms are created stepwise and reduce the temperature as well as increase the distance from heater. And there is no common isotherms are create near the both heater.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 5, May 2017

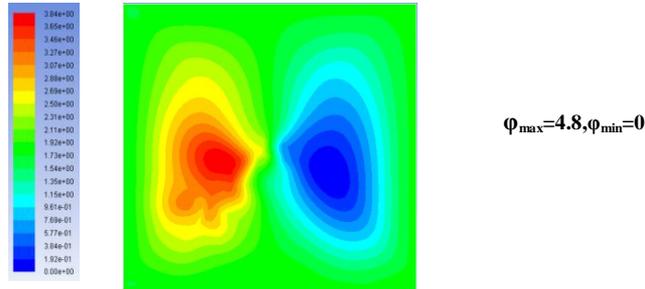


Fig.7 Stream function contour at Heater distance 0.25

In fig. 7 a stream function contour are shown in this we see that the max. stream function value is less than above. Which show that the heat flow are high compare to low heat gap.

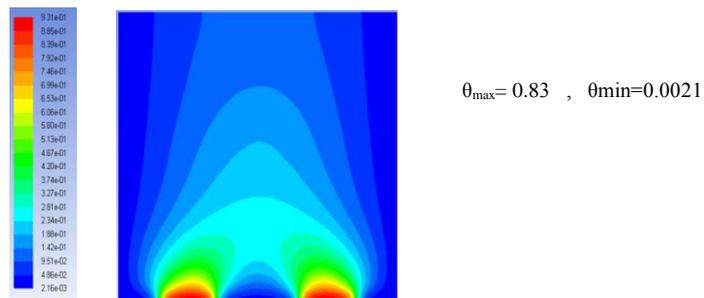


Fig.8 Temperature contour at Heater distance 0.30

In fig.8 we show the temperature contour which state that there are number of isotherms are created and that isotherms are define a definite temperature. The max. temperature are reduce in case of heater gap 0.3.

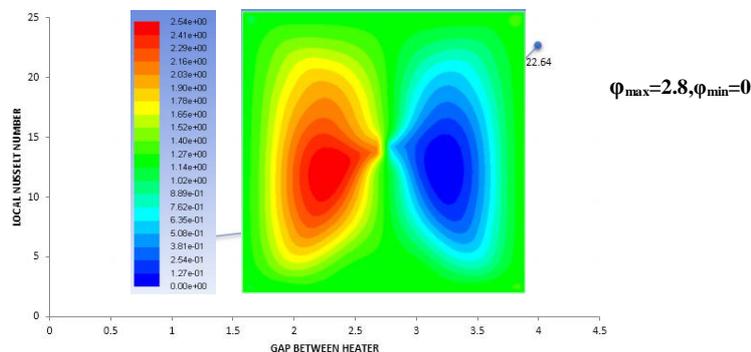


Fig.9 Stream function contour at Heater distance 0.30

In fig.9 we show stream function contour that show that there are max. stream function contour are less than above that also again define that the flow rate of heat are increase.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 5, May 2017

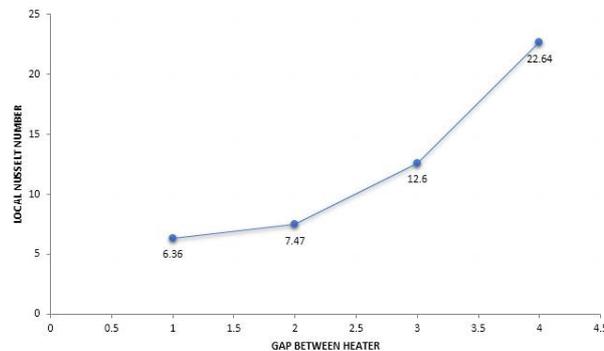


Fig.10 avg. Nusselt number

As shown above fig. 10 there are temperature, stream function contour and graph of comparison between Avg. Nusselt numbers are present. If we study the temperature contour at different heater distance, it show that the flow of heat is create a large curvature in case of 0.15 distance of heater and that curvature is going to be reduce as well as the heater distance increase. In the study of Stream function there are two contour are created between enclosures first are cooler and second is hotter. To study about the heat flow rate we are analyse the average Nusselt number at different heater distance and we got that to increase the distance between heater the value of Avg. Nusselt number are increase. For the distance of 0.15 it got minimum and for 0.3 it got maximum. The change in Avg. Nusselt number is not increase linearly, it is increasing rapidly according to increase the heater distance.

V. CONCLUSION

CFD on flow past a heated obstruction has been successfully carried out to analyse the complex heat transfer phenomena involved in the process. General conclusions that stemmed from this analysis are presented here, together with a brief recapitulation of some of the important remarks made earlier. In the present paper a problem on natural convection laminar flow from two discrete heating at the bottom wall has been investigated in Ansys14.5 (Fluent). The results have been presented for the chosen fluid of Prandtl number $Pr = 0.71$ and the Rayleigh number $Ra = 10^4$. We conclude that the Avg. Nusselt number are increase owing to increase in the distance between heaters.

REFERENCES

1. G. De Vahl Davis, 'Natural Convection of air in a square cavity: A bench mark numerical solution', International Journal for numerical methods in fluids, vol. 3, pp.249- 264, 1983.
2. Farah sadikinZaman,Tanmeysahaturja, 'Buoyancy Driven Natural convection Flow in an enclosure with two discrete Heating', International Journal for numerical methods in fluids, vol. 3,pp. 227- 248, 2013.
3. M M. Ganzarolli, L. F. Milanez, 'Natural convection in rectangular enclosures heated from below and symmetrically cooled from the sides', Int. J. Heat Mass Transfer, Vol. 38, No. 6, pp. 1063-1073, 1995.
4. O. Aydin, W-J Yang, 'Natural convection in enclosures with localized heating from below and symmetrically cooling from sides', International Journal of Numerical Methods for Heat and Fluid Flow, Vol. 10 No. 5, pp. 518-529,2000
5. GoutamSaha, SumonSaha, M. Quamrul Islam and M. A. RazzaqAkhanda, 'Natural convection in enclosure with discreet isothermal heating from the below', Journal of Naval Architecture and Marine engineering, Vol. 04 No.1, 2007.
6. K. Ben Nasr, R. Chouikh, C. Kerkeni, and A. Guizani, 'Numerical study of natural convection in cavity heated from the lower corner and cooled from the ceiling', Applied Thermal engineering, Vol. 26 No.7,pp. 772-775, 2006.
7. AmareshDalal and Manab Kumar Das, 'Laminar natural convection in an inclined complicated cavity with spatially variable wall temperature', International Journal of Heat and Mass Transfer, Vol. 48 .pp2986-3007,2005.