International Journal of Analytical, Experimental and Finite Element Analysis (IJAEFEA), Issue. 2, Vol. 1, April 2014

Numerical Analysis on Natural Convection from Dual Heating Element in an Enclosure

MuhammedShefeek K[1]
shefeekcea@gmail.com

Venkitraj K P[2]

College of Engineering
Adoor/Mechanical Department,
Kerala, India

***Abstract*—*This*paper reports the results of numerical investigation on natural convection in a three dimensional rectangular enclosure with heated parallel vertical plates located inside the enclosure. Simulation has been performed used commercial computational fluid dynamics package. The effect of spacing between the heated vertical plates and heat input on the flow and heat transfer characteristics inside the enclosure will be explored and discussed.**

***Index Terms*— Enclosure, Volumetric heat generation, Modified Rayleigh number**

I. INTRODUCTION

There are so many studies done in natural convection over isothermal flat surfaces. The flat surfaces are heated to a particular temperature by using a heater and after attaining a steady state it temperature readings are noted and calculations are done. Instead of this heater setup, we are conducting a study on ice block. Here ice block is assumed as isothermal flat surface. Studies are conducting for different orientation and shape of the ice block. The orientation and shape may strongly influence the flow and heat transfer rate. Therefore there is a need for additional research to predict the heat transfer rates from such narrow flat surfaces and short cylinder and to clarify the significance. In the recent years, to find the convective heat transfer coefficient an experimental setup consisted of 4 main parts, glass water reservoir, a vertical Aluminum flat plate, a heating system, a data acquisition system is used. It was verified a good agreement among the experimental, the numerical and the analytical results for both laminar and the beginning of transition regimes [1]. There isanother experiment to find the heat transfer rate over a horizontal isothermal circular disc and to develop a dimensionless correlation for natural convection for a wide range of Rayleigh number [2]. In 2013 an experimental study of steady state natural convection heat transfer from vertical, horizontal and inclined heated cylinder with different cross sections situated in a vented enclosure, opened from the top and the bottom are investigated [3]. The maximum value of the average heat transfer coefficient occurs in the horizontal position (θ=0º) and the minimum value in the vertical position (θ=90º), for the same heat input and aspect ratio. In the study by comuneia et al they conduct an experiment to determine the influence of neighborhood on the convective heat transfer. Setup consists of a vertical plate (copper), heater, and Heat fluxmeters. Experiment begins when steady state is attained [4]. They concluded that Heat transfer coefficient increases as the plate width decreases. With lowest plate width used, the heat transfer coefficient was about 150% above than for the widest plate. From the literature review, it was seen that most of the studies are conducted on heated plates and finding out the convective heat transfer coefficients. Our aim is to make a low cost natural convection setup using isothermal flat surfaces and short cylinders. An ice block of specified dimension is used as the isothermal flat surface and to determine the convective heat transfer coefficient (h), by natural convection of air over an ice block in different orientations and shape. The objective of the project is to find and compare the value of heat transfer coefficient experimentally. Experimental result for various orientations and shape is compared and analyzed.

II. METHODOLOGY

The purpose of this study is to determine the effectiveness of heat transfer in enclosures purely by natural convection only. For the analysis of the same a rectangular shaped enclosure similar to that dimensions having in real life applications has been selected. Inside the cabin two heat generating elements of thin rectangular plate shape are placed. The model of the above said setup is created and analysed in a commercially available computational fluid dynamic software namely ANSYS 14. For the purpose of analysis and simplicity the whole model is divided into solid zone and fluid zone. The heat generating elements and enclosure wall consists of the solid zone whereas the regions between them are selected as fluid zone. The objective of the analysis is to find out the optimum distance (S) between the two heat generating elements. Steady state heat transfer is assumed for the analysis. Also Boussinique approximation is taken for the study. The study is conducted for constant modified Rayleigh number but varying distance between heat generating elements for different aspect ratio.

III. METHODS AND MODEL

The setup is modelled as shown in the figure. The enclosure of rectangular shape is selected and adiabatic condition is assumed except in left and right walls. The left and right walls of the enclosure are taken as aluminium. The distance between heat generating elements , denoted by S and height of the heater , denoted as L. The ratio between these two is known as aspect ratio.

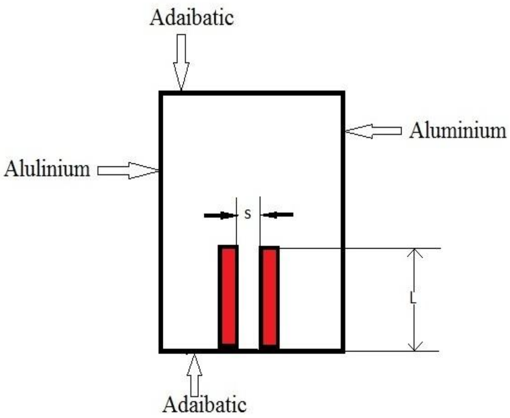

Figure1 Schematic problem representation

The aspect ratio of range 0.2-0.8 is taken for the analysis. Also the nature of heat transre occuring during the variation of the heat input of 5W and 10W are evaluated. The left and right walls of the enclosure are assumed of constant temperature (room temperature). All other boundaries of the enclosure are assumed to be adiabatic.

IV. ASSUMPTIONS FOR ANALYSIS

To simplify the problem, the following assumptions were made.

a) The solid and fluid zones are material with constant thermal properties.
b) Steady state heat transfer is only considered.
c) Variations in air velocities are neglected.
d) Bossinesq approximation is considered since the temperature difference is very small.
e) Properties of air inside enclosure are varied according to modified Rayleigh number.

V. SIMULATION PROCEDURE

The model is created in commercially available computational fluid dynamic software ANSYS 14 and its workbench. The rectangular enclosure is modelled as hollow rectangular cabin (80mmx360mmx140mm) and heat generating element (8mmx100mmx40mm). The enclosure walls, heat generating elements and the region between them are categorised as solid and fluid zones respectively. The solid zone and fluid zone are treated as two different domains. The two domains are then discretized using unstructured hexahedral mesh. Maximum attention is given to maintain the orthogonal mesh quality as nearer to 1.

Analysis is carried out with simple algorithm and Presto for pressure discretization, second order upwind scheme for momentum and energy. Relaxation factors are taken to be default values. Convergence criterion set for $10^{-4}$for continuity, x-momentum and y-momentum and 10^{-4} for energy. Constant properties of air are considered. Natural convection is gravity based. Then, the energy equation is selected and the flow condition is taken as laminar. Different material properties, cell zone conditions and boundary conditions are applied properly for analysis. The calculation procedure is based on simple gradient type. Green gauss cell based pressure is selected as PESTO (since natural convection is selected). Momentum and energy is selected as second order upwind. The solution is initialized as standard and computed from all zones. Results and reports are generated for different aspect ratio.

Modified Rayleigh number

$$Ra = \frac{g\beta q^{'''}\ L^{2}}{k\mathrm{U}} * pr$$

VI. RESULTS

A. *Temperature Distribution*

The study was carried out under specified conditions under varying aspect ratio and heat input. The temperature distribution was obtained for two different heat input, 5W and 10 W respectively. The aspect ratio in which the temperature distribution studied were 0.2, 0.4, 0.6 and 0.8.

Temperature distributions were studied along three lines parallel to the bottom of rectangular cabin. The three lines selected were one along the top of heat along the heat generating elements, second along the middle and third at the bottom.

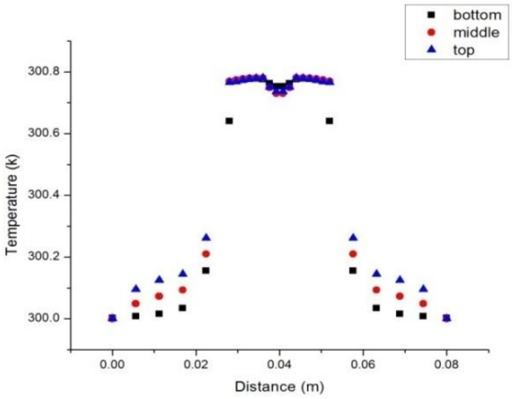

Figure2. Temperature distribution for 5W, Aspect ratio 0.2

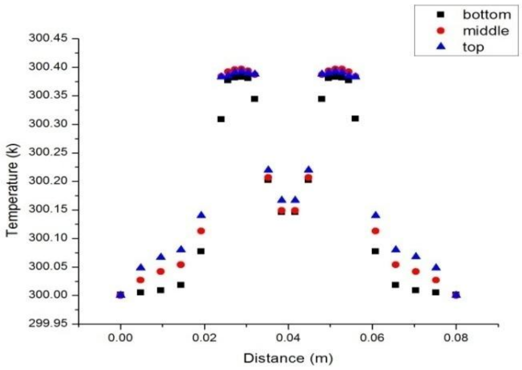

Figure3. Temperature distribution for 5W, Aspect ratio 0.4

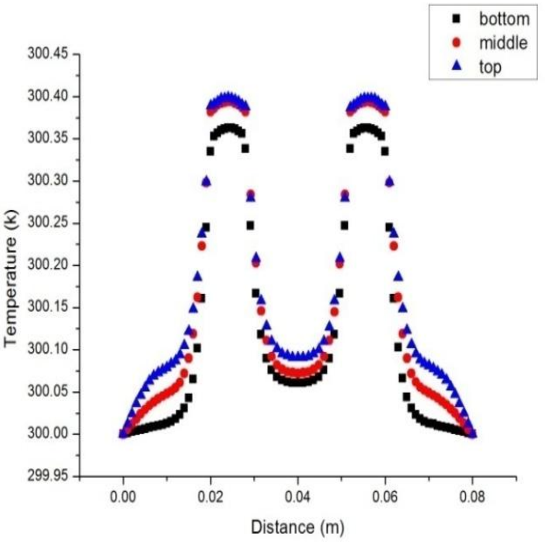

Figure 4. Temperature distribution for 5W, Aspect ratio 0.6

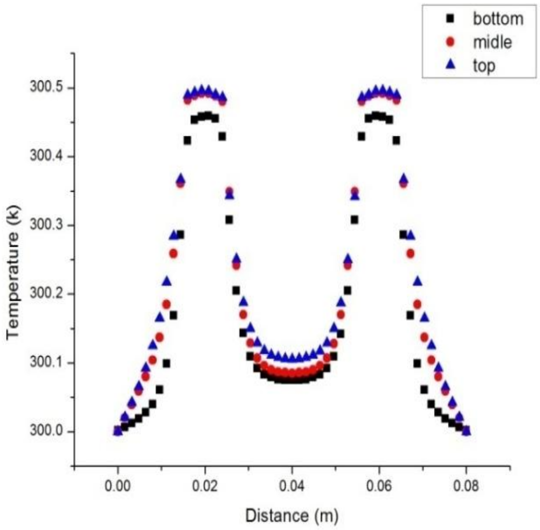

Figure 5. Temperature distribution for 5W, Aspect ratio 0.8

By observing temperature distribution for 0.2, 0.4, 0.6, 0.8 we can see that the maximum temperature attained for each aspect ratio decreases from aspect ratio 0.2 to 0.6 and it is again high for aspect ratio 0.8. The maximum temperature among the four aspect ratio is shown by that of 0.2, 300.5K. The lowest value of maximum temperature is obtained for aspect ratio 0.6.

The highest value of maximum temperature (T_{max}) obtained at aspect ratio 0.2 is due to the interaction of thermal boundary layer. The lowest value of T_{max} is obtained from aspect ratio 0.6 .

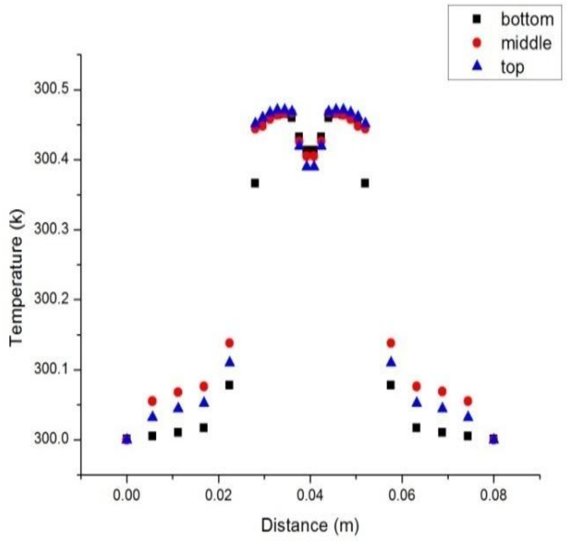

Figure 6. Temperature distribution for 10W, Aspect ratio 0.2

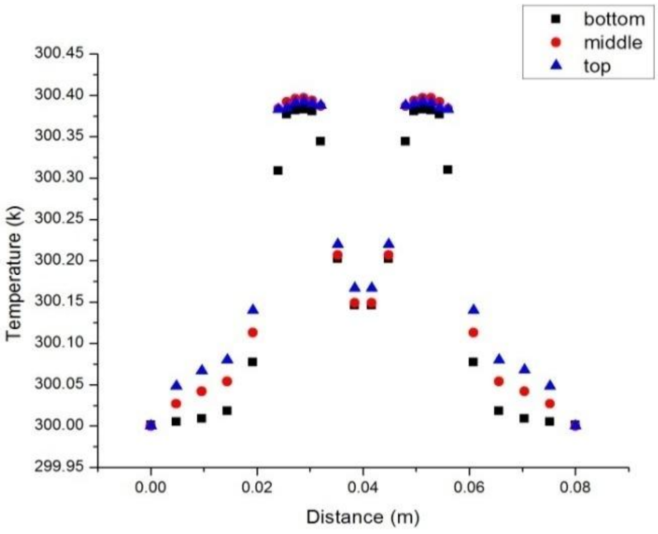

Figure 7. Temperature distribution for 10W, Aspect ratio 0.4

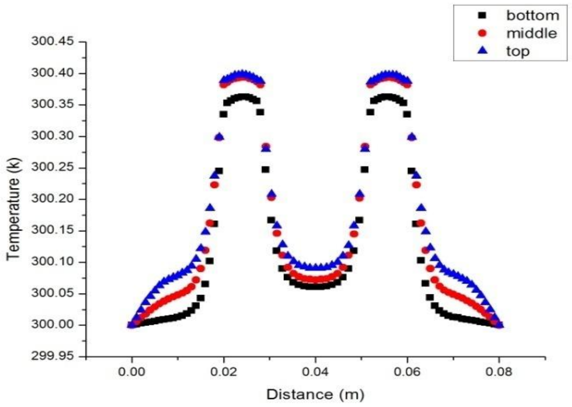

*Figure 8.*Temperature distribution for 10W, Aspect ratio 0.6

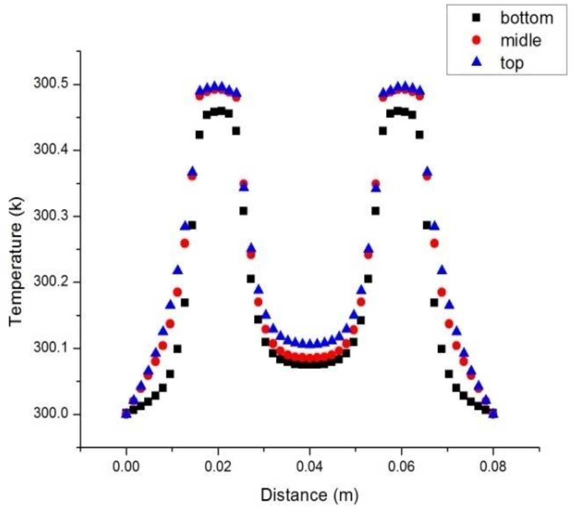

Figure9 .Temperature distribution for 10W, Aspect ratio 0.8

By observing heat input of 10w temperature distribution for 0.2, 0.4, 0.6, 0.8 we can see that the maximum temperature attained for each aspect ratio decreases from aspect ratio 0.2 to 0.6 and it is again high for aspect ratio 0.8 as same as the heat input of 5w . The maximum temperature among the four aspect ratio is shown by that of 0.2, 300.5K. The lowest value of maximum temperature is obtained for aspect ratio 0.6. The highest value of maximum temperature (T_{max}) obtained at aspect ratio 0.2 is due to the interaction of thermal boundary layer. The lowest value of T_{max} is obtained from aspect ratio 0.6 .

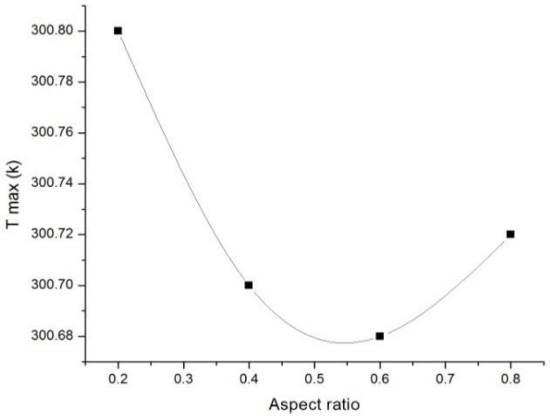

Figure 10. T max Vs aspect ratio for 5w

For more clarification of temperature distribution with aspect ratio two graphs are shown below, heat input of 5w and 10w respectively. From the graphs we can clearly seen that the maximum temperature is minimum in aspect ratio 0.6 in both 5w case and 10w case.

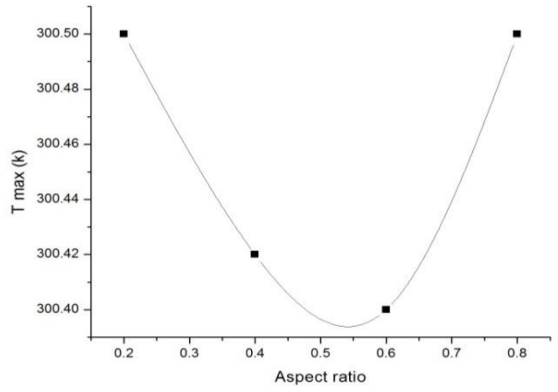

Figure 11. T max Vs aspect ratio for 10w

VII. CONCLUSION

CFD analysis of natural convection is done using varying aspect ratio. From the analysis following conclusions are made.

- The symmetric flow feature of the fluid in the cavity is retained even in presence of heating elements.
- Velocity vortex intensity is increased with increase in aspect ratio.
- Average temperature in spacing is decreasing with increasing the aspect ratio.
- Maximum temperature T_{max} is minimum in AR 0.6, because optimum spacing is 24mm.

REFERENCES

[1] Akihiko Horibe a, Rikio Shimoyama , Naoto Haruki a, Akira Sanada, Experimental study of flow and heat transfer characteristics of natural convection in an enclosure with horizontal parallel heated plates ,International Journal of Heat and Mass Transfer.(2012)

[2] T.N. Anderson ,M. Duke, J.K.Experimentaldittermination of natural convecton heat transfer coefficient s in attic shaped enclosure, International Communications in Heat and Mass Transfer 37 (2010) 360–363

[3] Kamil Kahveci,Natural convection in a partitioned vertical enclosure heating with a uniform heat flux,Heat transfer engg.ASME2007.

[4] Yasin Varol,Hakan F.Oztop,Tuncay Yilmaz, Natural convection in triangular enclosures with protruding isothermal heater,Heat transfer engg.50(2007)2451-2462.

[5] Pravez Alam , Ashok Kumar b, S. Kapoor b, S.R. Ansari a,Numerical investigation of natural convection in an enclosure, International Journal of Heat and Mass Transfer.17 (2012) 2403–2414

[6] Lino R. C., Ramos D. M., and da Fontoura Rodrigues J. L. A. (2003), Experimental dettermination of natural convection heat transfer coefficents in an attic shaped enclosures, International Journal of Heat and Mass Transfer.(2010)