International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-9 Issue-1, October, 2019

CFD Simulation of Natural Convection Flow and Heat Transfer Process in Rectangular Cavity

R. Harish

Abstract: ***In this paper we investigate the natural convective heat transfer process inside a ventilated rectangular cavity with a projected heat source. The heat source block is mounted on the bottom wall and a horizontal vent is provided on the top wall of the rectangular cavity. The flow is induced due to the density difference which arises due to the variations in temperature between the heat source block and the surrounding ambient fluid. A FORTRAN 90 CFD solver is developed to simulate the natural convection phenomena by solving the Navier-stokes equation, energy equation coupled with Realizable k-ε turbulence model. The transient flow behavior inside the cavity is simulated by varying the heat source aspect ratios, Grashof number and the heat source locations. It is found that the heat source aspect ratio and its locations significantly influences the flow and heat transfer characteristics inside the cavity. The bidirectional exchange rate across the horizontal opening increases linearly with Grashof number and heat source aspect ratio. A chaotic flow behavior pattern is observed across the opening and the strength of the instabilities increases linearly with heat source aspect ratio. It is identified that by varying the aspect ratio $0.1 \leq \beta \leq 3$, the average Nusselt number and mass flow rates are increased by 28% and 43% respectively.***

Keywords: ***Natural convection, Horizontal vent, Rectangular cavity, Nusselt number***

I. INTRODUCTION

Natural Convection is one of the most reliable options in thermal cooling due to its simplicity and efficiency in cooling the systems having complex geometries or packing. Moreover it requires less cost. For these reasons, natural convection finds broad range applications in the environmental and engineering problems such as thermal management of batteries, cooling of electronic devices, nuclear reactors. In thermal management of batteries, the strength of the heat source, the battery packing arrangement, shape and its size significantly affects the heat transfer rate. Most of the classical problems [1-3] on natural convection flows were based on differentially heated cavity. The left wall of the cavity is maintained at a constant wall temperature and the right walled is cooled. The upper and lower walls are considered as adiabatic and the natural convection flows were modeled using Boussinesq approximation. Ganzarolli and Milanez [4] investigated the natural convection flows inside a rectangular enclosure heated from the below and the side walls are cooled. The parametric study is performed by varying the Rayleigh number. They observed the formation of convective cells due to buoyancy driven mixing inside the enclosure. Tanny et al. [5] performed experiments in enclosure with vertical opening to measure the air flow patterns across the openings. They measured the air velocity and temperature distribution using three-dimensional sonic anemometer and found that the buoyancy force significantly affects the flow phenomena. Yang et al. [6] conducted experiments to investigate the bidirectional exchange across openings in ventilated cavity. They measured the temperature distribution of air across the openings and found that the buoyancy force increases the temperature and the shape of the opening has lower effect on the mass flow rate across the opening. Venkatasubbaiah and Jaluria [7] numerically investigated the buoyancy driven flows inside a square cavity with horizontal opening. The heat source is mounted on the bottom wall. They varied the size of the heat source and vents and found that the recirculating patterns and mass flow rates decreases with decrease in the vent width. They also observed that the buoyancy induced instabilities increases linearly with Grashof number. They identified the critical Grashof number above which the bidirectional flows becomes unidirectional flow. Harish and Venkatasubbaiah [8] performed numerical simulations in partial open enclosure under the turbulent flow regime. They considered a square enclosure with internal heater and horizontal vent. The flow characteristics inside the enclosure is studied by varying the strength of the heat source, vent size and changing the heat source and vent location. They observed that the mass flow rates across the vents drastically decreased when the heat source was positioned near the walls. Moreover the vent size affects the velocity and temperature distribution across the openings. They also investigated the natural convection flows [9] in a rectangular enclosure with dual heat source. They found thermal plume rises in the vertical direction due to buoyancy force and the spacing between the heat source significantly affects the thermal plume propagation rate. They found that when the heat sources are close to each other the unification of thermal plume appears at an early stage of the flow evolution. Biswas et al. [10] numerically and experimentally investigated natural convection inside a rectangular enclosure with protruding a heat source. They observed, heat transfer increases with the increase of aspect ratio of heater at higher Rayleigh number, whereas, the opposite trend is observed at lower Rayleigh number mainly due to the change in the thermal boundary layer thickness. Pretrel [11] investigated the buoyancy induced flows in a two compartmental enclosure connected by a horizontal opening which has applications in nuclear reactors.

Revised Manuscript Received on October 30, 2019.
* Correspondence Author
R.Harish*, Assistant Professor, Thermal and Automotive Research Group, School of Mechanical and Building Sciences, Vellore Institute of Technology Chennai, Tamilnadu-600127, India. Email: harish.r@vit.ac.in

The objective of the present study is to investigate the natural convective heat transfer rate in a rectangular cavity with heat source block mounted on the bottom wall. The free convection flow inside the cavity is modeled using the incompressible Navier stokes equation and energy equation. The turbulent flow is modeled using the realizable k-ε turbulence model. The parametric study is performed by varying the strength and the aspect ratios of the heat source. The aspect ratio is varied between the range $0.1 \le \beta \le 3.0$. The results are presented by plotting the stream function and temperature contours. The mass flow rates across the opening are analyzed by varying the Grashof number.

II. MATHEMATICAL MODELING AND NUMERICAL METHOD

The cubical enclosure considered in the present study is of 1.0 m long, 1.0 m wide and 1.0 m high. The heat source is of 0.2 m long and 0.2 m wide and is centrally located in the bottom wall of the enclosure. The enclosure top boundary consists of a horizontal opening which is naturally ventilated to the ambient atmosphere. The numerical simulations are performed by using a Finite difference Solver (FDM), where the problem is modeled as buoyancy induced turbulent flows. The turbulent flow problem is modeled by solving the Reynolds Averaged Navier-Stokes (RANS) equation for the velocity fields along with the time averaged energy equation for the temperature field. The turbulence is modeled by the realizable k- ε turbulence model for the kinetic energy and dissipation rate. The radiation effects are included by using the surface to surface radiation model. The time averaged governing equations are as follows:

$$\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial \overline{u_i}}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j}\frac{\partial \overline{u_i}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial \overline{x_i}} + \frac{\partial}{\partial x_j}\left[\upsilon \frac{\partial \overline{u_i}}{\partial x_j} - \overline{u_i' u_j'}\right] \quad (2)$$

$$\frac{\partial \overline{T}}{\partial t} + \overline{u_i}\frac{\partial \overline{T}}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\alpha \frac{\partial \overline{T}}{\partial x_i} - \overline{u_i' T'}\right] \quad (3)$$

$$\frac{\partial k}{\partial t} + \overline{u_i}\frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\frac{\upsilon_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right] + \upsilon_t\left[\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right]\frac{\partial \overline{u_i}}{\partial x_j} - \varepsilon \quad (4)$$

$$\frac{\partial \varepsilon}{\partial t} + \overline{u_i}\frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\frac{\upsilon_t}{\sigma_k}\frac{\partial \varepsilon}{\partial x_i}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}\upsilon_t\left[\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right]\frac{\partial \overline{u_i}}{\partial x_j} - C_{2\varepsilon}\frac{\varepsilon^2}{k} \quad (5)$$

'ρ' indicates the density of fluid, 'u'and 'T' represents the velocity and temperature fields, 'k' and 'ε' indicates the kinetic energy and dissipation fields. The strength of the heat source is denoted by a dimensionless number called Grashof number (Gr). The Grashof number is the ratio of the buoyancy force to viscous force. The Grashof number is mathematically represented as follows:

$$Gr = \frac{g\beta \Delta T L^3}{\upsilon^2} \quad (6)$$

where 'g' indicates gravity, 'β' is co-efficient of thermal expansion of air, 'ΔT' is the temperature difference between the heat source and ambient air, 'L' is the length of the enclosure and 'ν' indicates the kinematic viscosity of air. The investigations are performed for two different Grashof numbers Gr = 107 and 1010 with corresponding temperature differences of ΔT= 700 K and 1000 K respectively. The inertia and buoyancy force are related by a dimensionless number called Froude number (Fr). It is defined as the ratio of inertia to buoyancy force. The Froude number is mathematically represented as follows:

$$Fr = \frac{u_c}{\left(\frac{T_s - T_\infty}{T_\infty}\right) gL} \quad (7)$$

rtrtrrtr

'uc' indicates the forced ventilation velocity, 'Ts' and 'Tꝏ' represents the heat source and ambient temperature, 'g' and 'L' indicates the gravity and length of the enclosure.

III. RESULTS AND DISCUSSION

Figure 1 represents the rectangular enclosure with an internal heat source block mounted on the bottom wall. . The aspect ratio of the rectangular enclosure is (i.e. AR = H/W) is 0.5. The horizontal vent is symmetrical and its width is given by D/W = 0.1. The heat source is placed symmetrically at the bottom. The size of the heat source varies with respect to its aspect ratio (i.e. AR = h/w) such that for Aspect Ratio 1 the width of the heat source is given by w/W = 0.1, for ar = 0.25 w/W = 0.2, for ar = 0.5 w/W = 0.2, for ar = 2 w/W = 0.05 and for ar = 4 w/W = 0.05. The horizontal vent is placed on the upper boundary and the aspect ratio of the heat source block is varied.

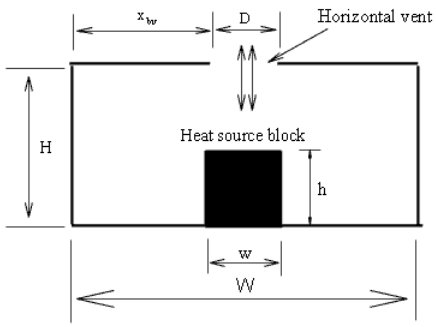

Fig. 1. Schematic diagram of rectangular enclosure with an internal heat source and horizontal vent

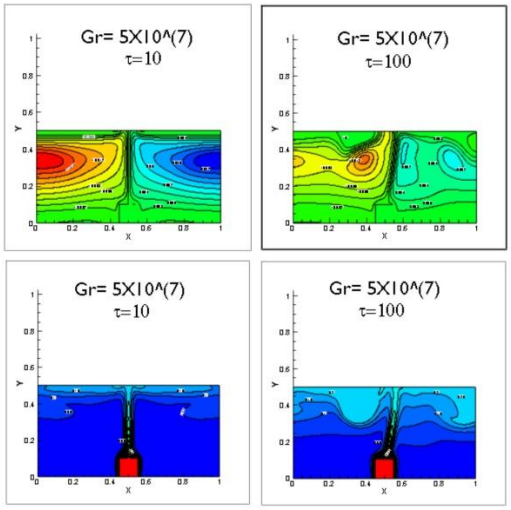

Fig 2. Evolution of stream function and temperature contours

Fig.2 depicts the time evolution study. As the time progresses, the mixing of hot and cold gases increases resulting in the formation of vortices and thus increasing the exchange rates and the rate of heat transfer. The stream function contours represents the formation of primary and secondary convective cells inside the enclosure. As time progresses, we observe formation of multi circulative patterns of cells which are unified into a single convective cell. Moreover, the thermal plume indicates the stratification of hot air and we observe layers of hot air propagating in the upward direction. The increase in buoyancy force strengthens the thermal buoyancy force and the propagation of thermal plume increases linearly with buoyancy force. Figure 3 represents the evolution patterns of stream function contours by varying the Grashof number. For Gr = 10^7, it is observed that the strength of the primary and secondary convective cells are uniform and as the Grashof number is increased, instability patterns are formed inside the rectangular cavity.

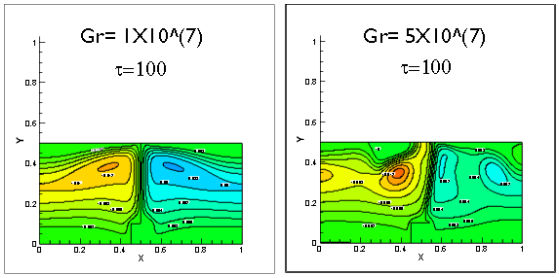

Fig.3 Effect of Grashof's Number on the Stream Function

In figure 3, as the Gr values are increased, the bidirectional exchange rate across the opening increases. The flow across the opening is bidirectional with hot air leaving through the left half of the enclosure and cold air entering through the lower half.

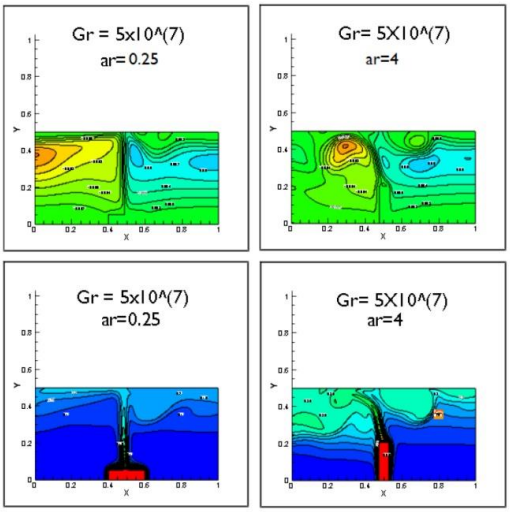

Fig 4. Effect of heat source aspect ratio on stream function and temperature contours

Figure 4 depicts the effect of aspect ratio variation of heat source on stream and temperature contours at Gr=5x10^7. As the aspect ratio is increased 0.25 to 4 the bi-directional flow is increased resulting formation of vortices which facilitates better mixing and thus increases the heat transfer rates.

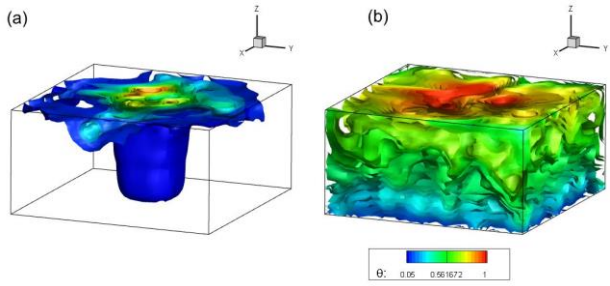

Fig.5 3-D numerical simulation results

Fig.5 indicates the iso-surfaces of temperature contours evolving from the heat source. It is observed that the thermal plume rises in the vertical direction due to the influence of buoyancy force and spreads in the lateral direction. The velocity of propagation of thermal plume increases with increase in buoyancy force and it fills the enclosure.

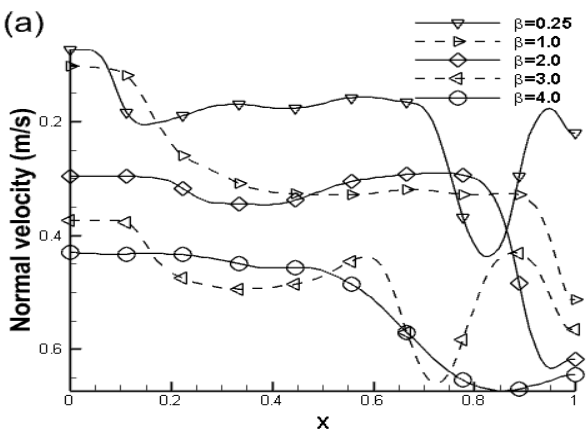

Figure 6 Normal velocity distributions

Figure 6 indicates the normal velocity distributions inside the rectangular enclosure by varying the heat source aspect ratio. It is evident that the flow velocity is higher for a lower value of heat source aspect ratio. The thermal buoyancy force increases the plume normal velocity and the velocity is stagnant in the central portion of the enclosure.

Table 1 Variation of average Nusselt number

Gr	β=0.5	β=1.0	β=2.0	β=3.0
10^3	5.79	6.22	7.66	8.25
10^4	8.32	8.46	9.63	10.42
10^5	12.14	12.92	13.42	14.15
10^6	16.85	16.99	17.36	18.54
10^7	20.31	20.75	21.65	22.91

Table 1 represents the variation of average Nusselt number by increasing the Grashof and heat source aspect ratio. It is evident that the Nusselt number is increased by 28% to 43% by increasing the Grashof number.

IV. CONCLUSION

The turbulent natural convection flow developed from an internal heat source inside a cubical enclosure is numerically investigated using computational fluid dynamics. The governing equations comprises of the continity equation, momentum equation and energy equation. The turbulent flow is modeled using realizable k-ε turbulence model. The stream line patterns and temperature contours are analyzed for a wide range of Grashof and heat source aspect ratios. A comparison is made for different values of heat source aspect ratio and it is found that the heat transfer rate increases linearly with heat source aspect ratio and Grashof number. The stream function contours indicated the formation of multi recirculating convective cell patterns by increasing the strength of the heat source. It was found that, the increase in Grashof number increases the buoyancy and thereby increases the rate of natural convection with increase in the fluctuations in the mass flow rate and increase in the exchange velocity through the vents. As the aspect ratio is increased 0.25 to 4 the bi-directional flow is increased resulting formation of vortices which facilitates better mixing and thus increases the heat transfer rates. It is evident that the Nusselt number is increased by 28% to 43% by increasing the Grashof number.

The average temperature inside the enclosure increased linearly with Grashof number. The average temperature at the vent also increases with the increase in aspect ratio and Grashof's number. The present results are useful in understanding the growth and spread of the thermal plume inside the vented enclosure with different aspect ratio of the heat source.

REFERENCES

1. G.D. Mallinson, G.D.V. Davis, Three-dimensional natural convection in a box-a numerical study, Journal of Fluid Mechanics, 1977, pp:1–31.
2. G.D.V. Davis, Natural convection of air in a square cavity: a bench mark numerical solution, International Journal of Numerical Methods in Fluids, 1983,pp:249–64.
3. J.L. Lage, A.Bejan, The Ra–Pr domain of laminar natural convection in an enclosure heated from the side, Numerical Heat Transfer Part A, 1991,pp:21–41.
4. M.M Ganzarolli, L.F. Milanez, Natural convection in rectangular enclosures heated from below and symmetrically cooled from the sides, lnternational Journal of Heat Mass Transfer, 1995, pp. 1063-1073.
5. J.Tanny, V.Haslavsky, M. Teitel, Airflow and heat flux through the vertical opening of buoyancy-induced naturally ventilated enclosures, Energy and Buildings, 2008,pp:637–46 .
6. X.Yang, K. Zhong, H. Zhu, Y. Kang, Experimental investigation on transient natural ventilation driven by thermal buoyancy, Building and Environment, 2014, pp:29–39 .
7. K.Venkatasubbaiah, Y. Jaluria, Numerical simulation of enclosure fires with horizontal vents, Numerical Heat Transfer part-A, 2012, pp: 179–196 .
8. R.Harish, K.Venkatasubbaiah, Numerical simulation of turbulent plume spread in ceiling vented enclosure, European Journal of Mechanics part-B Fluids, 2013, pp:142–58.
9. R.Harish, K.Venkatasubbaiah, Mathematical modeling and computation of fire induced turbulent flow in partial enclosures, Applied Mathematical Modelling, 2013, pp:9732–46.
10. N.Biswas, P.S.Mahapatra, N.K.Manna, P.C. Roy, Influence of Heater Aspect Ratio on Natural Convection in a Rectangular Enclosure, Heat Transfer Engineering, 2016, pp:125-139.
11. H. Pretrel, R. Sayada, K. Varrall, L. Audouin, O. Vauquelin, Experimental study based on large-scale smoke propagation fire tests through a horizontal opening connecting two mechanically ventilated compartments, Fire safety journal, 2017, pp:28–43.

AUTHORS PROFILE

Dr.R.Harish is working as an Assistant Professor in the school of Mechanical and Building Sciences at VIT Chennai campus. His research interests are in the field of computational fluid dynamics, Heat Transfer and Turbulent flows.