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

CFD Analysis of Turbulent Heat Transfer Characteristics in Cubical Enclosure

R. Harish

Abstract: ***In this paper we present the turbulent flow and convective heat transfer phenomena inside a cubical enclosure with an internal heat source. The enclosure is designed with an inlet and outlet vent and the heat source is mounted on the bottom wall. The turbulent flow is modeled by the computational fluid dynamics (CFD) approach using Lambremhorst k-ε turbulence model. A finite difference method is used to discretize the governing equations and an in-house CFD used is developed for simulating the turbulent characteristics. The parametric study is performed for the assisting and opposing flow characterstics inside the enclosure by varying the Grashof (Gr) and Reynolds (Re) number in the range of $105 \leq Gr \leq 1010$ and $102 \leq Re \leq 106$. The present study emphasises that the inertial force and buoyancy force has significant impact on the recirculation flow pattern inside the enclosure. The heat transfer rate is drastically influenced by the assisting and opppsing flow behavior developed inside the enclosure. It is observed that the mass flow rate across the outlet vent increases linearly with the Reynolds number. The flow behavior is highly chaotic with the development of instabilities inside the enclosure. The streamlines and temperature distribution patterns inside the enclosure indicated that the assisting flow enhanced the heat transfer rate by 48% while the opposing flow suppressed the heat transfer rate by 45% inside the enclosure. A multi recirculating convective cell pattern is formed at higher Grashof number and the size of the cell increases with increase in Grashof number. It is also found that the mass flow rate across the outlet vent increases linearly for assisting flow case while it decreases for the opposing flow case. It is evident from the present study that the assisting flow case is best suited for heat transfer enchancement in cubical enclosure.***

Keywords : Mixed convection, Assisting flow, Opposing flow, Convective cells, Grashof number

I. INTRODUCTION

Mixed convection is the combination of free and forced convection which has wide applications in building ventilation, cooling of electronic devices and nuclear reactors. The flow of air through openings such as doorways and windows affects the thermal comfort of occupants inside the building. In case of cooling of electronic devices the location of heat source, the direction of cold air flow and its velocity affects the heat transfer characteristics from the heat source. When the heat source is placed inside an enclosure the buoyancy force significantly affects the thermal plume behavior inside the enclosure. Moreover, for assisting flow case the inertial force of the forced air acts in the direction of gravity, while for opposing flow the inertial force acts opposite to gravity. Chan and Tien [1] performed experiments in a partial open cavity and found that the flow across the opening is bidirectional. They measured the temperature and velocity distribution across the openings and found that the flow across the openings is highly non-linear and is influenced by the conditions outside the cavity. Harish [2] numerically investigated the flow characteristics inside a partial open enclosure by varying the Grashof number and heat source locations. It was found that the heat source location had significant impact on the flow characteristics and heat transfer rate inside the enclosure. The thermal mixing characteristics inside the enclosure increased with increase in Grashof number. Mercier and Jaluria [3] performed experiments in a partially open enclosure and they observed formation of wall plume structure inside the enclosure. They also observed that the buoyancy force affects the thermal plume velocity and increase in buoyancy force increases the plume propagation velocity. Li et al. [4] performed experiments to investigate the oscillating behaviours of fire-induced exchange flow through a horizontal ceiling vent in a compartment. They investigated the various flow patterns inside the enclosure by using the principle of smoke particles of laser scattering. They obtained the ratio of cross section of outflow and ceiling vent area and the oscillation frequency of outflow by using image processing technology. Horikiri et.al [5] used a computational fluid dynamics approach for modeling a ventilated room with localized heat source with window glazing. Their results showed that the heat source and glazing sizes have significant impact on the temperature field, wall thickness and thermal conductivity Xaman et al. [6] performed numerical investigation on the transient heat and mass transfer by turbulent natural convection inside a ventilated cavity filled with air and carbon dioxide. They used volume method to solve the governing equations. Based on the results of the transient heat and mass transfer analysis, they concluded that from the flow patterns results, it can be observed that when natural convection prevails, the heat transfer phenomenon was more sensible than mass transfer process. They found that the Reynolds number significantly affect the flow characteristics. Arellano et al. [7] found that at large Reynolds numbers, the inertia forces, which are proportional to the density and the velocity of the fluid, are large relative to the viscous forces, and thus the viscous forces cannot prevent the random and rapid fluctuations of the fluid. At small Reynolds numbers, however, the viscous forces are large enough to overcome the inertia forces and to keep the fluid "in line". Thus the flow is turbulent in the first case and laminar in the second.

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 Reynolds number at which the flow becomes turbulent is called the critical Reynolds number. Similar investigations [8-10] were performed using different ventilation methods to remove pollutants from the ventilated chamber, and the numerical model was compared with the experimental results.

The objective of the present study is to investigate the turbulent mixed convection flow and heat transfer characteristics inside a cubical enclosure. The heat source block is placed at the bottom wall of the enclosure and the side walls of the enclosure are positioned with inlet and outlet openings. There is a horizontal vent which is provided on the upper boundary which acts as a bidirectional opening. The problem is modeled by solving the continuity equation for the mass conservation, momentum equations for the velocity field, energy equation for the temperature field, kinetic and dissipation equations for modeling the turbulent flow. The stream lines and temperature contours are analyzed for a wide range of Grashof and Reynolds number. The mass flow rate across the horizontal opening and the recirculating pattern inside the enclosure are compared between the assisting and opposing case. A Fortran 90 code based on finite difference method is developed to solve the governing equations.

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 Volume Solver (FVM), 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 = 7.88 x 1011 and 1.18 x 1012 with corresponding temperature differences of ΔT= 600 K and 900 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 'T∞' represents the heat source and ambient temperature, 'g' and 'L' indicates the gravity and length of the enclosure. The pressure velocity coupling is avoided by using SIMPLE algorithm and the convergence criterion is set to a small time step of 10-5.

III. RESULTS AND DISCUSSION

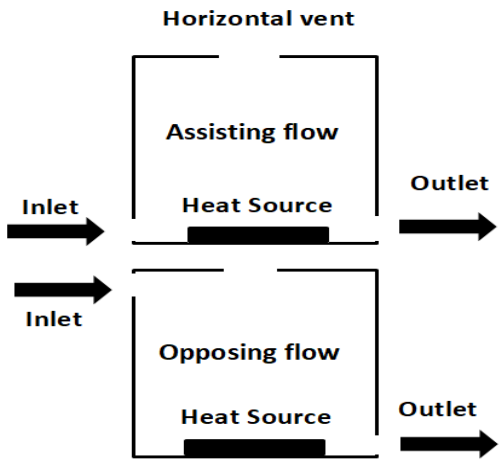

Figure 1. Schematic diagram of cubical enclosure

Figure 1 indicates the assisting and opposing flow cases considered in the present study. The forced air is supplied through the inlet port mounted on the left wall of the enclosure. For opposing flows the inlet port is shifted near the upper boundary of the enclosure. The horizontal vent is naturally ventilated and the flow is bidirectional across the vent. Figure 2 indicates the evolution of temperature contours inside the enclosure for Grashof number of 107. The cold air initially present near the heat source is heated because of the presence of heater. The heat source is considered as a constant wall temperature source. The density of hot air is lower than the density of cold air and hence the heated air rises in the vertical direction because of the upward buoyancy force. The thermal plume rises in the vertical direction and reaches the enclosure upper walls and the hot air leaves through the horizontal vent and the remaining plume fills the enclosure. The stratification of thermal plume is visualized from the temperature contours. The recirculation zone seen in figure 2 signifies that the cold air is trapped on the upper half of the enclosure and the lower half is filled with hot air. The inertial force from the forced air assists the thermal buoyancy force and significant amount of hot air leaves the enclosure. Moreover, the inertial force diffuses the heat generated from the heat source and tilts the thermal plume towards the enclosure right boundary.

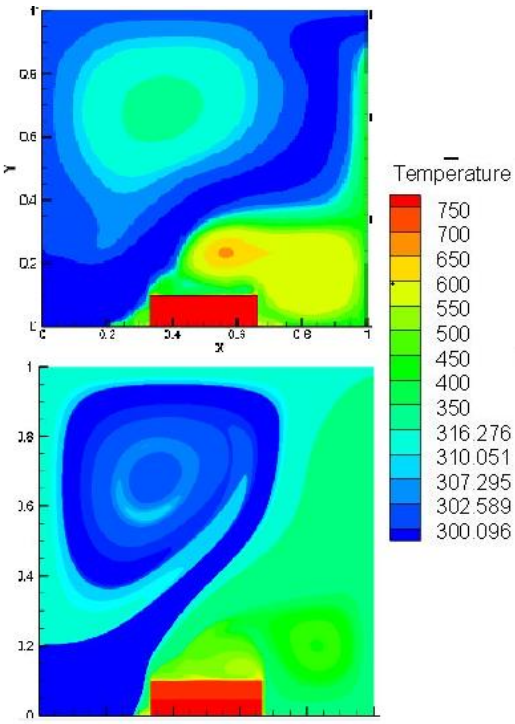

Figure 2. Temperature contours for Gr=107

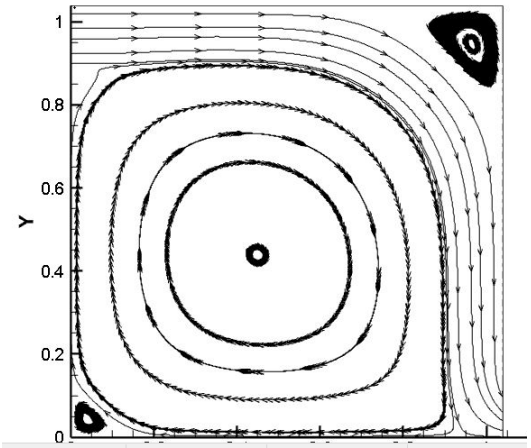

Figure 3 Stream line contours for opposing flow

Figure 3 represents the stream function contours inside the enclosure for Grashof number 107.for the opposing flow. The stream line pattern indicates the formation of multi recirculating convective cells inside the enclosure. It is visualized that the ambient air that enters through the inlet port suppresses the thermal buoyancy force and confines the hot air to the lower portion of the cavity. The primary central convective cell indicates the mixing of hot and cold air, while the secondary and tertiary cells formed near the corners indicates the trapped hot and air. Figure 4 indicates the stream function contours for the opposing flow and it is seen that the inlet air stream assists the thermal buoyancy force and significant amount of hot air escapes through the horizontal vent. It is also evident from the flow direction that most of the hot air is confined on the upper portion of the enclosure.

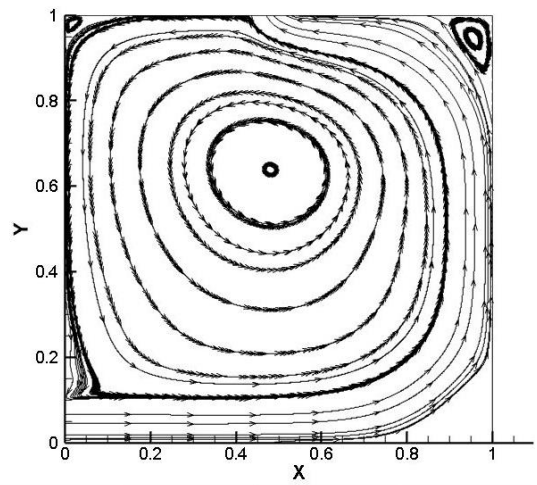

Figure 4 Stream line contours for assisting flow

Figure 5 indicates the average temperature distribution inside the cubical enclosure for a wide range of Grashof and Reynolds numbers. It is evident that the temperature distribution inside the enclosure increases linearly with Grashof number. The Grashof number is directly proportional to the temperature gradient and an increase in Gr value increases the strength of the heat source.

Hence it evident that an increase in Grashof number strengthens the thermal buoyancy force and increases the temperature distribution inside the enclosure.

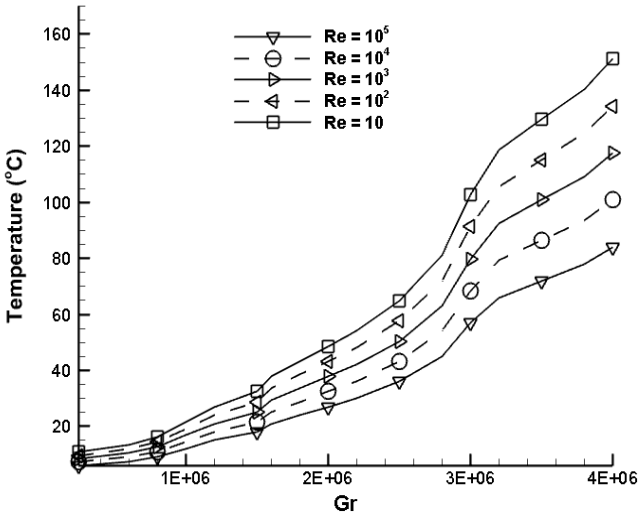

Figure 5 Average temperature distributions for assisting flow

However, an increase in Reynolds number strengthens the inertial force and decreases the thermal buoyancy force. This is visualized with the reduction in temperature distribution with increase in Reynolds number.

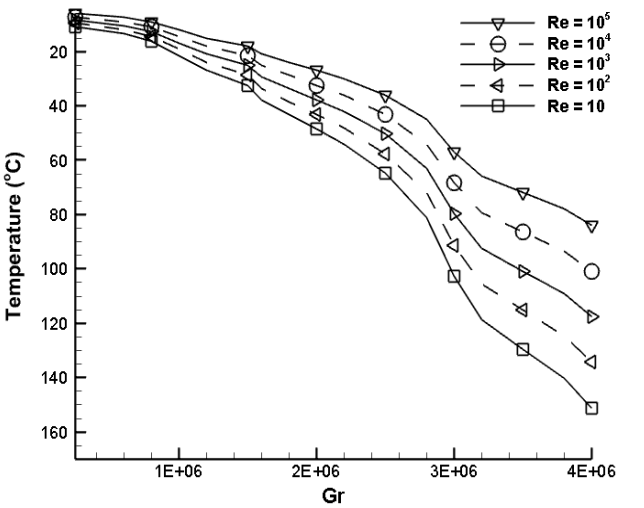

Figure 6 Average temperature distributions for opposing flow

Figure 6 indicates the temperature distribution inside the enclosure for opposing flow. It is observed that the inertial force suppresses the thermal buoyancy force and temperature inside the enclosure is decreased drastically. The temperature distribution patterns inside the enclosure indicated that the assisting flow enhanced the heat transfer rate by 48% while the opposing flow suppressed the heat transfer rate by 45% inside the enclosure.

Table 1 Variation of average Nusselt number

Gr	Re=102	Re=103	Re=104	Re=105	Re=106
105	11.45	13.56	17.53	19.56	22.67
106	14.56	16.66	18.54	20.11	23.63
107	16.88	18.53	20.90	21.43	24.81
108	18.63	20.03	22.59	24.25	26.93
109	20.68	22.91	24.38	25.87	28.71
1010	22.53	24.82	26.61	28.05	30.94

Table 1 indicates the variation of average Nusslet number by varying the Grashof and Reynolds number. It is evident that the Nusselt number increases linearly with Grashof number. The convective current rises with increase in Grashof number which strengthens the buoyancy force and enhances the heat transfer rate inside the enclosure. It is observed that as the Gr value is increased from 105 to 1010 the heat transfer rate is increased by 58.37%.

IV. CONCLUSION

The turbulent mixed convection flow characteristics inside a cubical enclosure with an internal heat source are numerically investigated by varying the flow direction. The turbulent flow is modeled by the computational fluid dynamics (CFD) approach using Lambremhorst k-ε turbulence model and the governing equations are discretized using finite difference method. The stream line patterns and temperature contours are analyzed for a wide range of Grashof and Rayleigh numbers. A comparison is made between the assisting and opposing flow case and it is found that the heat transfer rate increases linearly with increase in Grashof number. The stream function contours indicated the formation of multi recirculating convective cell patterns by varying the flow direction. It is identified that the assisting flow strengthens the thermal buoyancy force and enhances the heat transfer rate, while the opposing flow decreases the heat transfer rate. The average temperature inside the enclosure increased linearly with Grashof number. It is observed that as the Gr value is increased from 10^5 to 10^{10} the heat transfer rate is increased by 58.37%. The streamlines and temperature distribution patterns inside the enclosure indicated that the assisting flow enhanced the heat transfer rate by 48% while the opposing flow suppressed the heat transfer rate by 45%. The results from the present study will be suitable for designing effective thermal management systems for cooling of electronic devices.

REFERENCES

1. Y.L.Chan, C.L. Tien, Laminar natural convection in shallow open cavities. International journal of heat and mass transfer, 1986, pp.108:305-309.
2. R.Harish, Buoyancy driven turbulent plume induced by protruding heat source in vented enclosure, International Journal of Mechanical Sciences, 2018, pp.209-222.
3. G.P.Mercier, Y. Jaluria, Fire-induced flow of smoke and hot gases in open vertical enclosures, Experimental Thermal Fluid Sciences, 1999, pp.77–84 .
4. Q.Li, J.Li, J. Zhang, C. Liu, S.Ren, Oscillating behavior of fire-induced exchange flow through a horizontal ceiling vent,Procedia Eng, 2016, pp:299–308 .

5. K. Horikiri, Y. Yao, J.Yao, Modelling conjugate flow and heat transfer in a ventilated room for indoor thermal comfort assessment, Building and Environment, 2014,pp:135-147.
6. J. Xamán, J. Tunb, G. Álvareza, Y. Cháveza, F. Noha, Optimum ventilation based on the overall ventilation effectiveness for temperature distribution in ventilated cavities, International Journal of Thermal Sciences,2009,pp:1574–1585.
7. J.S.Arellano, M.G.Rivera J.M.R.Avila, J.Xaman, G.Alvarez, Numerical investigation of transient heat and mass transfer by natural convection in a ventilated cavity: Outlet air gap located close to heat source, International Journal of Heat and Mass Transfer, 2014, pp.268-278.
8. 8. J. Srebric, V. Vukovic, G. He, X. Yang, CFD boundary conditions for contaminant dispersion heat transfer and airflow simulations around human occupants in indoor environments, Building and environment, 2008, 294-303.
9. 9. Y. Xu, X. Yang, C. Yang, J. Srebric, Contaminant dispersion with personal displacement ventilation, Part I: base case study, Building and environment, 2009, 2121-2128.
10. X. Yang, J. Srebric, X. Li, G. He, Performance of three air distribution systems in VOC removal from an area source, Building and environment,2004,1289–1299

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.