A study of thermal distribution and fluid flow by natural convection inside a building enclosure with a centrally placed hot channel section using CFD

Jeseema Nisrin J¹, Velkennedy R², Kalidasan K³

¹Research Scholar, Department of Civil Engineering, Thiagarajar College of Engineering, Madurai - 625 015, Tamilnadu, India.
²Professor, Department of Civil Engineering, Thiagarajar College of Engineering, Madurai - 625 015, Tamilnadu, India.
³Lecturer, Department of Civil Engineering, Arulmigu Palaniandavar Polytechnic College, Palani-624 001, Tamilnadu

Abstract

The thermal distribution and fluid flow by natural convection inside a closed cool enclosure is studied. A channel section is placed at the center of the compartment and its temperature is hot. Bouyancy flow is induced inside the enclosure due to the thermal variation. Air is considered as the fluid inside the enclosure and its Prandtl number is 0.71. The fluid is expected to obey Boussinesq approximation. To evaluate the flow, the streamline, isotherms, horizontal and vertical centerline velocity and Nusselt numbers were calculated. The flow is modified by changing the Rayleigh number from 10¹ to 10⁶. The results show the influence of Rayleigh number on the flow pattern, velocity of fluid flow, and heat transfer rate. Computational fluid dynamics is used as the tool for this study.

Keywords: Natural convection; Channel section; Fluid flow; Thermal distribution; Rayleigh number.

Nomenclature

| Symbol | Description |
|--------|-------------|
| u      | Dimensionless horizontal velocity |
| v      | Dimensionless vertical velocity |
| T      | Dimensionless temperature |
| t      | Dimensionless time |
| w      | Dimensionless vorticity |
| Pr     | Prandtl number |
| Ra     | Rayleigh number |
| Nu     | Local Nusselt number |
| N̄Nu   | Average of Local Nusselt number |
| CFD    | Computation Fluid Dynamics |
| FDM    | Finite Difference Method |

1. Introduction

Due to urbanization and industrialization, the buildup area of the world is increased exponentially and the time spent by people in an indoor environment is also increased. According to International Energy Agency (IEA), the average time spent by the individuals in an indoor environment got increased in recent days and more than 53% of residential end energy is utilized for space heating and cooling[1]. Normally to increase energy efficiency, these buildings are enveloped. The heat transfer and fluid flow inside these enclosures have to be analyzed in detail. This is due to the fact that these pieces of knowledge will help in an efficient design of HVAC systems, the heat dissipating units in nuclear power plants, glazing units, fire-safe structures, and so on. Natural convection inside the closed cavity was studied by many researchers in the last three decades due to its vast engineering applications [2][3]. Enclosure with conductive and adiabatic blocks has been studied by the researchers like Bhave et al.[4], Mousa [5]. The heat transfer in an apartment building by natural convection was studied by Fan et al. [6]. They have studied the turbulent airflow with respect to different floor heights and interpreted the temperature of the floor. Saha et al. [7] investigated the heat flow pattern inside an attic of a building. In this study, they have considered a closed triangle enclosure having sinusoidal temperature variation on the sloping sides and an ambient temperature on the bottom side. They have studied the influence of aspect ratio and Rayleigh’s number. Computational fluid dynamics has been used for the above studies.

*Jeseema Nisrin J. Tel.: +91 97897 89853.
E-mail address: jeseema.nisrin@gmail.com
The current problem is considered to explore the fluid flow and temperature distribution in an enclosed room having a hot angle section. The flow considered is laminar, and the fluid motion is created as a result of buoyancy force induced inside the enclosure. The enclosure is assumed to have a wall of cool temperature and a hot inverted channel section acts as a conveyer platform of an industrial structure.

2. Problem Formulation:

The problem taken to study the fluid flow and temperature distribution is a square build enclosure having an outer wall of temperature $T_c$ and channel shaped section with temperature $T_h$ where $T_c < T_h$. The flow considered inside the enclosure is laminar natural convection induced by buoyancy force. The fluid inside the enclosure is considered with 0.71 Prandtl number. This property represents common air. The system is considered as two dimensional, and other perpendicular dimension is assumed to be long. The horizontal and vertical velocity components are assumed to be zero at the outer wall and inner channel wall as there is no cross flow across the boundaries. This constitutes the no-slip boundary condition. The outer wall and the inner channel section have a dimensionless temperature of 0 and 1 respectively. The width of the channel is assumed to 0.4 times the width of the enclosure, and its depth is 0.2 times the depth of the enclosure. The thickness of the channel section is 0.05 times the width of the enclosure. The intensity of heat transfer was varied by varying the Rayleigh number from $10^3$ to $10^6$. The schematic representation is shown in Figure 1

3. Governing Differential Equation

The airflow inside the room is governed by the Navier Stokes equation. The flow is assumed to obey the Boussinesq approximation. The fluid flow inside the room is assumed to be unsteady, laminar, and incompressible. The continuity, momentum, and energy equations are solved to evaluate the flow. The stream function vorticity formulation is used to study the problem. The equations are adopted as used by Kalidasan et al. [8] and KK Raja et al. [9]. The governing equations are in dimensionless form as in the above literature.

The dimensionless form of continuity, vorticity transport, and energy equations are

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$  \hspace{1cm} (1)

$$\frac{\partial \omega}{\partial t} + u \frac{\partial \omega}{\partial x} + v \frac{\partial \omega}{\partial y} = \frac{\partial^2 \omega}{\partial x^2} + \frac{\partial^2 \omega}{\partial y^2} + \frac{RaPr}{Pr} \frac{\partial T}{\partial x}$$  \hspace{1cm} (2)

$$\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right)$$  \hspace{1cm} (3)

$u$ velocity, $v$ velocity, vorticity, and stream function are related as follows

$$w = \frac{\partial \omega}{\partial t}$$  \hspace{1cm} (4)

$$u = \frac{\partial \psi}{\partial y}, v = -\frac{\partial \psi}{\partial x}$$  \hspace{1cm} (5)

Figure 1: Physical representation of the problem
The following Poisson equation gives the relationship between stream function and vorticity
\[
\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = -w
\]  
(6)

The vorticity at the boundaries was calculated using the following formula used by Adlam [10] and Das and Kanna [11].
\[
w_w = \frac{2(s_w - s_{w+1})}{\delta n^2}
\]  
(7)

Nusselt number (\(Nu\)) quantifies the heat transfer rate at the cavity boundaries. The local Nusselt number at the wall is calculated as follows
\[
Nu = -\frac{\partial \theta}{\partial n}
\]  
(8)

whereas the average Nusselt number of the respective wall and overall average of the cavity is calculated as follows
\[
\overline{Nu} = \frac{\int_1^1 \frac{\partial \theta}{\partial n} d\eta}{\int_1^1 \frac{\partial \theta}{\partial n} d\eta}
\]  
(9)

The convergence criteria for the defined problem is given by
\[
\sum_{i=0}^{i_{max}} \sum_{j=0}^{j_{max}} (\emptyset_{ij}^{n+1} - \emptyset_{ij}^{n}) \leq 10^{-5}
\]  
(10)

where \(\emptyset\) implies stream function, vorticity, or temperature.

4. Numerical scheme

A code is written in MATLAB to assess the thermal distribution and flow of fluid of the stated problem. The governing equations are solved by adopting the FDM. Initially, the enclosure region is divided into equally spaced grids in the \(x\) and \(y\) direction and the nodes are assigned. Next, the governing equations are discretized and the boundary conditions are imposed on the boundary nodes. Internal node values are solved by using Alternate Direct Implicit (ADI) scheme. In the first half-time \(x\)-direction sweep, the temperature equation (equation 3) is solved using the initial assumed values. In the second half-time \(y\)-direction sweep, the nodal values are updated with the previous results and the equation is solved. By the end of one iteration, all the nodal temperatures are updated and the values are utilized to calculate the vorticity from equation 2 in the same way, using the ADI method. These \(T\) and \(w\) values are used to solve equation 6 to find stream function followed by velocity (equation 5). This ends the first cycle. The updated values are utilized for the next iteration and the cycle is continued till it reaches the convergence, as given in equation 10.

5. Code validation

The reliability of the code is ascertained by verifying benchmark results of established problems published in the literature. The closed cavity free convection is validated by comparing the \(\overline{Nu}\) of the hot wall of differently heated cavity problem and the results are given in Table 1. The code with the central block inside the cavity is validated by comparing the results of \(\overline{Nu}_{avg}\) of the problem taken by Asan [12], shown in Figure 2. The results of the present code gives similar findings of the published work with limited tolerance as shown below.

| \(Ra\) | \(10^3\) | \(10^4\) | \(10^5\) |
|-------|--------|--------|--------|
| Davis[13] | 1.116  | 2.4519 | 4.519  |
| Kalidasan et al.[14] | 1.086  | 2.214 | 4.447  |
| Present | 1.124  | 2.293 | 4.508  |

| Deviation (%) | 0.72,3.46 | 2.64,3.56 | 0.24,1.37 |

Table 1: Validation of Natural convection in differently heated cavity
6. Results

The problem studied here is to find the influence of a hot channel section inside the cold enclosure. The thermal distribution, flow pattern of the streamlines, the horizontal and vertical components of velocity and rate of heat transfer inside the enclosure are investigated. The findings are discussed in the subsequent sections.

6.1 Streamlines

Streamlines are the path in which the massless fluid particles move along the flow. $Ra \ 10^3$ shows a passive movement of fluid around the central hot section.

Four vortices are formed inside the enclosure and the top two vortexes are big in size. This is due to the fact that the area of the heating section is large on the upper side compared with the bottom one. The movement of streamlines is equal on either side of the channel. It indicates that the heat transfer takes place by conductive mode.
and the buoyancy is not yet induced inside the enclosure. Ra \(10^4\) shows a similar trend except that the vortex bubble is pushed slightly to the top of the cavity. But the force is not enough to break the inertial force. When Rayleigh’s number increases further to \(10^5\), the buoyancy force increases, the fluid gets some potential energy to break the inertial force and the fluid moves upward. At Ra \(10^5\), the buoyancy force is higher within the laminar range and the streamlines are pushed to the top of the channel section, leaving the bottom part with cool fluid. These phenomena can be visualized from Figure 3.

6.2 Isotherms

Isotherms are contours connecting the points with the same temperature. The visualization of temperature flow inside the building enclosure having a hot section is done with the help of isotherms show in Figure 4. Inside the enclosure, the isotherms are concentrically formed and equally spaced around the channel section at lower Ra (Ra≤\(10^4\)). At this stage, the influence of buoyancy force is very less. This can be seen from the equally space isotherms moving around the hot section. The influence of buoyancy force begins when the Ra increased to \(10^5\) it makes the thermal contours move up, heating the upper portion of the enclosure, leaving the space below the channel section relatively cool. This phenomenon becomes high at Ra \(10^6\) and fluid recirculation begins. At higher Ra, the contour lines are crowded in between the space between the channel flanges. This indicates that the hot temperature is formed only around the close region of the channel due to the rising hot air plumes.

![Figure 4: Thermal distribution inside the enclosure (a) Ra \(10^4\) (b) Ra \(10^5\) (c) Ra \(10^6\) (d) Ra \(10^6\)](image)

6.3 Velocity

The horizontal and vertical components of velocity are calculated in midsection of the enclosure in x and y directions. The midsection velocity plots are shown in Figure 5 and Figure 6. The figures show that the velocity increase with the increase in the Ra. In Figure 5, it can be seen that the \(u\) and \(v\) velocity distribution at Ra≤\(10^4\) is negligible. It indicates the minor influence of buoyancy force and ascertains the fact that the temperature distribution, as seen in Figure 4 (a) and (b), is due to the conduction taking place inside the enclosure. The fluid flow in between the channel flange is dominantly the horizontal component of velocity, as seen in Figure 5 (a). From Figure 6, it can be noted that \(u\) velocity in the y direction is too small at the bottom portion of the enclosure, even at higher Ra. This is because the buoyancy force pushes the fluid particle upward, improving the \(v\) velocity leaving little horizontal movement of the particle at the base. The \(u\) velocity at Ra \(10^5\) is higher at the top of the cavity, and this leads to the formation of hot air plume of temperature contour, as seen in Figure 4 (d).
6.4 Rate of heat transfer

The rate of heat transfer in the building compartment is estimated by calculating the Nu value using equation 9. The local Nusselt number is calculated on the left wall and the values are plotted in Figure 7.

The figure shows that the heat transfer increases with each increments of Ra more dominantly at the upper part of the enclosure. As the fluid moves up at the higher Ra, the heat transfer is decreased in the bottom portion of the enclosure. This can also be seen in Figure 8, which shows the variation of Nu on the outer wall with respect to S-coordinate as used by Saeidi and Khodadadi [15]. It shows an equal amount of heat is transferred to all the outer wall at lower Ra and the heat transfer rate decrease to the bottom wall and increases to the other three side walls at higher Ra.
7. Conclusions

Natural convection inside a cold enclosure having a hot central channel section is studied. Four variations of Rayleigh numbers have been considered as $10^3$, $10^4$, $10^5$, and $10^6$. The study shows that conductive mode of heat transfer takes place at lower $Ra \leq 10^4$ and convection is induced at higher $Ra \geq 10^5$. The higher buoyancy force pushes the hot fluid particles upward, leaving a cool temperature below. The streamline movements are clockwise on one side of the hot section and anticlockwise on the other side. The lower Rayleigh number shows an equal amount of heat transfer on all four sides of the compartment. At the top wall, the rate of heat transfer is maximum at higher $Ra$. The $u$ and $v$ velocity of the fluid variation at the upper half of the enclosure results in the formation of thermal recirculation. The results showed that the heat flow pattern depends on the intensity of the heating source and it does not require driving force to induce fluid movement. The findings will be helpful in prediction of temperature flow in the hot platform of industrial structure and thermal distribution prediction in case of fire.

References

[1] “Energy Efficiency Indicators – Analysis - IEA,” [Online]. Available: https://www.iea.org/reports/energy-efficiency-indicators. [Accessed: 16-Feb-2021].

[2] S. Ostrach, “Natural Convection in Enclosures,” Adv. Heat Transf., vol. 8, no. C, pp. 161–227, 1972.

[3] G. de Vahl Davis, “Laminar natural convection in an enclosed rectangular cavity,” Int. J. Heat Mass Transf., vol. 11, no. 11, pp. 1675–1693, 1968.

[4] P. Bhave, A. Narasimhan, and D. A. S. Rees, “Natural convection heat transfer enhancement using adiabatic block: Optimal block size and Prandtl number effect,” Int. J. Heat Mass Transf., vol. 49, no. 21–22, pp. 3807–3818, 2006.

[5] M. M. Mousa, “Modeling of Laminar Buoyancy Convection in a Square Cavity Containing an Obstacle,” Bull. Malaysian Math. Sci. Soc., vol. 39, no. 2, pp. 483–498, 2016.

[6] Y. Fan, Y. Li, J. Hang, K. Wang, and X. Yang, “Natural convection flows along a 16-storey high-rise building,” Build. Environ., vol. 107, no. May 2018, pp. 215–225, 2016.

[7] S. C. Saha, J. C. Patterson, and C. Lei, “Natural convection and heat transfer in attics subject to periodic thermal forcing,” Int. J. Therm. Sci., vol. 49, no. 10, pp. 1899–1910, 2010.

[8] K. Kalidasan, R. Velkennedy, and P. Rajesh Kanna, “Buoyancy enhanced natural convection inside the ventilated square enclosure with a partition and an overhanging transverse baffle,” Int. Commun. Heat Mass Transf., vol. 56, pp. 121–132, 2014.

[9] K. K. Raja, P. R. Kanna, and M. K. Das, “Numerical simulation of mixed convection in a two-dimensional laminar plane wall jet flow,” Numer. Heat Transf. Part A Appl., vol. 52, no. 7, pp. 621–642, 2007.
[10] J. M. Adlam, “Computation of two dimensional time-dependent natural convection in a cavity where there are internal bodies,” vol. 14, no. 2, pp. 141–157, 1986.

[11] M. K. Das and P. R. Kanna, “Application of an ADI scheme for steady and periodic solutions in a lid-driven cavity problem,” Int. J. Numer. Methods Heat Fluid Flow, vol. 17, no. 8, pp. 799–822, 2007.

[12] H. Asan, “Natural convection in an annulus between two isothermal concentric square ducts,” vol. 27, no. 3, pp. 367–376, 2000.

[13] G. De Vahl Davis, “Natural convection of air in a square cavity: A benchmark numerical solution,” Int. J. Numer. Methods Fluids, vol. 3, no. 3, pp. 249–264, 1983.

[14] K. Kalidasan, R. Velkennedy, and P. Rajesh Kanna, “Numerical investigation on natural convection inside the side ventilated square enclosure with vertical mid-partition,” Numer. Heat Transf. Part A Appl., vol. 66, no. 12, pp. 1389–1418, 2014.

[15] S. M. Saeidi and J. M. Khodadadi, “Forced convection in a square cavity with inlet and outlet ports,” Int. J. Heat Mass Transf., vol. 49, no. 11–12, pp. 1896–1906, 2006.