International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 8 **Issue:** VIII **Month of publication:** August 2020

DOI: https://doi.org/10.22214/ijraset.2020.30917

Modelling Radiation and Natural Convection Heat Transfer in a Square Enclosure using Surface to Surface Radiation Approach

Karri Maheswari[1], G Sunil[2], K. Ravindra[3], U. S. Ramakanth[4]
[1, 2, 3]*M. Tech Student, Department of Mechanical Engineering, #Visakha Technical Campus, Vizag, AP*
[4]*Associate Professor, Lendi Institute of Engineering and Technology, Vizianagaram, AP*

Abstract: Numerical simulations were used to investigate natural radiation and convection interactions in small rooms with three-dimensional geometry. The opposite vertical walls are heated and cooled, while the other walls are supposed to be fixed. Governance flow, momentum equations, and radiative conversion are solved using Ansys Fluent CFD. In estimating thermal terminology, a second-degree wind diagram and a dual solution algorithm are used. The cubic box is filled with air and the flow is considered protective. Air characteristics are assumed to be constant, except for the difference in density in which a Bosnic approximation is used. The surface heat model (S2S) is used as the radiation transfer model. The arithmetic field is a square container Length of the edge = 0.225 m In a surrounding environment at T = 293 K. One of the walls of the case is heated up to 473 K. Note that the radiation from the opposite wall is high, with the improved model. Radiant heat transfer is reduced.
Keywords: Radiation, conduction, Discretization, Laminar, Temperature

I. INTRODUCTION

Natural convection and heat transfer by heat radiation in fluid-filled cavities have received much attention in recent years due to its relationship to the thermal performance of engineering applications such as cooling electronic components, electrical boxes, assembler designs, solar designs, heat exchangers, etc. . Therefore, the common characteristics of natural convection and heat transfer by heat radiation are more important. The study of heat transfer in the room was the main topic of the researchers due to a wide range of applications in engineering and working life. It finds its application from cooling the nuclear reactor to removing heat from the micro-electronic components as well as for designing the room air conditioning, thermal design for commercial buildings, cold storage, oven and many more. In addition to a wide range of applications, the simultaneous thought of wall conduction and convection in fluid flow remains an exciting area of research in recent decades. Several articles can be found in the literature to study natural convection in the can. There are many boundary conditions that can be taken into consideration, but the most complex and practically applicable area of least study has been found in the literature. Most of the areas already studied are bottom warming or lateral warming. But in this study, we studied a viable limit state with associated heat transfer. Energy has long been a major topic of discussion among researchers. During any development program for any technology, energy consumption has always been of great importance. An important part of the total energy consumed in the useful life of any building is the operating capacity. This involves maintenance and, most importantly, the energy involved in maintaining the building in comfortable thermal and visual conditions. The building energy analysis tool used before starting any project aims to reduce this operating capacity. It is important to understand that this decrease in energy consumption must be achieved without regard to low performance. Therefore, the building should provide a comfortable environment compared to its external environment. Previous studies have shown the fact that the passenger quickly responds to any discomfort to restore his comfort, but this can negatively affect energy consumption. Therefore, accurate prediction of thermal comfort is very important when designing a building to maintain less energy consumption.

A. Definition of the Problem

When we talk about the box here, it covers a very wide area from warehouse to a small cold storage room where the heat should be preserved. The current study was driven by the fact that although most researchers work to improve the efficiency of refrigeration devices, very few of them are interested in developing a thermally optimized container or environment that can reduce the load on refrigerators. Thus reducing energy consumption. Non-dimensional allows us to apply the obtained result to any scale and depends only on the non-dimensional number. Moreover, the thought of conjugated heat transfer allows us to understand its effect and gives us a true analogy with the real system. The influence of different parameters, with or without insulation, is studied in the current work. This comparison was made in the current study and the result gives us a clear idea of the standard scope to consider when designing any cold storehouse or commercial building.

B. Objectives and Methodology

The main objective of this work was to study the effect of the various parameters on the heat transfer and flow characteristics of the air trapped inside of the enclosure. The CFD model is developed for this problem and a non-dimensionalised study was carried out so that the model could be applied for various scale. The finding of the current work will also present a reference for further developing of CFD model for enclosure with conjugate heat transfer.

II. LITERATURE SURVEY

Various traditional cooling concepts are used in different combinations to properly cool the turbine blades and blades. Gas turbine cooling technology and heat transfer by Han, J. C., et al. (2001) provides a detailed description of the heat transfer and cooling technology in the turbine blade. The author has compiled a comprehensive review of the technology for gas turbine cooling that includes techniques for improving heat transfer in internal cooling channels. The book also includes many studies conducted over the years in a wide range of rib configurations in different cooling channels using many experimental techniques.The first studies examined cooling channels using orthogonal ribs. Han, J.S. (1988) performed a pilot study on grooved channels with orthogonal square ribs. Three different polygonal channels with different aspect ratios were considered. The distances between the sides and the Reynolds number were also taken into account. The work collected a detailed study on the effect of the rib spacing on the heat transfer performance of the slotted channels. The author has also provided us with heat transfer and friction correlation. This article provides us with the experimental results that were compared to the numerical study obtained through numerical analysis.[1] Han, J. C. et al. (1992) now performed multichannel experiments with angled ribs. The ribs are oriented to angles 30, 45, 60, and 90. The main objective of the study is the effect of the direction of the ribs on thermal performance. It was concluded that angled ribs had a higher heat transfer performance than orthogonal ribs. The friction and heat transfer correlations are derived from their studies. 4 4[2] Iacovides, H. (1998) did the math on polygon rectangular paths. The fixed and circular rectangular sections were considered in his study. The work of the calculation was based on the square orthogonal ribs. The work mainly focused on the impact of the turbulence model on channel performance. A differential tension model was developed that proved to have produced better results than the standard k-e model. [3] Bonhoff, B., et al. (1999) He conducted an experimental and numerical study in cooling channels with 45 ° ribs. A stationary channel is used in the study. Experimental and numerical results are obtained using Reynolds pressure model. The results were well compared. The case of periodic boundaries was used for numerical study. Speed distribution and heat transfer were recorded. [4] Iacovides, H. and Raisee, M. (2000) performed a computational study on a rough rib pathway using the Reynolds low number disorder model. Standard k-e and k-w standard turbulence models were used. The periodic limit condition was used and parameters such as the Nusselt number were obtained. The differential pressure model was also developed. The DSM model provided improved predictions of heat transfer after re-joining the flow and on the ribs. However, the model was unable to predict the impact of Reynolds' number.[5] Lin, Y. L., et.al (2001) conducted a numerical study of flow and heat transfer in a 45-degree channel with ribs. Rotary and non-periodic channels were considered. The analysis was performed in 3D flow. With 3D analysis, secondary flow was predicted. These results also explain how the nature of the fluid flow affects the heat transfer on the surface. Moreover, the secondary flow has clear effects on heat transfer. 5 5 [6] Agarwal, P. (2001) conducted a detailed study of heat transfer and flow in serpentine cooling channels. The channel was constructed with a aspect ratio of 1: 4 and 4: 1. The ribs were used at an angle of 90 and 45 degrees. Comparisons were made between the angle direction and the effect of channel aspect ratios. Slanted ribs showed that the heat transfer performance was higher than orthogonal ribs. Also, the wider channels proved to be better than the narrow channel.[7] conducted by Bridberg, c. (2002) An Intensive Study on Modeling Turbulence in Internal Cooling of Gas Turbine Blades. The fixed and rotary channels are taken into consideration. It was concluded that the standard k-w model produced an accurate method for simulating complex geometric figures.[8] Wright conducted to. M. (2008) Experimental Analysis on 3: 1 Rectangular Channel with Angled Ribs. Spacing between 10 and 20 sides was considered, the width of the sides was adjusted and their effect on performance was recorded. The influence of Reynolds number is also taken into account. The study concluded that the angled channel has the effect of heat transfer more than the soft channel. Heat transfer and friction links were obtained. Many researchers such as Al-Qahtani, a. M. (2002), Han, J. These works were useful for studying the effect of rib spacing, rib width, and flow parameters on heat transfer performance. The works gave an idea of how to change the heat transfer when the channel hardened. The amount of particles present hardens the matrix by increasing the content of fly ash and B4c particles. Therefore, the observation was more resistant to corrosion. The MMC welding machine wears a lot with a smaller fraction of weight of fly ash and B4c particles, and the corrosion increases linearly over time. [9-11].Han and Chen presented a summary of numerical and experimental research on the internal cooling steps of a turbine blade using rib inverters.

They highlighted, on average, the ability of the RANS models to solve the 3D complex. Flow physics in the internal cooling paths of the rotary turbine blade. Murata and Mochizuki compared laminar heat transfer and turbulence in a fixed square channel with angled or transverse rib turbines. The channel heat transfer is simulated numerically using the second order finite difference method. In turbulent cases using 60 ° and 90 ° ribs, reflux at the midpoint between the ribs and the unstable reverse flow against the rib formed high values of heat transfer at the average time. In the case of laminar flow, the low-torque fluid near the grooved wall has a lower impact on the flow field. Viswanathan and Tafti performed separate spiral simulations (DES) for turbulent flow and heat transfer in a two-step internal cooling channel. The simulation was applied to a 90 ° channel fixed channel. The analyser used a generic system that does not compress direct digital simulation and a great simulation of turbulence disorder using the DES version of the k-modelo model. "The flow in the space between the ribs is dominated by the heat transfer enhanced by the vortex in front of the rib, in the anchorage area behind the rib and at the intersection of the rib with the smooth walls[12-13]."

III. MATHEMATICAL FORMULATION

Initially, a simple square enclosure is filled with air with certain boundary conditions. Then an insulation is provided directly below the upper wall to study its effect. All four boundaries have different and practical boundary conditions as shown in Figures 1.1. The two walls, the upper wall and the right wall, are supposed to have some limited thickness to enforce the boundary conditions of the conjugate. The lower wall is assumed to be thermally insulated while the upper wall is assumed to have a constant hot temperature. The left isothermal wall is maintained at a low temperature and the right wall has a thermal reaction with air in environmental conditions. Three different cases were considered by changing the thickness of both boundary walls simultaneously and for each case the Reynolds number and the conductivity of the solid wall varied to study the effect of these two parameters on the flow field and temperature of the liquid in the enclosure. The insulation is provided at a height of 0.9 of side length and the thickness of the insulation is as small as 0.01.

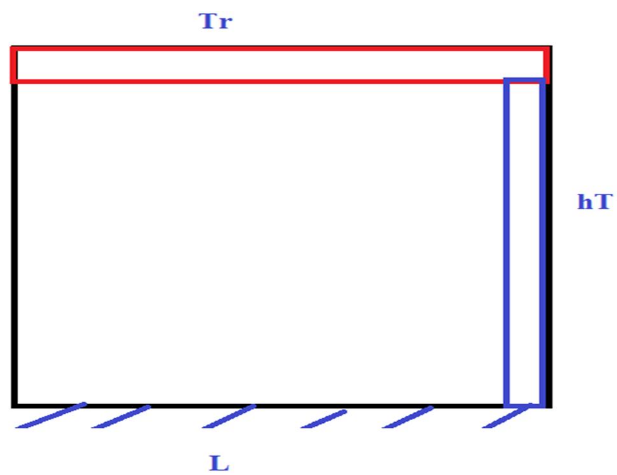


Fig 1.1: Boundary conditions of domain

The field is assumed to be two-dimensional with a natural convection heat transfer mode. The velocity components along the x and y directions are u and v, respectively. All liquid properties in the can should be stable. The density is taken as constant except for the term of buoyancy governed by the Boussinesq approximation. Moreover, heat transfer by radiation is assumed to be negligible compared to other modes of heat transfer, and hence the radiation conditions are neglected.The four boundaries of the square field are maintained under different boundary conditions. The left border is kept at the lowest temperature, that is, the cold wall. The upper wall is considered to be at a high temperature throughout the area. The heat loss is considered to be convective through the right wall, which is also of limited thickness. The bottom of the case should be completely insulated. Figure 1.1 shows the boundary conditions considered in case of isolation and without insulation respectively.

The method most used in the past is based on the Patankar method. This method is based on the assumption of the high viscosity value of the solid region when considering the conjugated heat transfer. Therefore, the entire field can be considered one and with this assumption the term velocity in the solid region can be neglected and simultaneous solution of the equation possible. Therefore, the energy equation for both regions is solved, but the moment equation is solved for the fluid region. A few other methods developed in the recent past include the submerged boundary case and the imaginary node method.

The method adopted here to solve the problem is known as the multi-block method. This application of the method on the problem of conjugate boundaries has not appeared in the literature before. The method followed above was based on the assumptions of S.V Patankar. However, the method adopted here is not without guesswork. In the multi-block method, the field is divided into an appropriate number of blocks separating the fluid region from the solid region. This allows the respective equation to be applied to the respective masses i.e. only the moment equation for the fluid and energy equation for all regions.The term diffusion is defined using a centralized difference diagram and the variables are stored according to the order placed. The governing equations are solved using finite size approach. The semi-implicit method of pressure related equation (SIMPLE) is used to relate the moment and continuity equations.

IV. NUMERICAL ANSLYSIS

Numerical simulations were used to investigate radiation and natural convection interactions in small enclosures of three-dimensional geometries. There are two cases taken for this analysis. The two model's geometry was shown in Fig 1.2. (1) square enclosure model,(2)square enclosure with obstructions model. The design of enclosure and enclosure with obstructions model was developed in the SPACE CLIAM Software in the ANSYS WORKBENCH, the steps are given below.

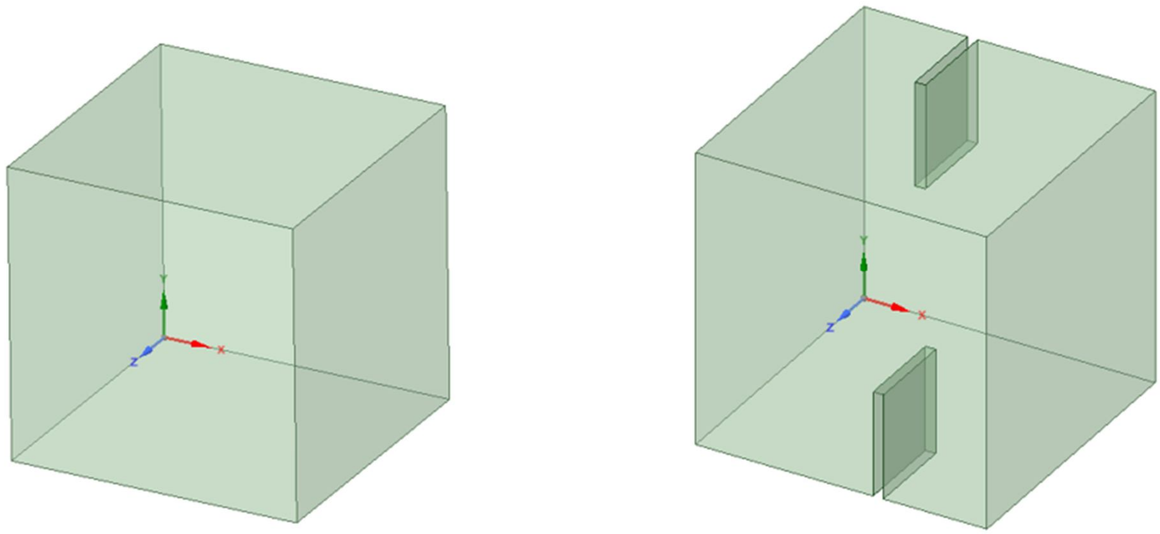


Fig 1.2 Geometry of the two models

A. Boundary Conditions

1) *Set the boundary conditions for the cold wall in X & Z direction, side wall Z0 & Z1 as follows*
a) Thermal Conditions = Mixed
b) Heat Transfer Coefficient (w/m2k) = 5
c) Free Stream Temperature (k) = 293.15
d) External Emissivity = 0.75
e) External Radiation Temperature = 293.15
f) Internal Emissivity = 0.95
g) Wall Thickness (m) = 0.05
h) Material Name = insulation
2) *Set the boundary conditions for the cold wall in Y direction, side wall Z0 & Z1 as follows*
a) Thermal Conditions = Mixed
b) Heat Transfer Coefficient (w/m2k) = 5
c) Free Stream Temperature (k) = 293.15
d) External Emissivity = 0.75
e) External Radiation Temperature = 293.15
f) Internal Emissivity = 0.95
g) Wall Thickness (m) = 0.05

In the Operating Conditions panel, set the gravity in the negative Y direction. The result is that for each iteration, the analyzer would calculate and use the average volume density for the run intensity. The operating temperature is only used if the Boussinesq is chosen for the density in the material plate, so it can be ignored here. Figure 1.3 represents the boundary condition.

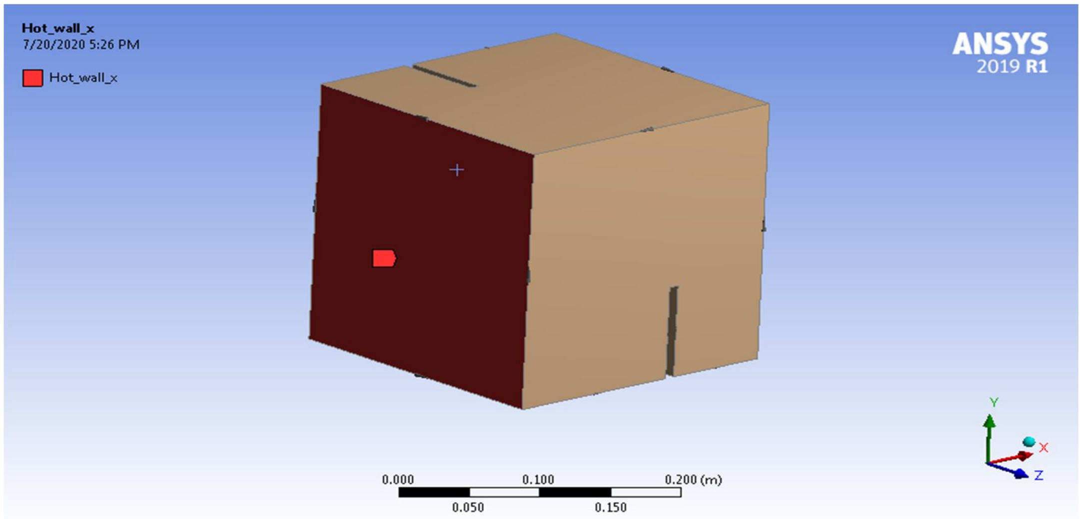


Fig 1.3 Hot wall of temperature 473K, internal emissivity of 0.95 -Boundary Condition

V. RESULTS AND DISCUSSIONS

A small square enclosure and square enclosure with obstructions of two types are consider and to investigate the radiation and natural convection using ANSYS FLUENT

A. Case-1: Analysis Results of square enclosure

1) Energy Balance

Total Heat Transfer Rate	(w)
bottom_wall_y	-9.8931248
cold_wall_x	-10.267297
hot_wall_x	51.287312
side_wall_z0	-10.435696
side_wall_z1	-10.435153
top_wall_y	-10.316431
Net	-0.060389131

The net value is -0.06 W which is almost zero. The energy input from hot wall is 51.3 W. The energy imbalance is 0.1%. Good energy balance based on Hot wall.

Radiation Heat Transfer Rate	(w)
bottom_wall_y	-10.22223
cold_wall_x	-8.5860589
hot_wall_x	41.031198
side_wall_z0	-8.4872936
side_wall_z1	-8.4769387
top_wall_y	-5.3045668
Net	-0.045889545

Radiation at hot wall is 41 W, the total heat supplied is 51.3 W. from which it indicates radiation is dominant heat transfer. Temperature contour on all walls. As expected from any cross section parallel to heat wall, temperature increases with y-coordinate with a raise in temperature of 118^0 K on cold wall_x purely due to radiation. Fig 1.4 and 1.5 shows the Temperature and heat flux contour

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

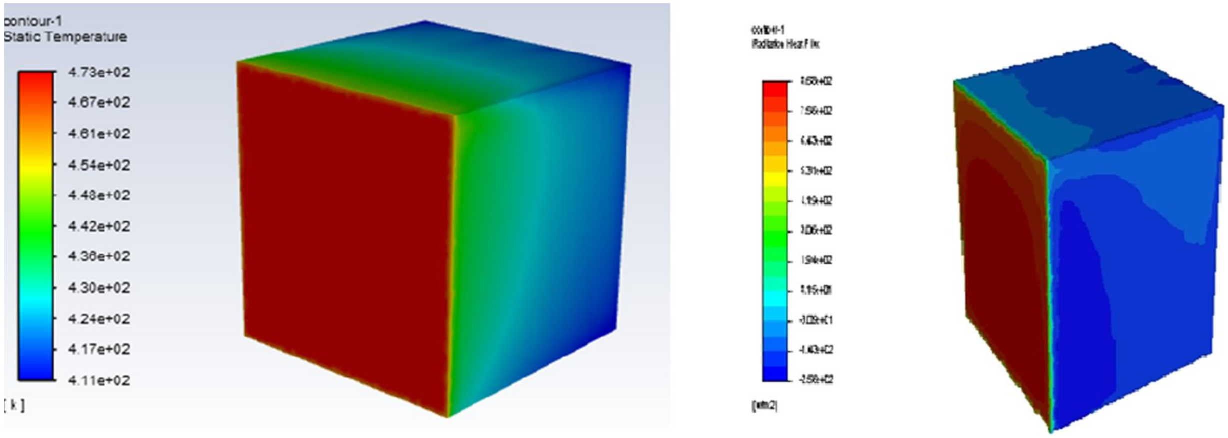


Fig 1.4 Temperature and heat flux contour

B. Case-2: Analysis Results of Square Enclosure with Obstructions

1) Energy Balance

Total Heat Transfer Rate	(w)
bottom_wall_y	-9.3986227
cold_wall_x	-9.8013784
hot_wall_x	53.914283
side_wall_z0	-9.9515201
side_wall_z1	-9.9973941
top_wall_y	-9.9386402
wall_internal	-4.8859468
Net	-0.059219488

The net value is -0.059 W which is almost zero. The energy input from hot wall is 54 W. The energy imbalance is 0.11%. Good energy balance based on Hot wall similar to case 1.

Radiation Heat Transfer Rate	(w)
bottom_wall_y	-10.446014
cold_wall_x	-7.5035002
hot_wall_x	42.368994
side_wall_z0	-8.1536306
side_wall_z1	-7.916904
top_wall_y	-4.5165947
wall_internal	-3.8808506
Net	-0.048499821

Radiation at hot wall is 42.36 W, the total heat supplied is 54 W. from which it indicates radiation is dominant heat transfer.

From cold_wall_x to:	Viewfactor	Incident Radiation
bottom_wall_y	0.181785	330.588792
hot_wall_x	0.171239	541.902574
side_wall_z0	0.188375	339.608558
side_wall_z1	0.179242	327.908618
top_wall_y	0.189932	369.429245
wall_internal	0.089427	93.197711

The view factors slightly reduced to 0.18 when compared to case 1. The incident radiation from cold wall_x to hot wall_x is higher but decreased in comparison with case 1 due to obstructions placed in square enclosure with decrease in view factor which indicates the net outgoing radiation is decreased due to incident and reflected radiation is attenuated.

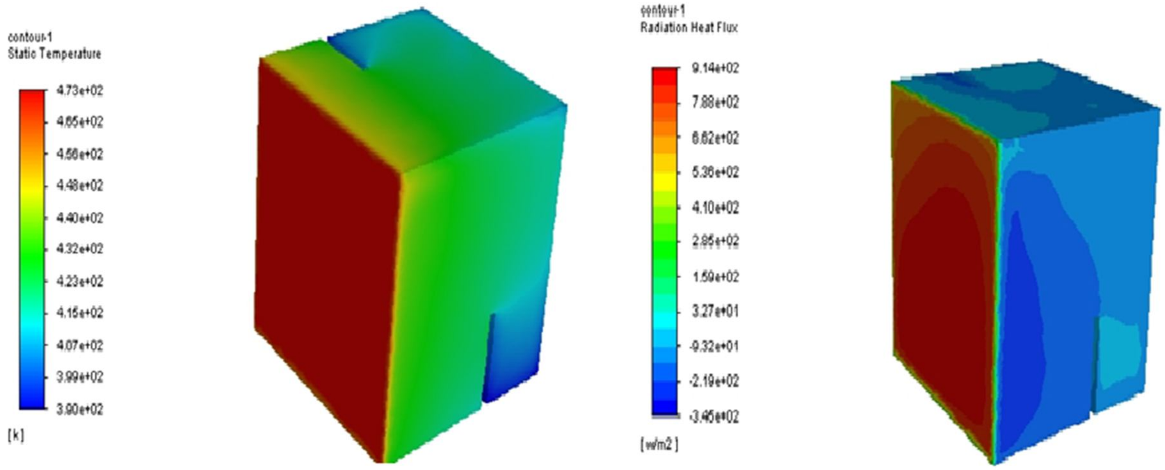


Fig 1.5 Temperature and heat flux contour

Regardless of the surface irradiance, the mean overall heat and radiative transfer and convection increased the cold wall temperature of the hot wall. Radiation is the predominant heat transfer of the overall thermometer. From numerical simulation, a good energy balance is achieved on the basis of the thermal wall (0.1% energy imbalance). The 3D numerical model was resolved using ANSYS FLUENT with a surface-to-surface radiation model with a conjugate solution. Mostly, the radiation falling on the hot wall is higher but in the second case with obstruction it is reduced by about 22%. The temperature in the cold wall increased due to the radiation and convection heat transfer from room temperature to 4110 K, but in Case 2 it decreased to 3900 K. Heat transfer by radiation prevailing in a closed room exposed to high temperatures on one side of wall surfaces.

REFERENCES

[1] M. F. Baigand A. Masood, "Natural convection in a two-dimensional differentially heated square enclosure undergoing rotation", *Numerical Heat Transfer Part A: Applications*, vol. 40, pp. 181-202, 2001.

[2] M. Rahmanand M. A. R. Sharif, "Numerical study of laminar natural convection in inclined rectangular enclosures of various aspect ratios", *Numerical Heat Transfer Part A*, vol. 44, pp. 355-373, 2003.

[3] L.F. Jin, K.W. Tou, C.P. Tso, "Effects of rotation on natural cooling from three rows of heat sources in a rectangular cavity", *International Journal of Heat and Mass Transfer*, vol. 48, pp. 3982-3994, 2005.

[4] T.S. Cheng and W.H. Liu, "Effects of cavity inclination on mixed convection heat transfer in lid-driven cavity flows", *Computers & Fluids*, vol.100, pp.108-122, 2014.

[5] H. Saleh and I. Hashim, "Conjugate natural convection heat transfer in a rotating enclosure", *Journal of Applied Fluid Mechanics*, vol. 9, pp. 945-955, 2016.

[6] H. Li, S. Tong, "Natural convective heat transfer in the inclined rectangular cavities with low width-toheight ratios", *International Journal of Heat and Mass Transfer*, vol. 93, pp. 398-407, 2016.

[7] U. V. Awasarmol and A. T. Pise, "An experimental investigation of natural convection heat transfer enhancement from perforated rectangular fins array at different inclinations", *Experimental Thermal and Fluid Science*, vol. 68, pp. 145-154, 2015.

[8] N. Ramesh and S.P. Venkateshan, "Effect of surface radiation on natural convection in a square enclosure", *Journal of Thermophysics and Heat Transfer*, vol. 13, pp. 299-301, 1999.

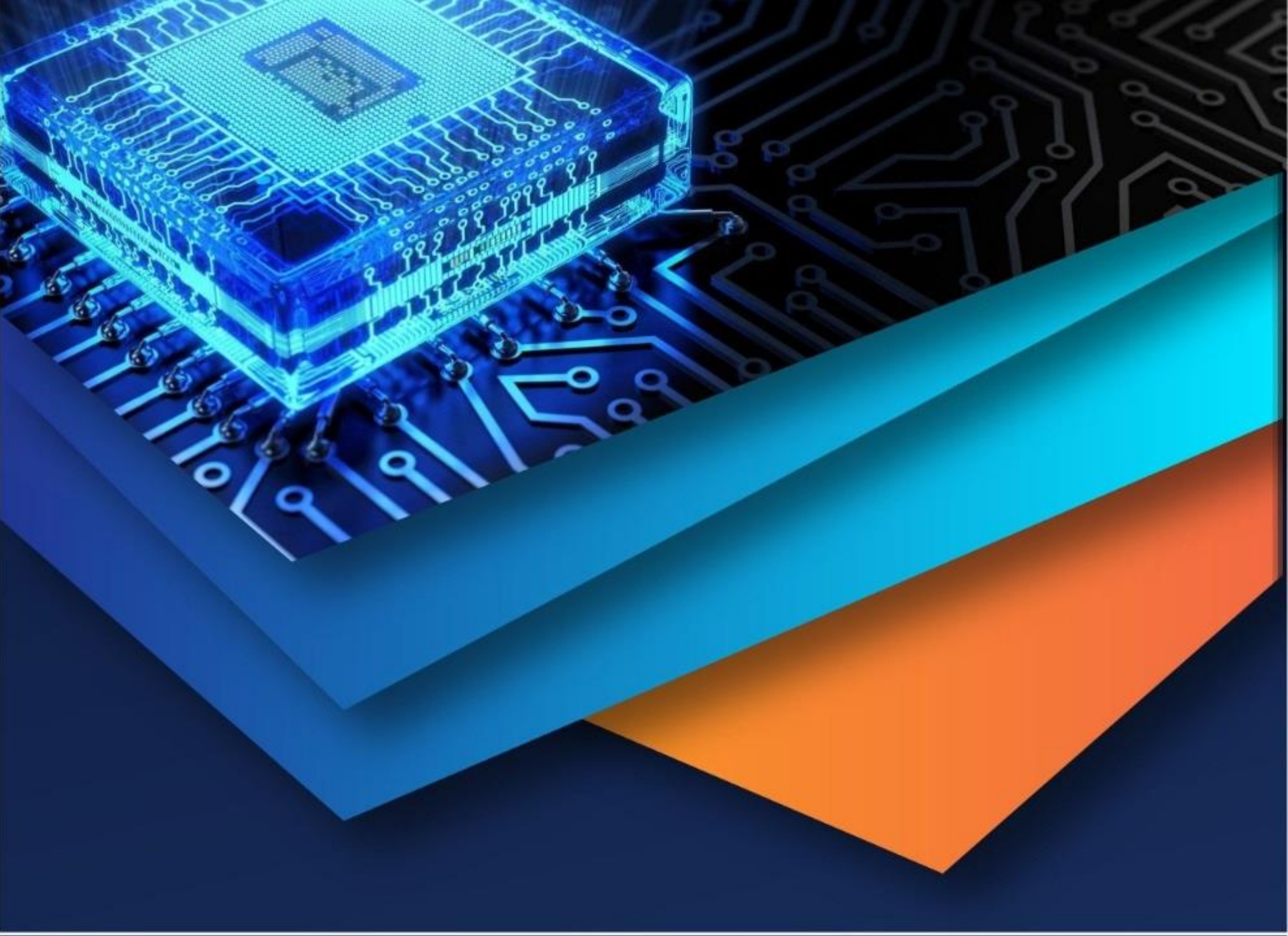
[9] H. Bouali, A. Mezrhab, H. Amaoui, M. Bouzidi, "Radiation-natural heat transfer in an inclined rectangular enclosure", *International Journal of Thermal Sciences*, vol. 45, pp. 553-566, 2006.

[10] M. Rabhi, H. Bouali, A. Mezrhab, "Radiation-natural convection heat transfer in inclined rectangular enclosures with multiple partitions", *Energy Conversion and Management*, vol. 49, pp. 1228-1236, 2008.

[11] H.F. Nouanegue, A. Muftuoglu, E. Bilgen, "Heat transfer by natural convection, conduction and radiation in an inclined square enclosure bounded with a solid wall", *International Journal of Thermal Sciences*, vol. 48, pp. 871-880, 2009.

[12] U. S. Ramakanth & Putti. Srinivasa Rao, "Wear Behavior of Al 7075/Fa/Sic Hybrid Composites", International Journal of Mechanical and Production Engineering Research and Development (IJMPERD) ISSN(P): 2249-6890; ISSN(E): 2249-8001 Vol. 9, Issue 2, Apr 2019, 331-338.

[13] U. Ramakanth & Putti. Srinivasa Rao, "Corrosion Studies on Ceramic Particulates Reinforced Hybrid Metal Matrix Composites", International Journal of Mechanical Engineering and Technology (IJMET) Volume 10, Issue 01, January 2019, pp. 661–666.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089 (24*7 Support on Whatsapp)