iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 4 Issue: IV Month of publication: April 2016

DOI:

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Impact Of Radiation Models In CFD Simulations Of Natural Convection Heat Transfer In A Side Heated Square Cavity

Sudha Thakur[1], Saurabh Kumar[2]
[1,2]Department of Mechanical Engineering, Raipur Institute of Technology Raipur, Chhattisgarh, 492101 INDIA

Abstract— ***The flow progress in a water-filled square cavity which is suddenly heated and cooled on the opposing sidewalls is noticed by using the CFD software package Fluent 6.3. Two approximations were used for CFD simulation of radiation in a liquid cavity, namely, Rosseland approximation the methods of spherical harmonic functions. Comparison of three different approaches to description of the radiation mechanism of energy transfer has made it possible to recognize the special features of these models. Temperature and stream distribution functions are acquired in a large range of the governing criterion. The horizontal boundaries of the cavity are insulated, while the heated and cooled vertical left and right walls are conducting. The present thesis continues the explanation of the flow growth until a steady state is reached. A clear visualization of the interaction between the second wave group and the laminar interruption flow across the roof of the cavity is given. The consequent collapse of the horizontal interruption cause the thermal layer in the cavity core and the adjustment of the vertical boundary layer are also observed. The final state is represented by waves which continuously travel along the boundary layers.***
Keywords— ***CFD, Natural Convection, Radiation, Rosseland approximation***

I. INTRODUCTION

Convection by natural means is crucial to Flows, in both nature and technology. There are a variety of real-world applications of free convection, such as insulation, cooling of electronic equipment, solar energy devices, nuclear power reactor, energy recovery systems, refrigeration and AC ventilation, etc. The fluid flow and heat transfer behavior of such systems can be predicted by Navier-stroke i.e. the mass, momentum, and energy conservation equations with appropriate boundary conditions. The fast-outcome branch of computational fluid dynamics (CFD) simulation of fluid flow and heat transfer features. A extensive analysis of the fluid flow and heat transfer patterns in fundamentally quiet geometries, such as the buoyancy-driven square cavity, is a necessary outcome of the evolution of better designs for more complex industrial applications. When a fluid-filled rectangular cavity is differently heated and cooled from the opposing sidewalls, a complex flow develops. This is one of the classical heat and mass transfer problems with understanding in fundamental fluid mechanics, as well as for engineering and geophysical appliance. Technical cooling systems, crystal boost procedures, building insulation and buoyancy-induced horizontal mass transfer in geophysical flows all fall into this category. In many of these applications, the trick of the horizontal temperature gradient is time-dependent, therefore an understanding of both the temporal development of the flow and of the corresponding heat transfer properties is as equally valuable, as is the analysis of the final steady state. Most of the more recent research, both experimentally and numerically, has been focused on either the transient start-up flow following the sudden heating and cooling of the side walls [l-3]. The experimental system under consideration is that of an initially isothermal and motionless fluid, which is contained in a square cavity with thermally conducting vertical sidewalls and insulating top and bottom boundaries. The flow is started by suddenly raising and lowering the temperatures of the opposing sidewalls by the same amount ΔT, and then maintaining those temperatures. The evolving flow consists of narrow vertical boundary layers adjacent to the heated and cooled walls, which exit into horizontal intrusions travelling across the roof and floor of the cavity before meeting with the opposing wall boundary layers. The complex interactions between the incoming intrusions and the receiving boundary layers establish the means by which the core region of the cavity stratifles. For a sulficiently low Rayleigh number, a laminar time-independent flow is eventually achieved. At a higher Rayleigh number, however, a time-periodic flow may result, or if the Rayleigh number is sutllciently high, the flow becomes turbulent Among the many phases of the flow development from the motionless and isothermal condition to one characterized by a variety of length and velocity scales, one particularly striking sequence of events in the early part of the flow has attracted a great deal of recent attention. That is the presence of two separate groups of short-lived travelling waves observed on both the vertical boundary layers and the intrusion flows. The

Volume 4 Issue IV, April 2016
ISSN: 2321-9653

first group of waves follows start-up, and the second group emerges from the first interaction between the incoming intrusion and the opposing boundary layer. The objectives of the present study are the following: (1) To introduce the radiation modles in the natural convection problems (2) to present impact of radiation modles, for the entire laminar natural-convection range of 103 ≤Ra ≤108; and (3) to present a focused and elaborate study on the problem of the buoyancy-driven cavity, which also looks into some of the discrepancies observed in the literature. T. d. upton and D. w. watt [1999] The angle of inclination was shown to have a significant effect on the flow and heat transfer in natural convection in an enclosure. Buoyancy in the intrusion layer was found to be the main factor determining the character of these flows. The scale analysis for transient natural convection in enclosures was extended to inclined enclosures and compared to the experimental results. Neither the boundary layer nor the whole cavity transient oscillations were evident. The results for the temperature in the boundary layer and the Nusselt number at the wall and center of the enclosure have the appearance that they were produced by a lower Rayleigh number flow.

Liaqat and Baytas[2000] Transient Congugate free convection analysys for a squre enclosure having thick walls have been performed. Result Indicate a strong effect of the wall conduction for thick walled enclose. It has been shown that a higher value of diffusivity ratio plays a significant role in cooling the fluid contained in the enclosure along with conductivity ratio.

Tim Persoons et al.[2011] By combining particle image velocimetry (PIV) and local convective heat transfer measurements using hot-film anemometry, a methodology has been established to simultaneously study fluid dynamics and natural convection heat transfer from a pair of horizontal cylinders. Multiple pulse separation PIV has been successfully applied to increase the dynamic range in measuring the velocity and turbulence fields. In typical conditions, a 5.5 ×× increase in dynamic velocity range has been obtained, resulting in more accurate flow and turbulence fields.

II. RESEARCH METHODOLOGY

This section defines the buoyancy-driven cavity problem, including the governing equations (CFD) and the boundary conditions. By using Ansys Fluent Software employed to solve the problem under investigation. The theoretical framework behind the discrete singular convolution is elaborated. The method of solution for the integration of the Navier-Stokes and energy equations by the Radiation models explained.

A. Problem Description

The CFD simulation reported here have been carried out with water as the working fluid. The mean temperature ranged from T0 = 20.4 to 20.8°C, giving Prandtl numbers between Pr = 7.0 and 6.9. The temperature differences between the vertical sidewalls and T0 usually ranged from AT = 3.2 to 3.5 K correspondingto Rayleigh numbers between Ra = responding to Ra = 7.2 x 108. Gravity acts downwards. A buoyant ow develops because of thermally-induced density gradients. The medium contained in the box is assumed to be absorbing and emitting, so that the radiant exchange between the walls is attenuated by absorption and augmented by emission in the medium. All walls are black. The objective is to compute the flow and temperature patterns in the box, as well as the wall heat flux. The values of physical properties used by the Boussinesq model and operating conditions (e.g., gravitational acceleration) have been adjusted to yield the desired Prandtl, Rayleigh, and Planck numbers.

Table1. Fluid property of free convection

Density (Kg/m3)	Specific heat(j/KgK)	Thermal conductivity(W/mK)	Viscocity μ (Pa-s)	Prandtl No.	Rayleigh no.	Absorption coeff.(σ)
1000 kg/m3	1.103×105	0.6	8.90 ×10−4	0.71	5×105	0.2

B. Governing Equations

The ultimate goal of the field of computational fluid dynamics (CFD) is to understand the physical events that occur in the flow of fluids around and within designated objects. These events are related to the action and interaction of phenomena such as dissipation, diffusion, convection, shock waves, slip surfaces, boundary layers, and turbulence. The governing PDEs are the coupled mass, momentum, and energy conservation equations, applicable for two dimensions. The equations are given as

Continuity
$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \quad \ldots(1)$$

Volume 4 Issue IV, April 2016
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

x- momentum
$$\frac{\partial u}{\partial t}+\frac{\partial (u^2+p)}{\partial x}+\frac{\partial (v)}{\partial y}=\Pr(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}) \qquad ...(2)$$

y-momentum
$$\frac{\partial v}{\partial t}+\frac{\partial (u^2+p)}{\partial x}+\frac{\partial (v)}{\partial y}=\Pr(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2})+Ra\Pr\theta \qquad ...(3)$$

Energy
$$\frac{\partial \theta}{\partial t}+u\frac{\partial \theta}{\partial x}+v\frac{\partial \theta}{\partial y}=\frac{\partial^2 \theta}{\partial x^2}+\frac{\partial^2 \theta}{\partial y^2}) \qquad ...(4)$$

Where, u,v - Velocity components in x and y directions respectively in m/s

θ - Temperature of the fluid in K

Pr - Prandtle number

Ra - Rayleigh number

In convection, density differences generate an additional force (popularly known as the buoyancy force), which competes with the inertial and viscous forces. The ratio of buoyancy to viscous forces is given by the parameter Grashof number (Gr), which controls natural convection. On the other hand, the ratio of momentum to thermal diffusivity, known as Prandtl number (Pr), governs the temperature field and its relationship with the fluid flow characteristics. Rayleigh number (Ra), which is the parameter of interest, is the product of these two dimensionless groups.

C. Modeling Natural Convection In A Closed Domain

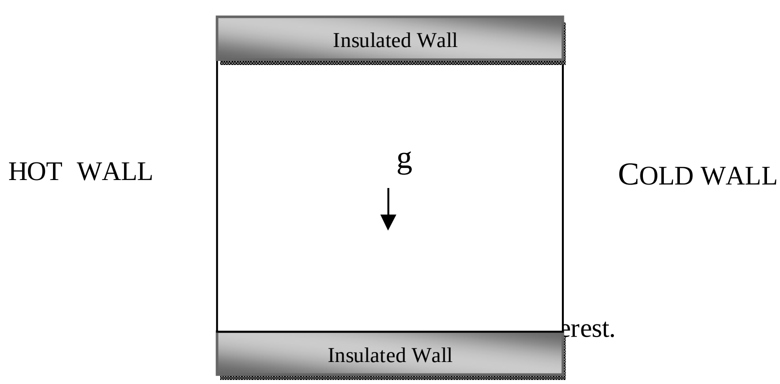
The model natural convection inside a closed domain, the solution will depend on the mass inside the domain. Since this mass will not be known unless the density is known, for this model the flow in one of the following ways:

Perform a transient calculation. In this approach, the initial density will be computed from the initial pressure and temperature, so the initial mass is known. As the solution progresses over time, this mass will be properly conserved. If the temperature differences in your domain are large, you must follow this approach.

Perform a steady-state calculation using the Boussinesq model . In this approach, we will specify a constant density, so the mass is properly specified. This approach is valid only if the temperature differences in the domain are small; if not, we must use the transient approach.

D. Physical Model

The computational physical model was nothing but the top view of the numerical model having physical dimensions given in Fig. 1. All the dimensions in the figure were in meter. The boundaries were chosen in order to maintain the actual model characteristics as far as possible. The computational domain that is shown in Fig. 1 is discretized using an structured grid as shown in Fig. 2.



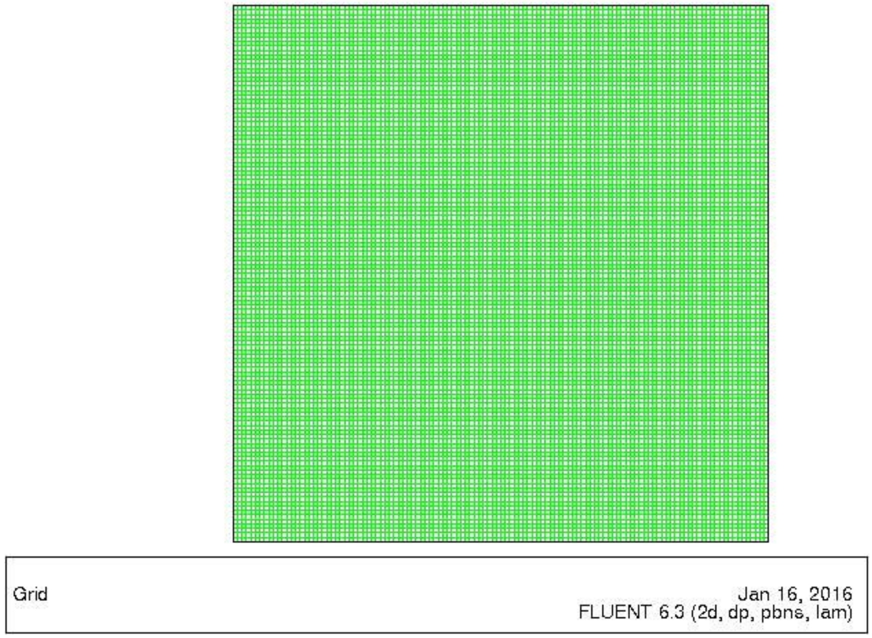


Figure 2. Computational domain of cavity (120×120)

III. CFD SIMULATION

For the CFD analysis of this squre cavity we had considered the two-dimensional simulation technique, which simplifies the solution methodology and consumes less computational time. The CFD tool FLUENT 6.2.29 was utilized to solve the governing equations sequentially using the control volume method. Initially total computational domain was divided into thousands of small cells and each cell acted as an individual control volume. The governing equations were then integrated over each control volume to construct discretized algebraic equations for the dependent variables. These discretized equations were linearized using an implicit method. Iterations are done to achieve a converged solution.

A. Solution Specifications

Appropriate solver, viscous model, material properties, realistic boundary conditions, and solution controls provided for this problem are shown as follows.

Solver: Steady, Viscous Laminar flow.

Material: Water =1000 kg/m3, μ =8.90 × 10−4 kg/m s.

Operating condition: Atmospheric pressure 1.0132 bars.

Boundary conditions:

- Left Wall: hot wall.
- Right Wall: cold wall.
- Top Wall: Adiabatic wall.
- Bottom Wall: Adiabatic wall.

Solution controls:

- Pressure-velocity coupling: Implicit Method for Pressure-Linked Equations SIMPLE.
- Under relaxation factor: 0.7 (momentum).
- Discretization: Momentum (Second-order upwind).
- Initialization: Left wall.
- Residual monitors: 0.001 (continuity, x velocity, y velocity, energy)

B. Convergence Study

In these Residual plots, the residuals get decreased slowly and it is noted that the results converge in the orde of 10-4. Fig. 3 and Fig.

Volume 4 Issue IV, April 2016
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

4 show the residuals of convergence with radiation and without radiation model respectively.

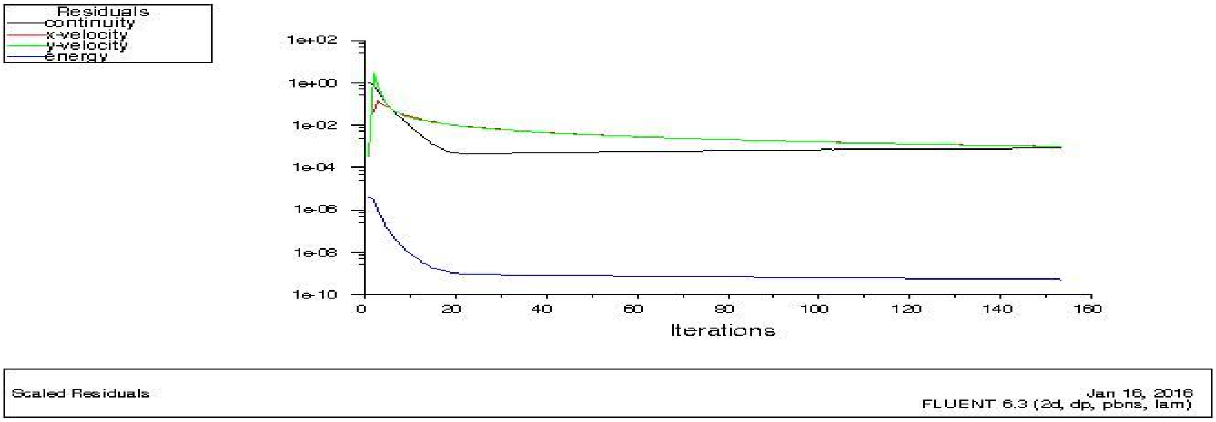


Figure 3. Residuals of convergenge with Rosseland radiation model

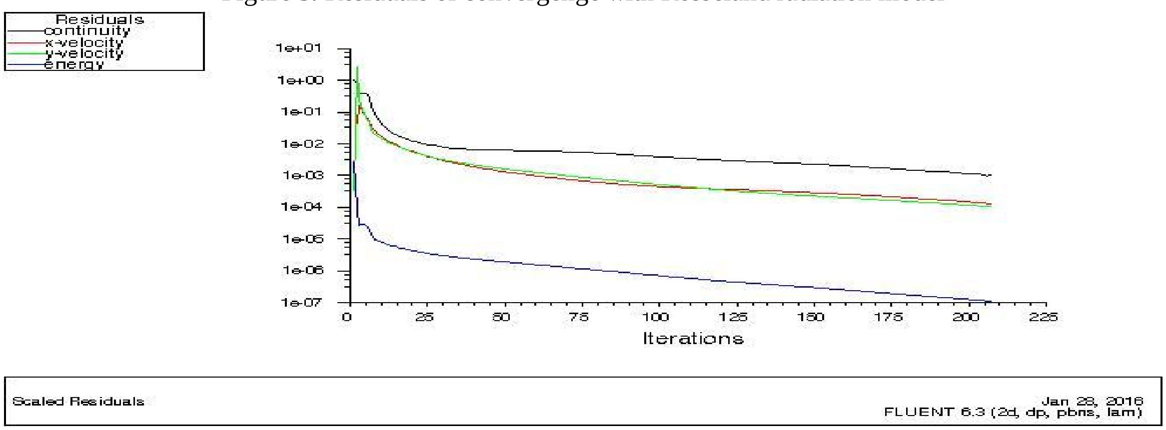
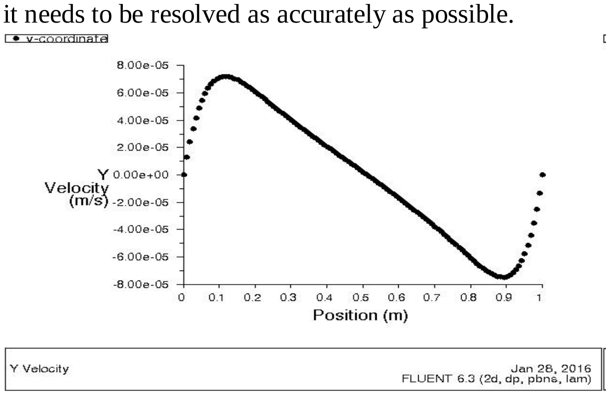


Figure 4. Residuals of convergence without radiation

IV. RESULTS

A. Velocity Distribution

Initially, the fluid inside the cavity is maintained at the same temperature as that of the cold wall. The fluid picks up heat from the hot wall and loses it to the cold wall. There is no transfer of heat through the horizontal walls (either inside or outside). A wide variety of fluid flow and heat transfer features evolve as a function of the Rayleigh number and their precise simulation is indeed a real challenge to any numerical scheme. To start with, comparison of vertical velocity distribution at the mid-height y= 0.5. as a function of the abscissa is presented in Figure 5, over the Ra range of 105 to 106, where a grid size of 120×120 was employed. In fact, the vertical velocity distribution has a direct relation to the size of the boundary layer formed on the hot and cold walls. Therefore, it needs to be resolved as accurately as possible.



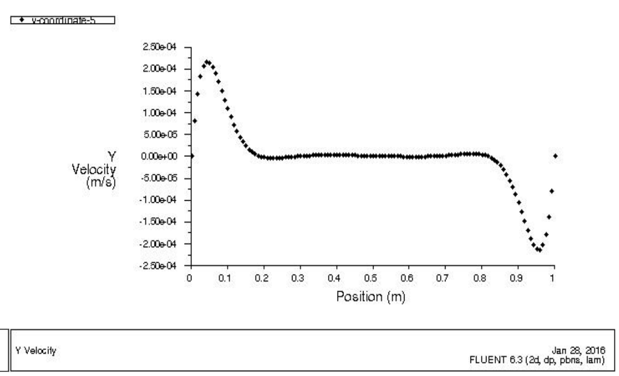


Figure 5. Velocity plot with Rosseland Model

Figure 6. Velocity plot Without radiation

Volume 4 Issue IV, April 2016
ISSN: 2321-9653

B. Natural-Convection Patterns

This subsection analyzes the natural-convection patterns set up by the buoyancy force. The extent of this force is indicated by the Rayleigh number, which in turn dictates both the fluid flow and heat transfer characteristics in the domain. Streamlines, velocity contours, and Temperature are plotted in Figure 6, to understand and analyze these wide variety of features. The square-cavity problem with differentially heated vertical walls has two distinct flow patterns: (1) growth of the boundary layer along the wall; (2) a recirculating motion in the core region. The latter is prominent in the lower-Ra range, while the former dominates the higher-Ra range. These features are very well depicted in the streamlines of Figure 6.

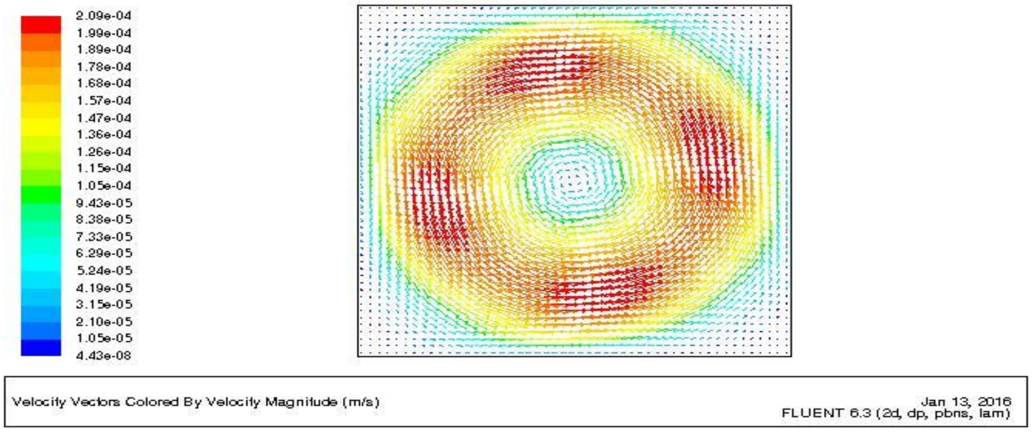


Figure 9.Velocity Vectors for the Rosseland Model

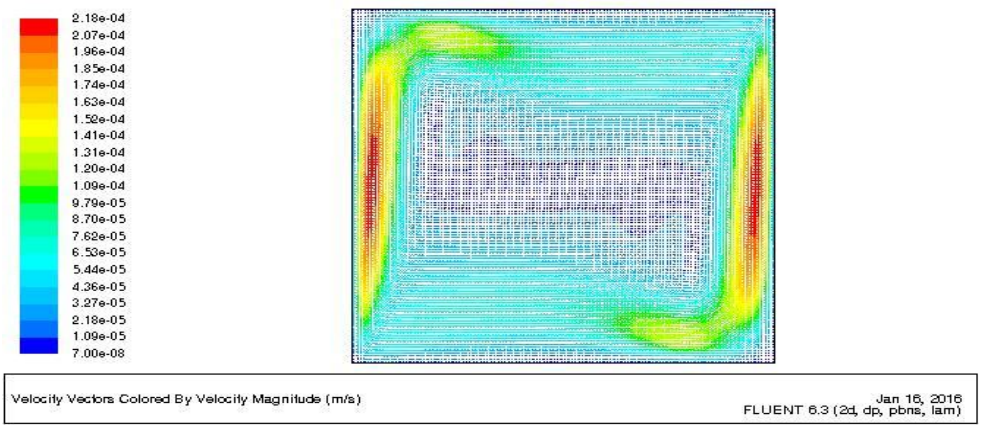


Figure 10 Velocity Vectors contours with no radiation

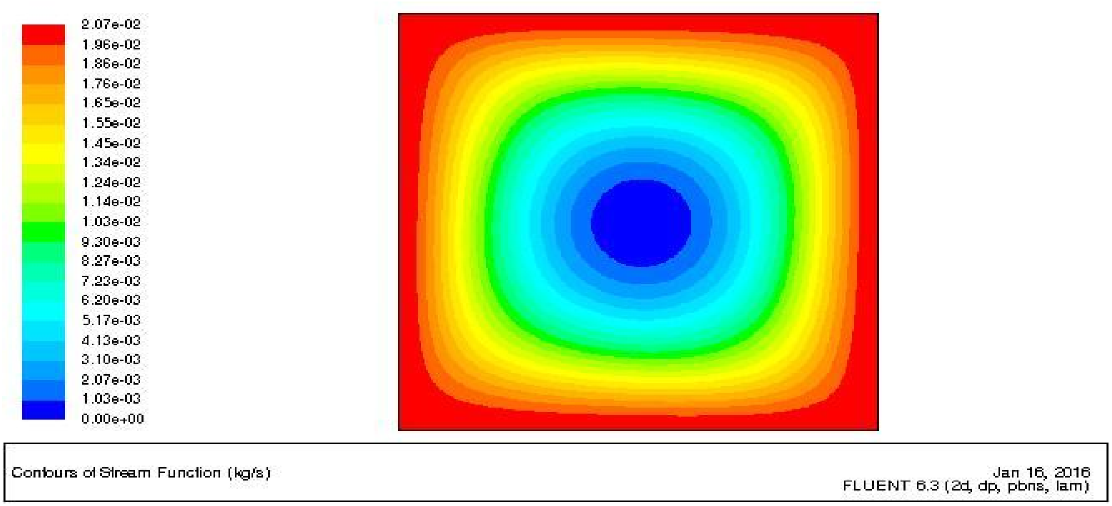


Figure 11 Contours of Stream Function for the Rosseland Model

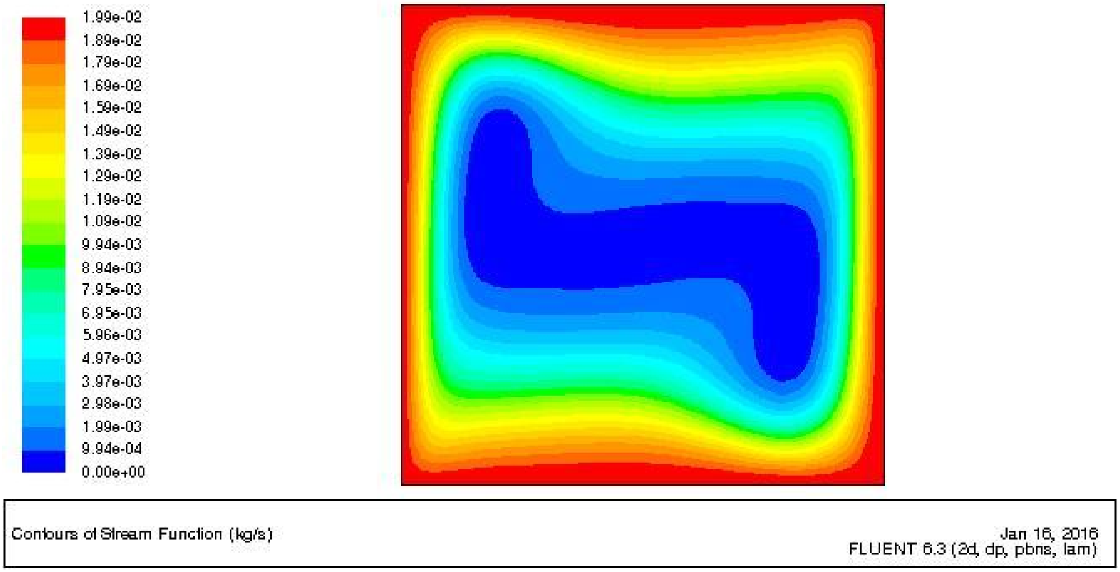


Figure 12 Contours of Stream Function with No Radiation (i.e. only Fee convection)

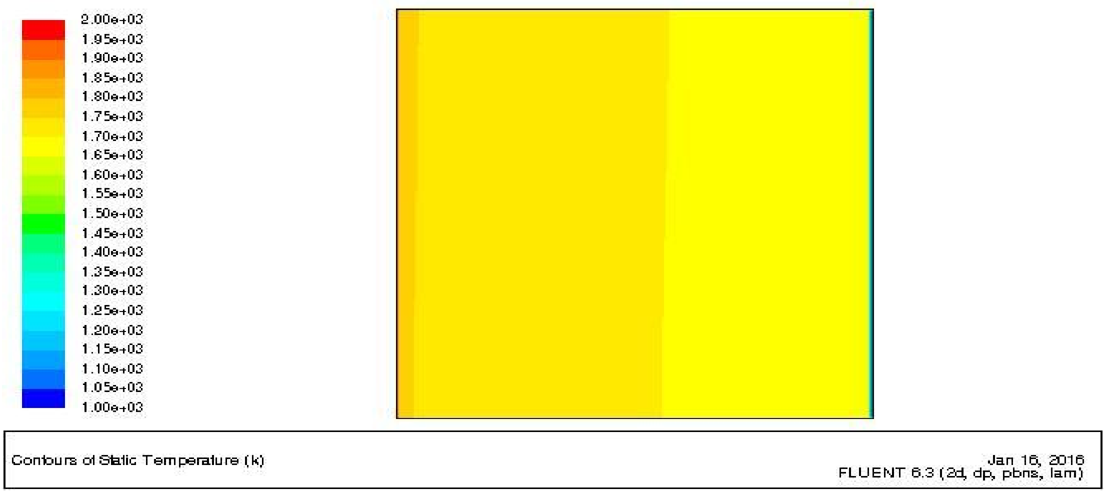


Figure 15 Contours of Temperature for the Rosseland Radiation Model

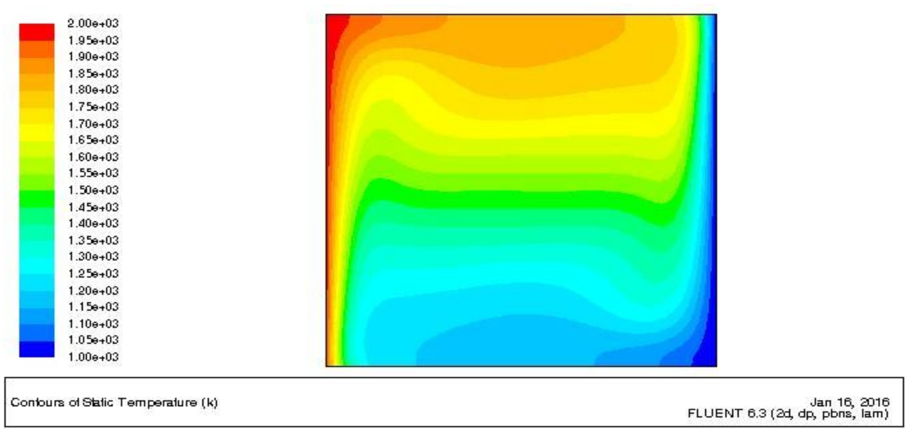


Figure 16 Contours of Temperature without Rosseland Model(i.e. only Fee convection)

V. CONCLUSION

The recirculatory patterns observed are due to the natural convection in the cavity. The radiation should not have a large influence on the flow. The flow pattern is expected to be similar to that obtained with no radiation Figure 14. However, the Rosseland Radiation model predicts a flow pattern that is very symmetric Figure 11 and 14, and quite different from the pure natural convection case.

If computing the results without radiation, set the under-relaxation factor for energy to 0.8 in the Solution Controls panel, and iterate the solution until convergence. (Reset the under-relaxation factor to 1 (the default value) before continuing in Fluent packeg). Compare the stream function contours without radiation (Figure 14) to the plot with the Rosseland radiation model enabled (Figure 11).

The Rosseland model predicts a temperature field (Figure 15) very different from that obtained without radiation (Figure 16).The temperature field predicted by the Rosseland model is not physical.

The velocity, Tempereture and pressure profile reflects the rising plume at the hot right wall, and the falling plume at the cold left wall as shown in figure 5, Fig. 7 ang Fig.8 Respectivly. Compared to the case with no radiation, the profile predicted by the Rosseland model exhibits thicker wall layers. As discussed before, the expected profile for the case with no radiation shown in Fig. 6.

The total wall heat transfer rate is reported for the hot and cold walls in Rosseland model as approximately 7.42 kW and -7.52 kW respectivly. The net heat flux on the lateral walls is a negligible imbalance.

REFERENCES

[1] Eylem Kaya and Sari Widyanti, Numerical model of the deep hydrothermal convection in the taupo volcanic zone, new zealand, Proceedings 37th New Zealand Geothermal Workshop, 18 – 20, 2015

[2] S. Kimura, The Boundary Layer NaturalConvection Regime in a Rectangular Cavity With Uniform Heat Flux From the Side, Journal of Heat Transfer, 2014

[3] A. Bejan, Mass and heat transfer by natural convection in a vertical cavity, duke university durhem, 1984

[4] Frank P. Incropera, David, Fundamentals of Heat and Mass Transfer, JOHN WILEY & SONS, 2011

[5] Chis Long , Naser sayma, Heat transfer, APS Publication, 2003

[6] Tim Persoons, Natural convection heat transfer and fluid dynamics for a pair of vertically aligned isothermal horizontal cylinders, International Journal of Heat and Mass Transfer 54 5163–5172, 2011

[7] A. Liaqat, Conjugate natural convection in a square enclosure containing volumetric sources, IJHMT 44, 3273-3280, 2001

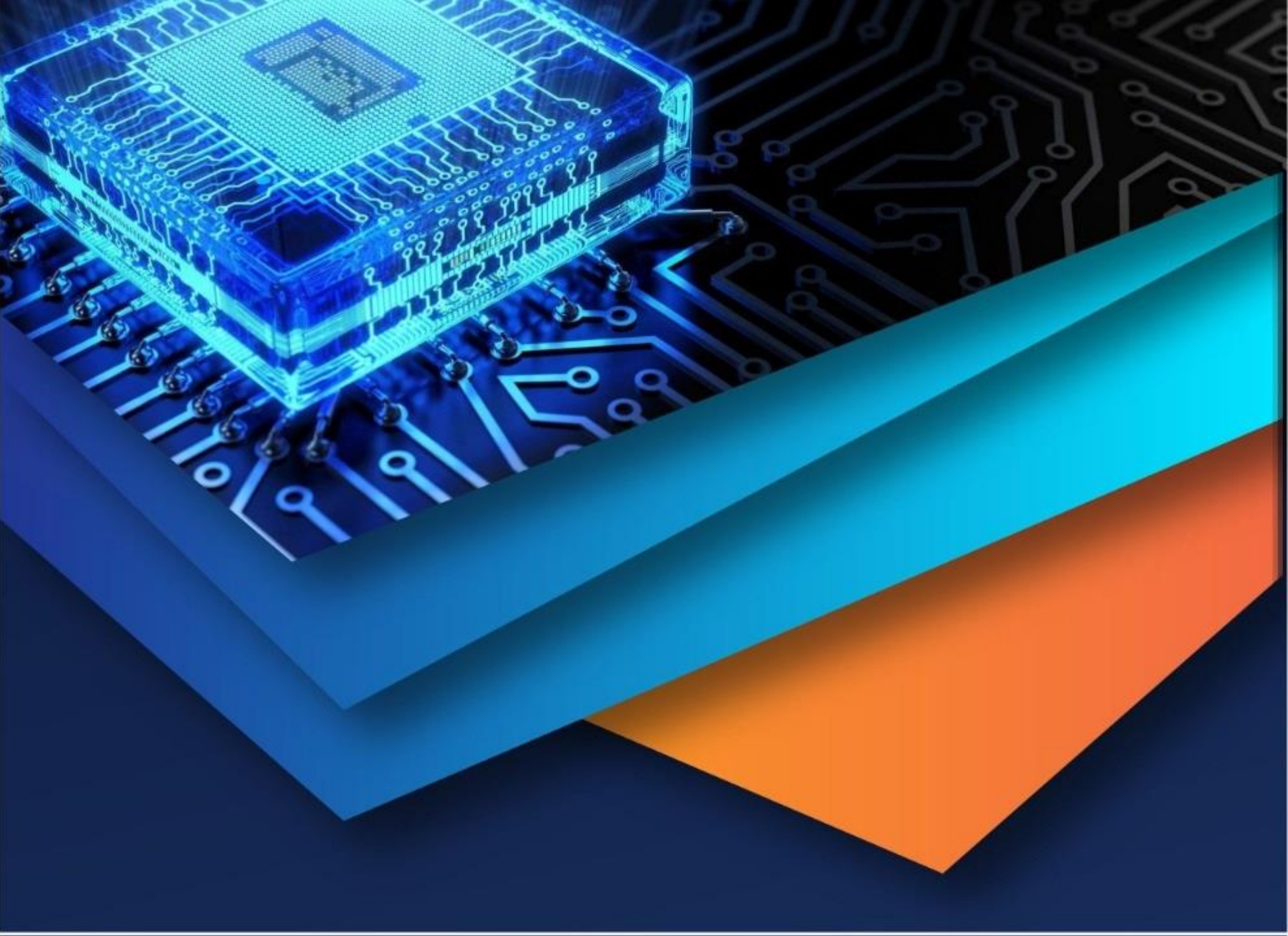
[8] T. D. UPTON, Experimental study of transient natural convection in an inclined rectangularenclosure, Inf. J. Heat Mass Transfer. Vol. 40, No. 11, pp. 2679-2690, 1997

[9] W. Schopf, Visualization of natural convection in a sideheated cavity: transition to the final steady state, Inf. 1. Hear Mars Trmrfer. Vol. 39, No. 16, pp. 3497-3509, 1996

[10] Ho Sang Kwak, Kunio Kuwahara, Jae Min Hyun, Prediction of the resonance frequency of natural convection in an enclosure with time-periodic heating imposed on one sidewall, Int ernational Journal of Heat and Mass Tran sfer 41 , 3157-3160, 1998

[11] H.T. Cheong, Z. Siri, S. Sivasankaran, Effect of aspect ratio on natural convection in an inclined rectangular enclosure with sinusoidal boundary condition, International Communications in Heat and Mass Transfer 45, 75–85, 2013

[12] D. C. Wan, B. S. V. Patnaik, and G. W. Wei, A new benchmark quality solution for the buoyancy-driven cavity by discrete singular convolution, Numerical Heat Transfer, Part B, 40: 199± 228, 2001



10.22214/IJRASET

INDEX COPERNICUS
45.98

IMPACT FACTOR:
7.129

TOGETHER WE REACH THE GOAL
IMPACT FACTOR:
7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY