Journal of Mathematical Analysis and Modeling
jmam.sabapub.com
ISSN 2709-5924

J Math Anal & Model
(2024)5(3) : 24-35
doi:10.48185/jmam.v5i3.1084

Numerical Investigation of Turbulent Convection Flow in a Rectangular Closed Cavity

GEOFREY MOTURI MANOTI [a*], KENNEDY AWUOR [b]

[a] Department of Mathematics and Actuarial Sciences, Kenyatta University, Nairobi Kenya
[b] Department of Mathematics and Actuarial Sciences, Kenyatta University, Nairobi Kenya

• *Received: 14 March 2024* • *Accepted: 12 July 2024* • *Published Online: 03 November 2024*

Abstract

Natural turbulent convection in closed cavities has many practical applications in the field of engineering such as the design of electronic computer chips, atomic installation and industrial cooling among others. In particular, it enables in achieving a desired micro-climate and efficient ventilation in a building. Recent studies show that turbulent flow is affected by variations in Rayleigh numbers, aspect ratio, and heater position among others. Temperature is kept constant in all these studies hence inadequate literature on the effects of temperature on a turbulent flow. In this study, aspect ratio and Rayleigh numbers are kept constant at 2 and 10^{12} respectively and natural turbulent convection flow in a closed rectangular cavity is investigated numerically as the operating temperature is varied from 285.5K to 293K. The rectangular cavity's lower wall was heated and cooling done at the top face wall while the rest of the vertical walls were kept in adiabatic condition. Material properties such as density of the fluid kept on changing at any given temperature. The thermal profile data generated influenced the nature of the turbulent flow. The non-linear averaged continuity, momentum, and energy equation terms were modeled by the SST $k-\omega$ model to generate streamlines, isotherms, and velocity magnitude for a different operating temperature and presented graphically. The finite difference method and FLUENT were used to solve two SST $k-\omega$ model equations, vortices, and energy with boundary conditions. It was discovered that, as the operating temperature increased turbulence decreased due to a decrease in the velocity of the elements and vortices became more parallel and smaller.

Keywords: Mathematical formulation, Aspect ratio, operating temperature, FLUENT software.
2010 MSC: Mathematics Subject Classification 2010 (MSC) .

1. Introduction

Turbulent flows are irregular and unsteady fluid motions in which the particles' energy or momentum changes in time and space at any given point. Reynold [1] defined a turbulent flow in terms of a dimensionless quantity known as the Reynold number (Re). He described a turbulent flow as a flow with Re$>$ 4000. A turbulent fluid is unsteady, rotational, its velocity and direction change with time irregularly, three-dimension diffusive and dissipative in terms of energy.

*Corresponding author: moturigeofrey@gmail.com

The knowledge gained in the study of turbulent convection in an enclosure is applied in the atomic installation and engineering field in the manufacture of electronic chips and the design of efficient room ventilation. The data is also very instrumental in industries to determine the most suitable velocity and temperature field for manufacturing, packaging, storage and preservation processes.

Solving a natural turbulent flow by direct substitution of the changing variables into the Navier Stokes equation by a method called direct numerical solutions (DNS) is nearly impossible because of the small fluctuations experienced in the flow. Mathematicians came up with the Reynold Average Navier Stokes equation (RANS) model that is used since it utilizes the average values of variables and is applicable in both dynamic and steady flows. The RANS equation models comprise the Spalart-Allmaras, $k-\varepsilon$, $k-\omega$, and SST $k-\omega$ among others. The SST $k-\omega$ model is a hybrid of the $k-\varepsilon$ and $k-\omega$. It uses the $k-\varepsilon$ model in the free stream and transitions to the $k-\omega$ model near the walls. In this study, the SST $k-\omega$ model was used. Awuor [2], in his study, ascertained that the two-equation SST $k-\omega$ model is the best to close the motion equation since it takes care of the entire flow field.

Awuor [2] investigated the performance of various turbulence models in a rectangular air-filled enclosure. The system was modeled such that it included the Rayleigh number of about 10^9 (Ra$10^{9)}$ and the turbulence effect. The non-linear momentum and average energy terms equations were modeled using turbulent models such as, and SST. The two opposite vertical walls to each other are maintained at 323K and 283K temperatures respectively and the rest of the walls are in a diabatic state. The pressure term was eliminated by the introduction of the vorticity-vector potential. The remaining terms of energy, vorticity-vector potential and boundary term of the two equations of the model used are solved and the findings are compared with experimental results. He noted that the SST models' Nusselt number, temperature, the kinetic energy of the turbulent flow and velocity value agreed with the experimental values as compared to the other two models. When using SST $k-\omega$ to solve the issue, he found out that the cavity was stratified into a hot region between the window and heater, at the lower part the region was warm and the upper part was cold.

Husain and Arqam [3] conducted a heat transfer by numerical analysis and experimental study on vertical annulus fluid flow. Among the things discussed are parameters of significant interest in its design such as the viz, the effect of geometric and operating parameters, porous media, cylinder rotation, magnetic effects, discrete heating, boiling flow, and use of nano-fluid and conjugate analysis. Their study equally highlighted weak points such as 3D and experimental studies that require further research.

Roshan *et al.* [4] Large-eddy simulation (LES) of natural convection turbulent in a stretched spiral is considered and Rayleigh Benard convection (RBC) for Prandtl numbers 0.768 and 1 at extremely high Rayleigh numbers ranging from 10^6 to 10^{15} is simulated. The study pointed out the LES can push thermal convection simulation to extremely high Rayleigh numbers and this helps in testing for a transition to ultimate convection.

Polasanapalli and Anuipindi [5] Off-Lattice Boltzmann was used in the study of mixed convection heat transfer in the 2-D annular cavity. The numerical was verified by mixed convection in the annular cavity and Taylor-Couette flow. The effect of direction and strength of rotation of the cylinders was equally analyzed. Rayleigh numbers between 10^4

to 10^6 and Reynold numbers $Re = 0$ to 10^4 were considered at Prandtl number $Pr = 0.71$. They find out that; the heat flow rate is low despite the change in Reynold numbers in forced convection. At $Ra = 10^4$ and below mixed convection is lower as compared to natural convection. At $Ra = 10^5$ and $Ra = 10^6$ certain Reynold numbers have a higher heat transfer rate but beyond that Reynold number, the rate of heat transfer decreases with the rise in Reynold number. Richardson function (Nusselt number) is also studied in all the configurations used and a map divided into three zones is obtained.

Weppe et *al.;* [6] carried out an experimental investigation of a natural turbulent in a cubical cavity with an inner obstacle that is partially heated. A uniform temperature is used to heat one of the vertical walls while the rest of the walls are kept in a diabatic condition. Temperature and velocity are measured using a micro-thermocouple and Particle Image Velocimetry. In both vertical channels, boundary layer flows were observed. The thermal stratification linearity found on the lower part is broken by the recirculating zone found at the upper heated channel part. The internal gravity in the thermally stratified heated channel is discussed, waves instability boundary layer and subsequent oscillating buoyant jets were equally analyzed.

Belharizi *et al.* [7] did comparative research between the RANS and LES on natural convection inside a square cavity. Natural turbulent convection confined in a 3D square cavity was heated differentially at its two side walls with ($Ra = 1.58\text{x}10^9$). Turbulent convection was numerically simulated using EDF code. The low Reynold number models, shear stress transport (SST $k-\omega$), $\varphi - f$ model, and the large eddy simulation (LES) technique were used and their results were compared with experimental results. Although the numerical results agreed with the experimental findings, the SST k-w model gave a good prediction for different temperature profiles. In contrast, the $\varphi - f$ model was more accurate for velocity profiles. This was because of the good resolution of turbulence properties near-wall region and the ability of the models to mimic the physical flow in this geometry.

Liu *et al.* [8]. Conducted research using five RANS (Standard $k-w$, three Eddy Viscosity Models, SST K-w and low Reynold number $k-\varepsilon$ and two Reynold Stress Models) to assess their ability to predict heat transfer across a range of Pandtl (Pr) and Reynolds numbers. The Rans results were compared with the corresponding Direct Numerical Simulation data. The effect of Reynold number on turbulent heat transfer is studied for a channel at Pr of 0.71 for friction Reynold number values of 180,395,640 and 1020. The results were accurately predicted using all models but as the Reynold number increased for all models the mean temperature dropped except for the SST $k-\omega$ model. The effect of Pr on turbulent heat transfer for values ranging from 0.025 and 10 was equally analyzed. An error analysis was conducted on results obtained from different models. SST $k-\omega$ had the smallest error margin for the prediction of Nusselt and mean temperature for different Reynold numbers and high Prandtl numbers. For a low Reynold number $k{-}\epsilon$ LS models had the smallest errors for low-Prandtl-numbers for different Reynold numbers. To identify Pr effects on forced convection an analytical solution is utilized in the transitional region, analytical region, and turbulent diffusion-dominated region. The findings provide insights into the performance of different RANS models for heat transfer predictions in a flow.

De Medeiros *et al.* [9] carried out research on the effect of heat conducted and adiabatic inserts for ray numbers Ra 10^7 to Ra 10^9. The study showed that due to recirculation restriction, heat transfer is detrimental irrespective of the angle of inclination of the cavity. Research on Nusselt and Reynold numbers for cavity insert types and sizes and inclination angle was equally conducted.

Gnanasekaran and Satheesh [10] Numerically investigated two-dimensional steady incompressible turbulent flow characteristics in a closed cavity. The governing equations are discretized using the finite volume method (FVM). To predict the flow characteristics. The speed ratio is varied$0.05 \leqslant S \leqslant 1.0$, aspect ratio $0.5 \leqslant K \leqslant 2.0$ and Rynold number$(1 \times 10^4 \leqslant 2 \times 10^5)$. The flow characteristics are analyzed using turbulent quantities, Reynold stress $u'v'$ and stream functionϕ. For a system of selected range of operating parameters, speed ratio, and Reynold numbers significantly affect vortices. Turbulent kinetic energy is reduced by increasing the aspect ratio and Reynolds number. For $K = 0.5$ and $S = 0.05$, the dissipation rate and kinetic energy decreased by 42.28% and 89.16% respectively. When Reynolds number is increased from 1×10^4 to 2×10^5, the turbulent viscosity increases by 92.10%. An increase in flow parameters causes a decrease in turbulent quantities.

Ali and Sharma [11] investigated laminar turbulent natural convection combined with radiation in a square enclosure with a vertical partition having different heights. The partition distance varies from the hot wall, the left wall is heated and the right wall is cold and the rest of the walls are considered adiabatic. Analysis was done for both turbulent and laminar flow for Rayleigh numbers ranging from 10^3 to 10^{10}. The partition thickness is fixed at $\frac{L}{20}$ and air is used as a medium. From the result, the increase in the convective Nusselt number in turbulent flow is lower than the radioactive Nusselt number. As the Rayleigh number increases from 10^8 to 10^{10} the Nusselt number increases by 352% and 379%. However, an increase in partition height decreases the value of Nusselt to 15% and 28% for Ra 10^5 and 10^{10} respectively.

2. Governing equations

In two dimensions the equations representing a rectangular direction are given as follows;

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{2.1}$$

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = F_x - \frac{\partial P}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{2.2}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = F_y - \frac{\partial P}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{2.3}$$

$$C_p\left(\frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}\right) = k\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) + \phi \tag{2.4}$$

but $\phi = \mu\, 2\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial x}\right)^2\right] + \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)^2$

3. Dimensionless energy, momentum and continuity equations

Non-dimensionalization of flow equations such as continuity, momentum and energy make the equations simpler since only important terms are highlighted. The equations are given as;

$$\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} = 0 \tag{3.1}$$

$$\frac{\partial U}{\partial \tau} + U\frac{\partial U}{\partial X} + V\frac{\partial U}{\partial Y} = -\frac{\partial P}{\partial X} + P_r\left(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2}\right) \tag{3.2}$$

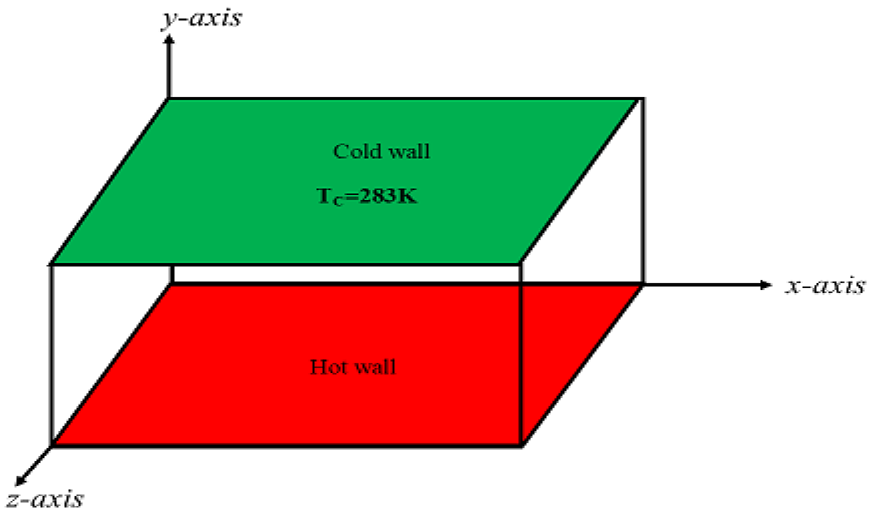
$$\frac{\partial V}{\partial \tau} + U\frac{\partial V}{\partial X} + V\frac{\partial V}{\partial Y} = -\frac{\partial P}{\partial Y} + P_r\left(\frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2}\right) + R_a \cdot P_r.\theta_f \tag{3.3}$$

$$\left(\frac{\partial \theta_f}{\partial \tau} + U\frac{\partial \theta_f}{\partial X} + V\frac{\partial \theta_f}{\partial Y}\right) = k\left(\frac{\partial^2 \theta_f}{\partial X^2} + \frac{\partial^2 \theta_f}{\partial Y^{\,2}}\right) + \varphi \tag{3.4}$$

R_a and $P_r$denote Rayleigh and Prandtl numbers respectively while $\theta_f$represents non- dimension fluid temperature. Where $R_a = G_{r,P_r} = \frac{g\beta(T_S - T\infty)L^3}{\nu\propto}$ and $P_r = \frac{V}{\alpha^2}$

4. Mathematical formulation

If a rectangle in 2-dimension whose top face temperature is maintained at a constant temperature T_c, the corresponding lower face temperature at T_h such that $T_h > T_c$ and the rest of the walls are adiabatic is considered. The T_h is increased at intervals varying the operating temperature from 285.5K to 293K. Due to the temperature difference between the hot and cold walls, natural convection inside the rectangle is generated and studied as the operating temperatures are varied. The aspect ratio of the geometry is chosen as 2 while the Rayleigh number at 10^{12} and maintained throughout the study.



5. Methods of solutions

The governing equations are solved using fluent 6.3.26 software together with bousinesq approximation.

6. Results and discussions

The results presented in this chapter for natural turbulent convection in a 2-D enclosure were obtained by solving governing equations by SIMPLEC for operating temperatures 285.5K, 290.5K and 293K. The boundary conditions were used and the numerical solutions of SST $k-\omega$ model variables were obtained. The experimental data can be used to validate the numerical results obtained.

i) **Isotherms**

Refers to lines of constant or equal temperature. In a graph, the lines or curves connect constant or equal temperature. From the results presented 261K, 253K and 241K are the maximum temperatures for figure 1,2 and 3. The isothermal operating temperatures for Figures 1,2 and 3 are 285.5K,290.5K and 293K respectively. High temperatures are experienced at the centre due to mixing of fluid particles. At extremes (left and right) the temperatures experienced are relatively low as compared to the center. Convectional currents flow inside the cavity increases with increase in operating temperature. In conclusion, as the operating temperature increases temperature inside the cavity decreases.

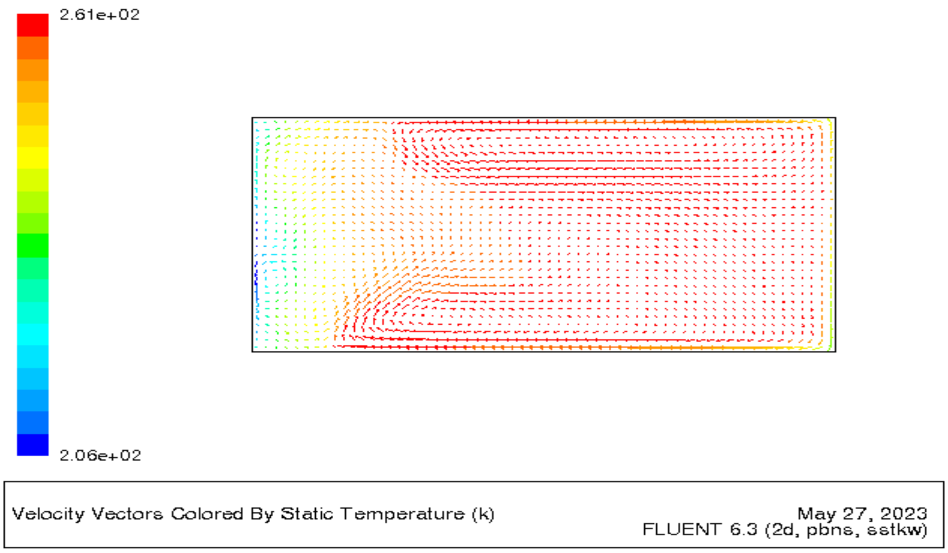


Figure 1 ***Isotherms of Operating Temperature 285.5K***

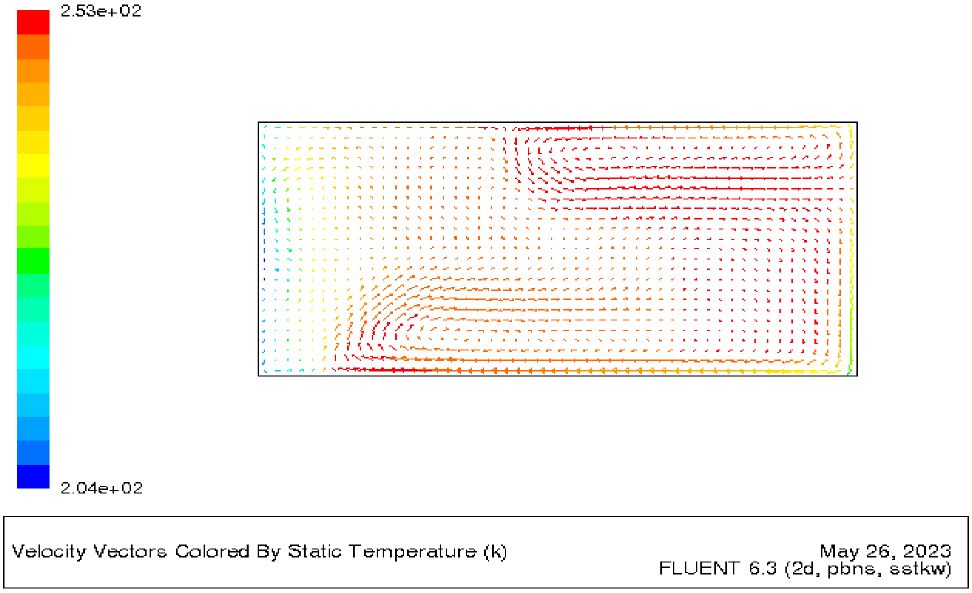


Figure 2 ***Isotherms of Operating Temperature 290.5K***

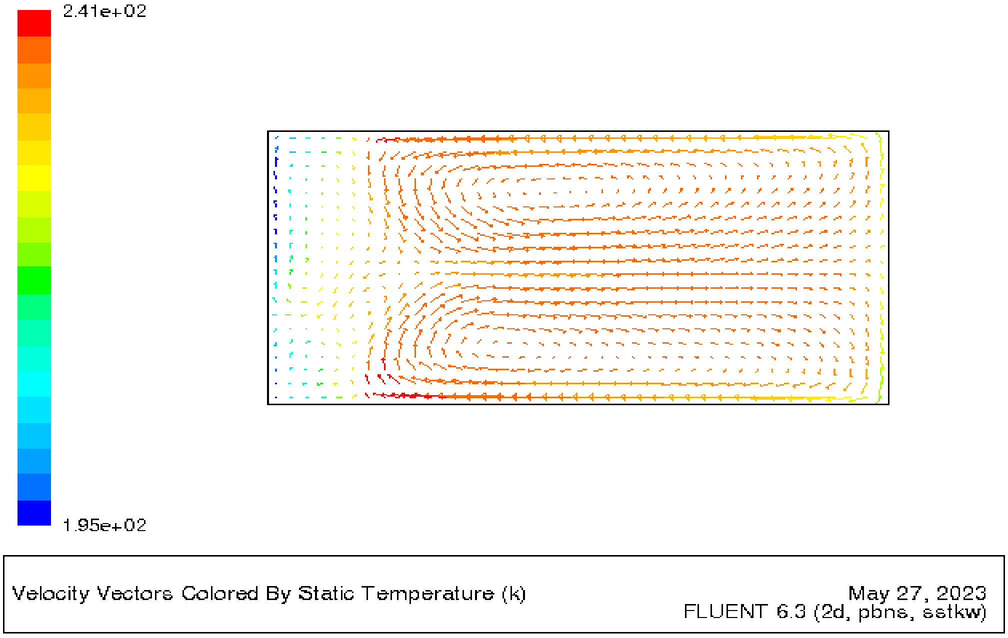


Figure 3 ***Isotherms of Operating Temperature 293K***

ii) **Contours of Velocity Magnitude**

In Fig 4, the highest velocity of air particles is 0.784m/s, in Fig 5, the highest velocity is 0.778 m/s and in Fig 6 the highest velocity is 0.590m/s. In Fig 4, the mixing region at the center has the highest speed and more vortices that become parallel due to an increase in operating temperature. In this study, fig 6 has the most parallel vortices this implies that an increase in operating temperature decreases turbulence.

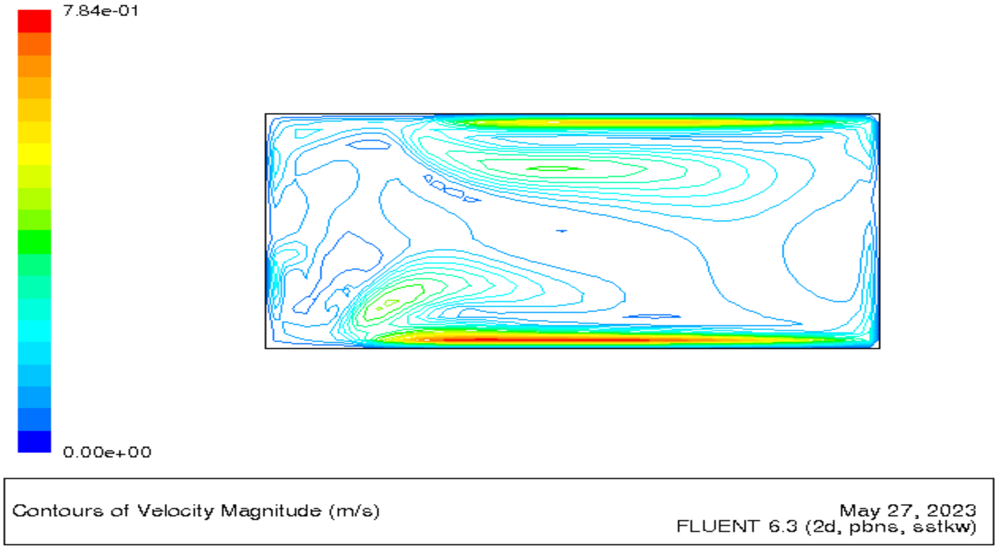


Figure 4 ***Contours of velocity magnitude Operating Temperature 285.5K***

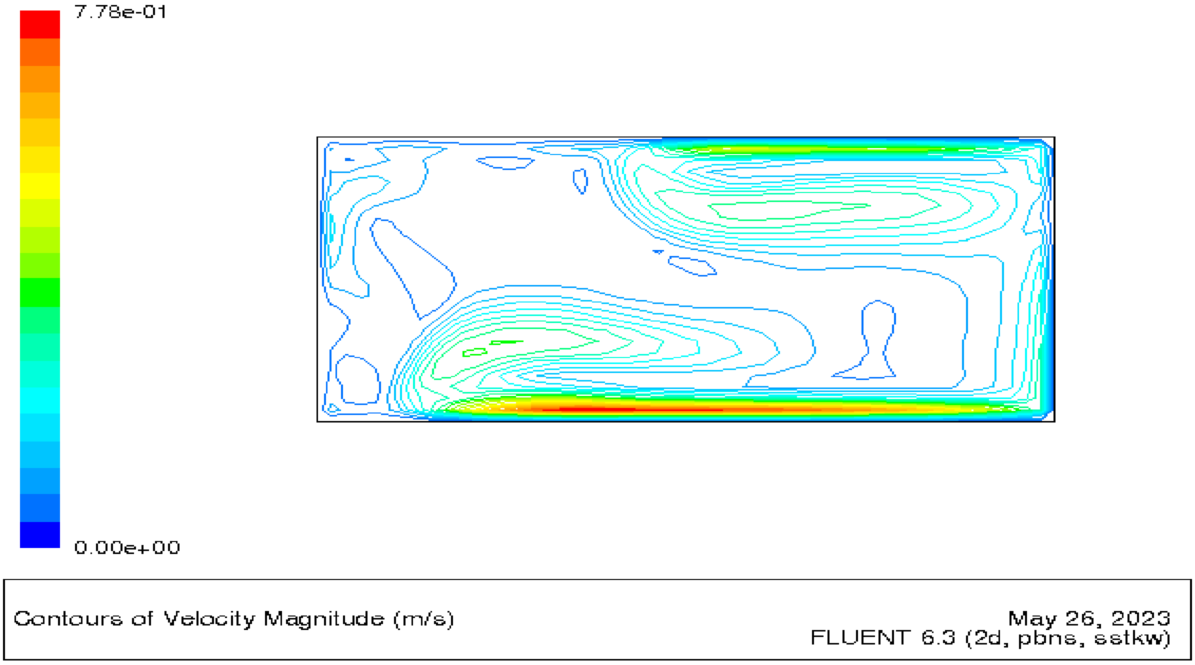


Figure 5 ***Contours of velocity magnitude Operating Temperature 290.5K***

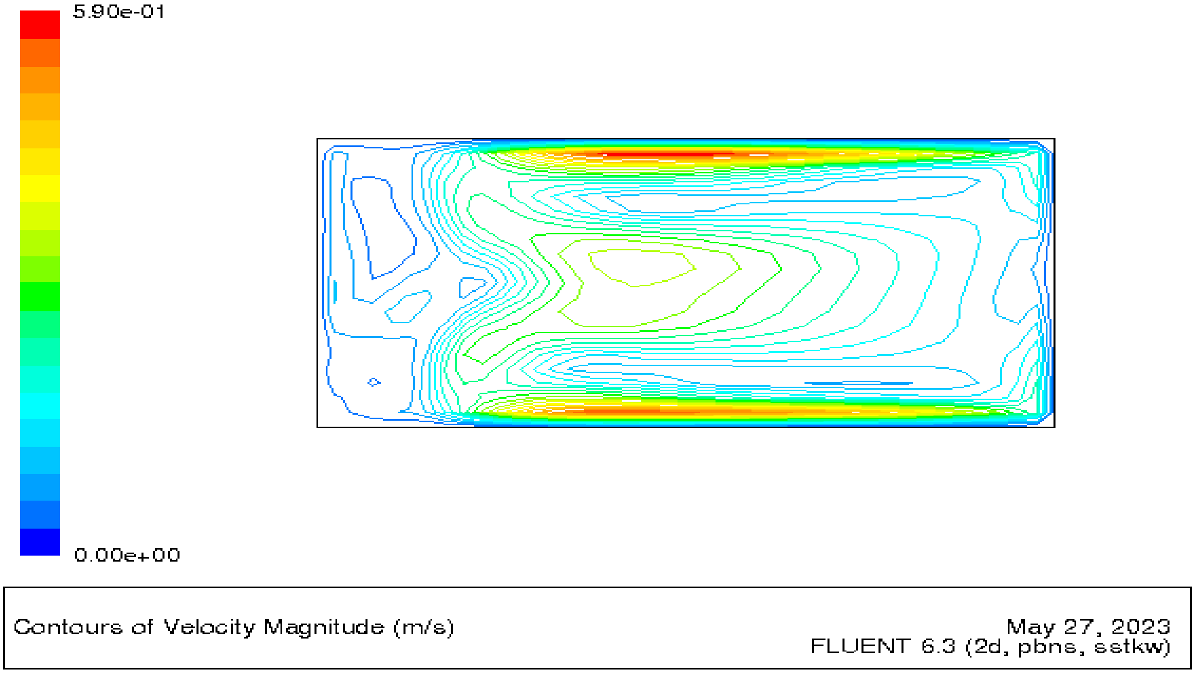


Figure 6 ***Contours of velocity magnitude Operating Temperature 293K***

iii) **Streamline Distribution**

Streamline refers to imaginary line in a fluid, a tangent at any point on the line gives the speed of the element in the fluid. Operating temperature 293K has the lowest value of 0.705kg/s followed by the operating temperature of 290.5K which is 0.707kg/s and 0.822kg/s for a temperature 285.5K. In fig.7, the vortices are big and they assume a circular path that deforms as distance increases from their centers. In fig 7 the radius of the center circle reduces which also decreases as the operating temperature increases as seen in Fig 9.

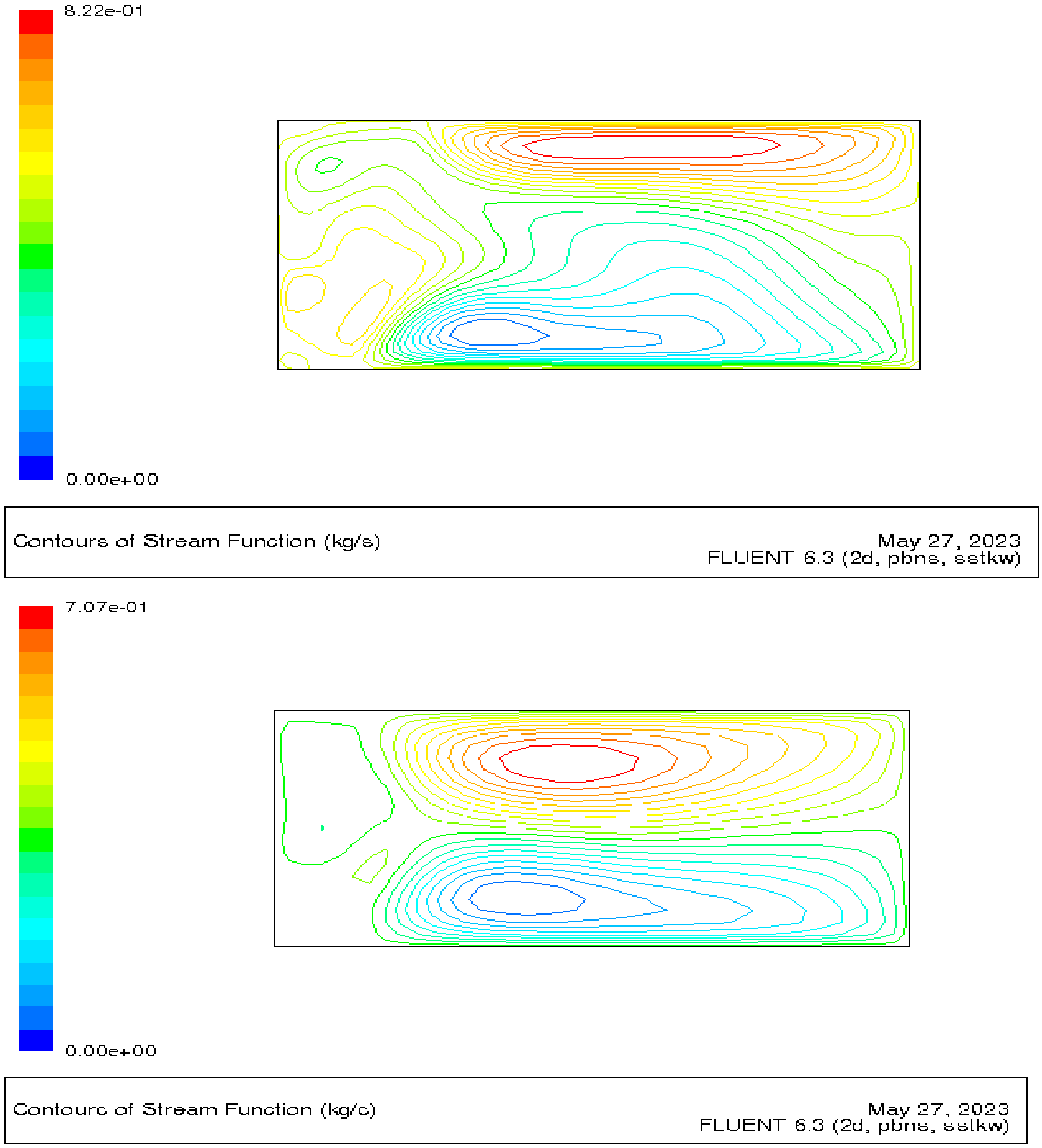


Figure 8 ***Contours of a streamline of Operating Temperature 290.5K***

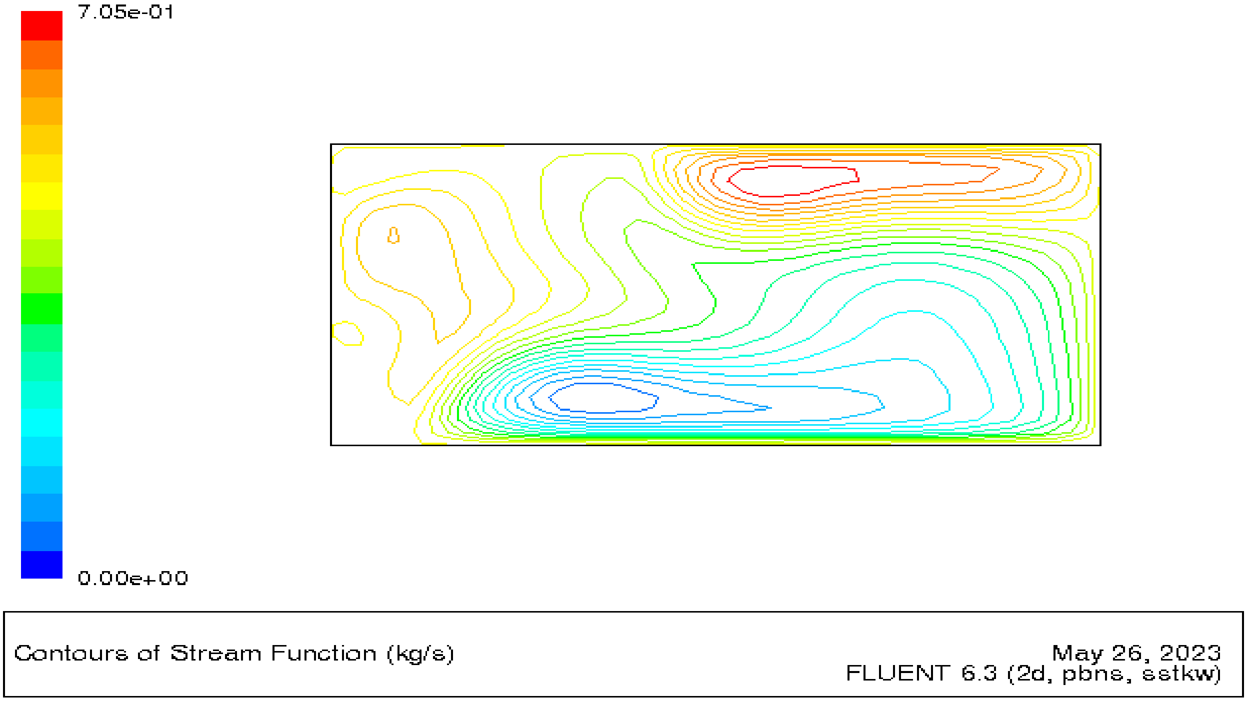


Figure 9 ***Contours of Streamline of Operating Temperature 293K***

7. Conclusion

An air-filled rectangular enclosure of aspect ratio 2 and Raleigh number 10^{12} was considered. The lower face of the cavity was heated while the upper face was maintained at a temperature of 283K. The rest of the vertical walls were kept in diabatic conditions. Natural convection was generated inside the cavity due to the temperature difference between the hot and cold face. The heating temperature is increased further varying the operating temperature inside the cavity.

The resulting natural convections at a given temperature were modeled using the SST $k-\omega$ model by setting the boundary conditions and the Boussinesq estimation was used to simplify the conservation equations. The finite difference and three-point forward approximation with limit conditions were used to discretize the simplified governing equations

Isotherms, velocity magnitude and streamline distribution for the specific operating temperatures was generated using Fluent 6.3.26 and presented graphically. It was noted from the results that, an increase in operating temperature resulted in a decrease in natural turbulence due to a decrease in vortices and speed. At different temperatures, two large circular cells that are in clockwise rotational motion are formed. This therefore shows that temperature distribution in an enclosure is determined by the nature of the turbulence flow within the cavity. The study equips engineers with knowledge for enhancing practical application of turbulence modelling.

References

[1] Reynolds, O., (1883), “An experimental investigation of the circumstances which determine whether the motion of water shall be direct or sinuous and the law of resistance in parallel channels” Phil Trans Roy Soc, 174, 935-982.

[2] Awuor K. (2012), Turbulent Natural Convection in an enclosure; Numerical study of different k-epsilon models, Kenyatta University, Kenya ,1-102.

[3] Husain, S. and Arqam, M (2021). A review on the thermal performance of natural convection in a vertical annulus and its application. Renewable and Sustainable Energy Reviews, Volume 150.

[4] Roshan, S., Ravi, S. and Mahendra, K. (2022) Large-eddy simulation of Rayleigh -Bernard convection at extreme Rayleigh numbers. Physics of fluids 34:7, 075133

[5] Polasanapalli, S. and Anupindi, K. (2022) Mixed convection heat transfer in the two-dimensional annular cavity using an off-lattice Boltzmann method. International Journal of Thermal Sciences 179,107677

[6] Weppe, A. and Moreau, F. (2022) Experimental investigation of a turbulent natural convection cubic cavity with an inner obstacle partially heated; Int. J. Heat Mass Transfer.

[7] Belharizi, M., Khorsi, & Sikula, O. (2023). RANS and LES Computations of Natural Convection in a Square Cavity. Journal of Engineering Physics and Thermophysics, 96(4), 1017-1027.

[8] Liu, L., Ahmed, U., & Chakraborty, N. (2024). A Comprehensive Evaluation of Turbulence Models for Predicting Heat Transfer in Turbulent Channel Flow across Various Prandtl Number Regimes. Fluids, 9(2), 42.

[9] De Medeiros, J. A. C., Alves, M. V. C., Zdanski, P. S. B., & Vaz Jr, M. (2024). Turbulent natural convection inside inclined enclosures: Effects of adiabatic and heat-conducting inserts. Numerical Heat Transfer, Part A: Applications, 1-16.

[10] Gnanasekaran, M., & Satheesh, A. (2024). Numerical analysis of turbulent flow characteristics with the influence of speed ratio in a double-sided cavity. MethodsX, 12, 102594

[11] Ali, M., & Sharma, A. K. (2022, December). Investigation of Laminar and Turbulent Natural Convection Combined with Radiation in a Square Enclosure with a Vertical Partition. In International Conference on Advances in Heat Transfer and Fluid Dynamics (pp. 265-275). Singapore: Springer Nature Singapore.