Enhanced Knowledge in Sciences and Technology Vol. 3 No. 2 (2023) 034-041
© Universiti Tun Hussein Onn Malaysia Publisher's Office

EKST

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/ekst
e-ISSN : 2773-6385

Heatline Approach of Mixed Convection Lid driven Cavity Having Heated Wall via Finite Element Method

Pang Sui Liang[1*]

[1]Department of Mathematics and Statistics, Faculty of Applied Sciences and Technology,
Universiti Tun Hussein Onn Malaysia (Pagoh Campus), 84600 Pagoh, Muar, Johor, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/ekst. 2023.03.02.005
Received 15 January 2023; Accepted 19 February 2023; Available online 30 November 2023

Abstract: In this study, mixed convection in lid driven cavity having heated wall is solved by COMSOL Multiphysics. The objectives of this study are display the isotherms, streamlines and heatlines within the cavity in different values of Reynolds number (*Re*) and Prandtl number (*Pr*), compare the results with earlier research and provide a suitable guideline for COMSOL Multiphysics 5.5. This study is showed the step by step in using COMSOL version 5.5 to create and to modify the streamlines, isotherms and heatlines plot of the heat transfer in lid-driven cavity. The results show that, the streamlines and isotherms flow patterns are almost equal compare with the results of previous study but heatlines may have slightly different. For future work, the study can compare different values or situations in the streamline, isotherm and heatlines plot to enhance understanding of the heatline flows in mixed convection lid driven cavity.

Keywords: Mixed Convection, Square, COMSOL Multiphysics, Streamline, Isotherm, Heatline

1. Introduction

Numerical modeling of convective heat transfer problem using heatline analysis has been an area of great interest in the recent years due to its wide application in engineering. Some of the earlier works were also devoted to a variety of practical engineering problems, such as electronic or computer equipment, thermal energy storage systems, and pollution control, among others [1]. Kimura and Bejan [2] introduced heatlines like streamline that are trajectories of flow of heat energy flow in a heat transfer situation involving convection or conduction and thus useful for visualising total energy flow. Ramakrishna et al. [3] analysed the mixed convection within porous square cavities using Bejan's heatlines. They studied various cases depending on effects of thermal aspect ratio and thermal boundary conditions. The dimensionless parameters governing mixed convection in a cavity filled with fluid-saturated porous media are Darcy number (*Da*), Grashoff number (*Gr*), Reynolds number (*Re*), and Prandtl number (*Pr*). Basak *et al.* [4] studied the conjugate natural convection within enclosure with

*Corresponding author: suiliang2345@gmail.com

different conductivity ratios and wall thicknesses. In temperature-sensitive applications, this heating approach can be employed for high-temperature shielding or to reduce thermal runaway. For example, heat flow is controlled by solid wall resistance in environmental control systems, chemical storage reservoirs, and other applications.

Additionally, Biswal et al. [5] analysed the fluid and heat flow during mixed convection in entrapped triangular cavities. The mixed convection in entrapped triangular cavity has application in heat exchanger in the daily life. Bondarenko *et al.* [6] investigated natural convection cooling of thermally conductive and heat-generating elements located in a square housing filled with alumina-water nanofluid. It has been discovered that adding nanoparticles to the cooling process can speed up the cooling process when the heat source is close to the vertical cold wall. Basak *et al.* [7] use heatline concept studied heat flow patterns under natural convection in a trapezoidal enclosure. Heatlines are continuous lines that run between the cold and heated walls. For conduction dominating heat transmission, the lines are perpendicular to the isothermal wall. Besides that, Ramakrishna et al. [8] used thermal aspect ratio to analyse the effects if thermal boundary condition in a square cavity. From that result, due to the large density of hot lines in this region, a larger thermal gradient is observed in the center of the bottom wall at $A = 0.1$ and decreases as A increases from 0.1 to 0.9, independent of *Re* and *Pr*. In this research, there are many different types of cavities can be simulated, however square cavities were chosen for this investigation. The finite element method, finite volume method, Galerkin weight residual approach, COMSOL, and MATLAB are among the methods used to solve the fluid dynamic issue. In this study, finite element method with the assist of COMSOL are used in this study and both of this method will be explained detail in Methodology.

2. Methodology

2.1 Physical model

Figure 1 is the 2D diagram for the study. The square cavity is modelled by referring to Ramakrishna [5].

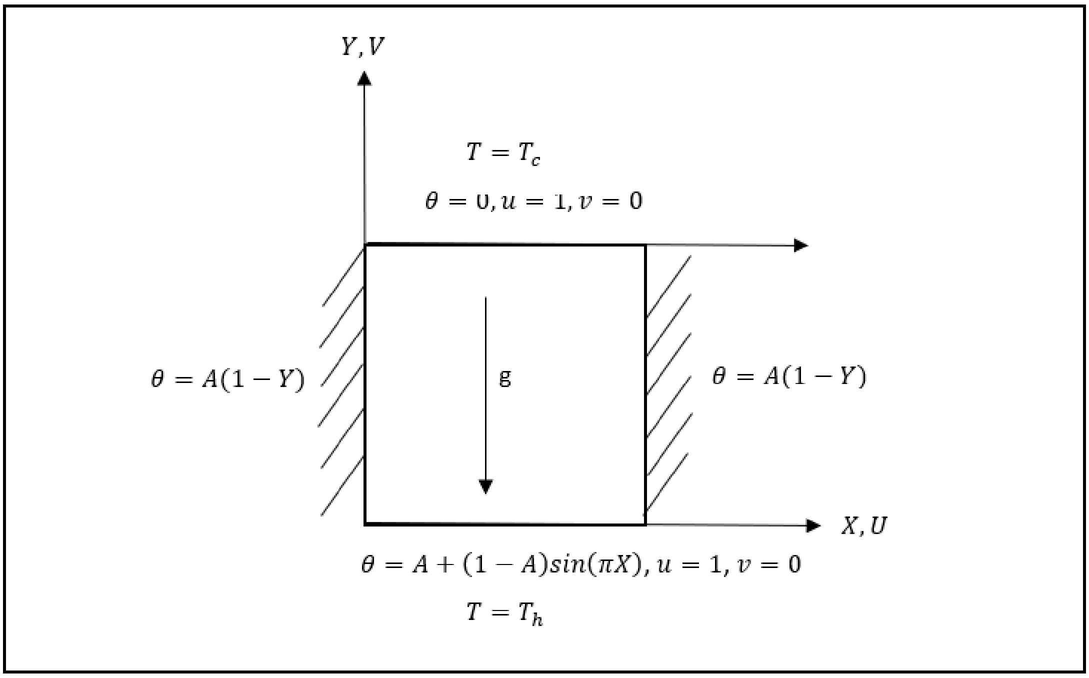


Figure 1: Schematic of the physical model for mixed convection within square cavity [5]

Left and right sides of the walls are set as an insulator wall, and bottom and top walls are kept at a constant temperature. Using the conservation of mass, momentum, and energy in dimensionless form, the governing equations for this study with mixed convection can be written as below:

The continuity equation is shown in Eq. 1 and the momentum equations are shown in Eq. 2 and Eq. 3 while Eq. 4 represent the energy equation.

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0 \qquad \text{Eq. 1}$$

$$U\frac{\partial U}{\partial X} + V\frac{\partial U}{\partial Y} = -\frac{\partial P}{\partial X} + \frac{1}{Re}\left(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2}\right) \qquad \text{Eq. } 2$$

$$U\frac{\partial V}{\partial X} + V\frac{\partial V}{\partial Y} = -\frac{\partial P}{\partial Y} + \frac{1}{Re}\left(\frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2}\right) + \frac{Gr}{Re^2}\theta \qquad \text{Eq. } 3$$

$$U\frac{\partial \theta}{\partial X} + V\frac{\partial \theta}{\partial Y} = \frac{1}{RePr}\left(\frac{\partial^2 \theta}{\partial X^2} + \frac{\partial^2 \theta}{\partial Y^2}\right) \qquad \text{Eq. } 4$$

With following boundary condition:

$$U = 0, V = 0, \theta = A + (1 - A)sin(\pi X), \text{for } Y = 0 \text{ and } 0 \leq X \leq 1$$

$$U = 0, V = 0, \theta = A(1 - Y), \text{for } X = 0 \text{ and } 0 \leq Y \leq 1$$

$$U = 0, V = 0, \theta = A(1 - Y), \text{for } X = 1 \text{ and } 0 \leq Y \leq 1$$

$$U = 1, V = 0, \frac{\partial \theta}{\partial Y} = 0, \text{for } Y = 1 \text{ and } 0 \leq X \leq 1$$

Eq. 5 and *Eq.* 6 represent the equation of streamfunction and heatfunction.

$$\frac{\partial^2 \psi}{\partial X^2} + \frac{\partial^2 \psi}{\partial Y^2} = \frac{\partial U}{\partial Y} - \frac{\partial V}{\partial X} \qquad \text{Eq. 5}$$

$$\frac{\partial^2 H}{\partial X^2} + \frac{\partial^2 H}{\partial Y^2} = \frac{\partial}{\partial Y}(U\theta) - \frac{\partial}{\partial X}(V\theta) \qquad \text{Eq. 6}$$

The equation of heatfunction on left wall, bottom wall, right wall, and top wall are shown in Eq. 7, Eq. 8, Eq. 9, and Eq. 10.

$$n \cdot \nabla H = \frac{A}{RePr} \qquad \text{Eq. 7}$$

$$n \cdot \nabla H = \frac{\pi(1 - A)\cos(\pi X)}{RePr} \qquad \text{Eq. 8}$$

$$n \cdot \nabla H = -\frac{A}{RePr} \qquad \text{Eq. 9}$$

$$n \cdot \nabla H = 0 \qquad \text{Eq. 10}$$

2.2 COMSOL Multiphysics 5.5

Since finite element method requires several iterations, therefore the advantages of COMSOL Multiphysics 5.5 can be obviously shown in this situation. COMSOL Multiphysics 5.5 is a finite element analysis, solver, and Multiphysics simulation software that runs on multiple platforms. It supports standard physics-based user interfaces as well as coupled systems of partial differential equations. COMSOL Multiphysics software simulation results show flow velocity, temperature distribution, and pressure within the vessel. Figure 2 shows the flow chart on how to use Comsol in solving the problem of mixed convection in a square lid driven cavity having heated wall. At the end of the processes, the results will obtain.

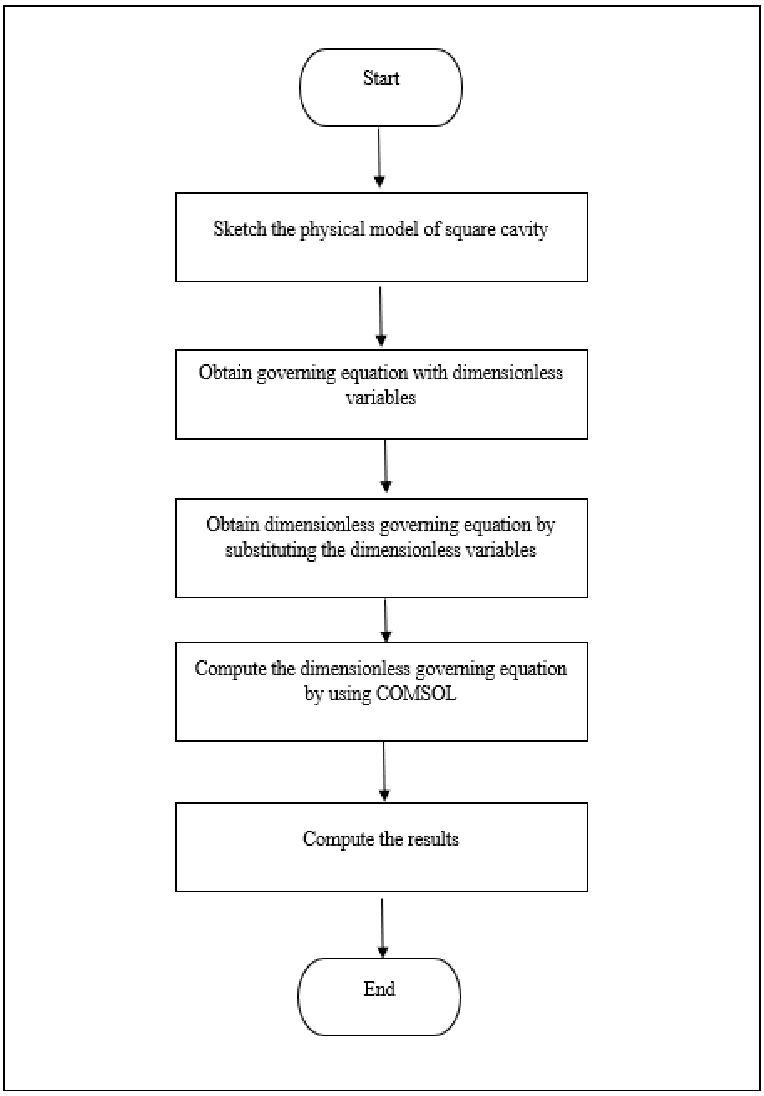


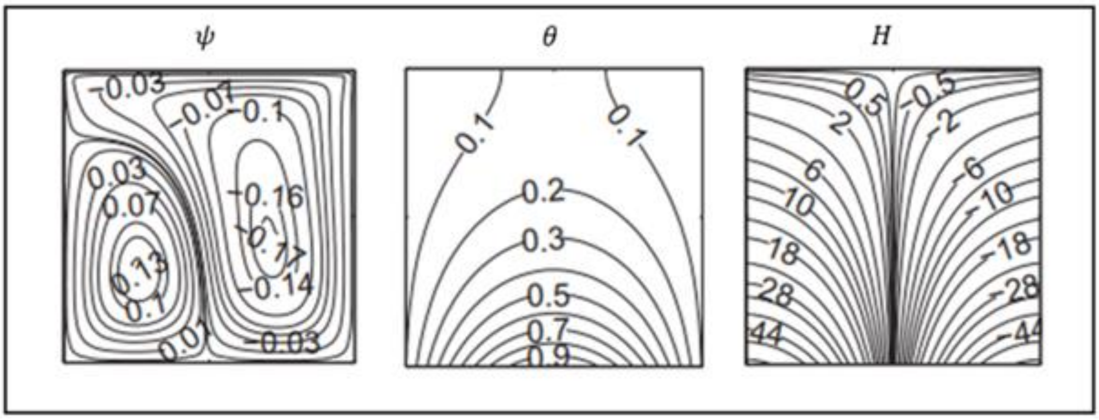
Figure 2: Flow chart in using COMSOL Multiphysics 5.5

3. Results and Discussion

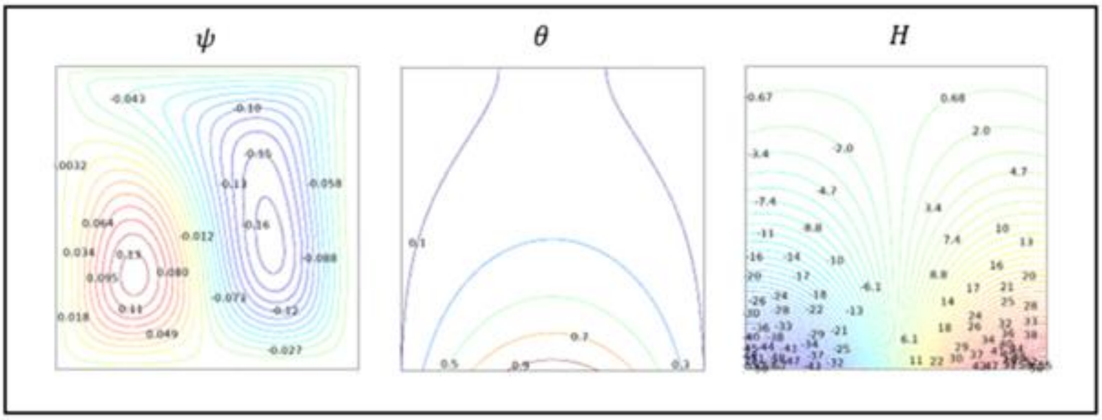
This study focused on the mixed convection heat transfers in a lid driven as the problem and solve by COMSOL Multiphysics version 5.5. This chapter shows three different results which are the streamlines, isotherms and heatlines plots and compare with the previous study by using COMSOL. Streamlines is used to visualize fluid flow, isotherms are used to visualize temperature distribution and heatlines represent velocity vectors and energy flux vectors.

3.1 Streamline, Isotherm and Heatline Plot

From previous research, the streamlines, isotherms and heatlines plot are solved by using the finite element method with a different value of governing parameters, like Prandtl number (*Pr*), Grashof number (*Gr*), Reynolds number (*Re*) based on different thermal aspect ratio (*A*). However, in this study we compute different values of Reynolds number (*Re*) and Prandtl number (*Pr*) to carry out this analysis simulations in streamlines, isotherms and heatlines. Figure 3 to Figure 6 is the streamline and isotherm plot compute by COMSOL Multiphysics version 5.5.



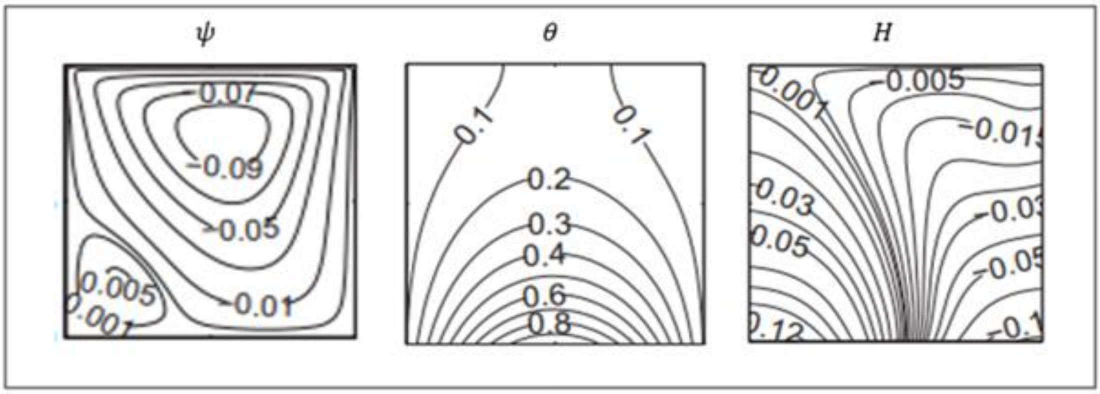
(a)



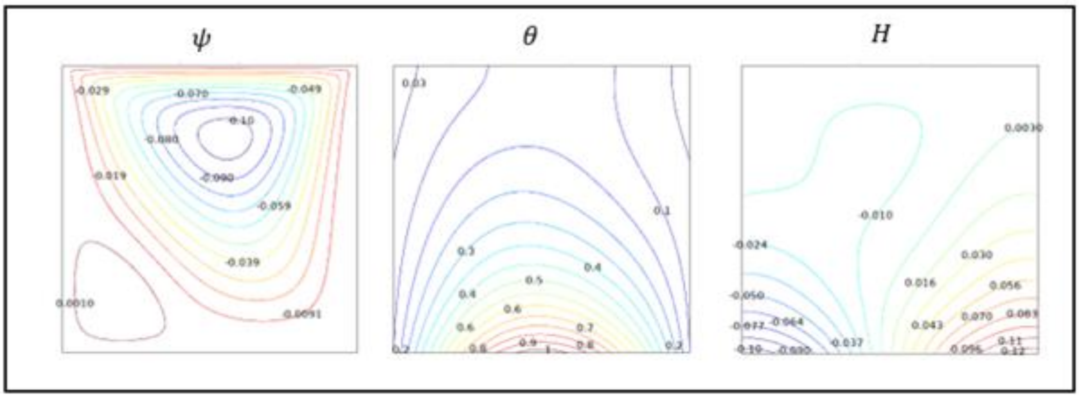
(b)

Figure 3: Streamlines,ψ , isotherms,θ and heatlines,H contours with for Re=1, Pr=0.015, Gr=10^3 and A=0.1 (a) Ramakrishna [5], (b) present

From Figure 3(a-b), the pattern plots of isotherms are symmetrical show that conduction is the major mode of heat transfer due to low Peclet number which are Pe=0.015. The pattern of heatlines plot also like symmetrical and this can show that they are found to be orthogonal to the isotherms indicating the conduction dominant heat transfer. The bottom wall's centre zone has a high degree of temperature variance because the heatlines there are discovered to be dense, indicating a larger heat flow.



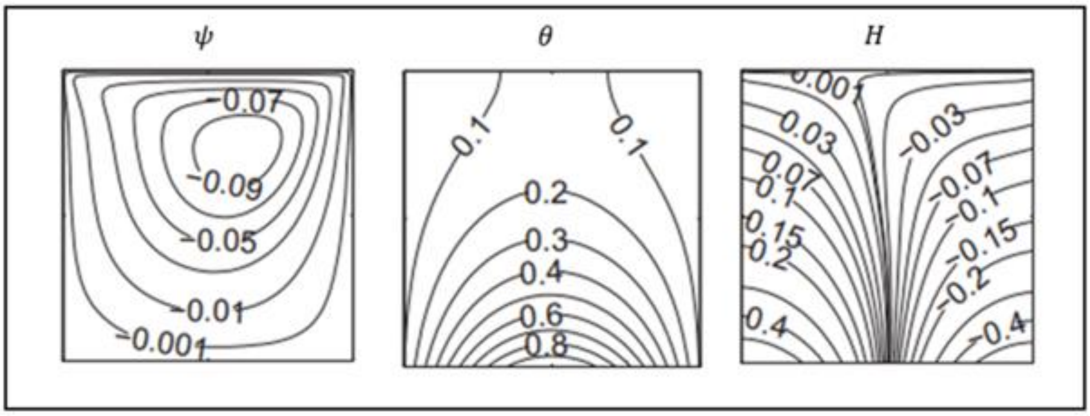
(a)



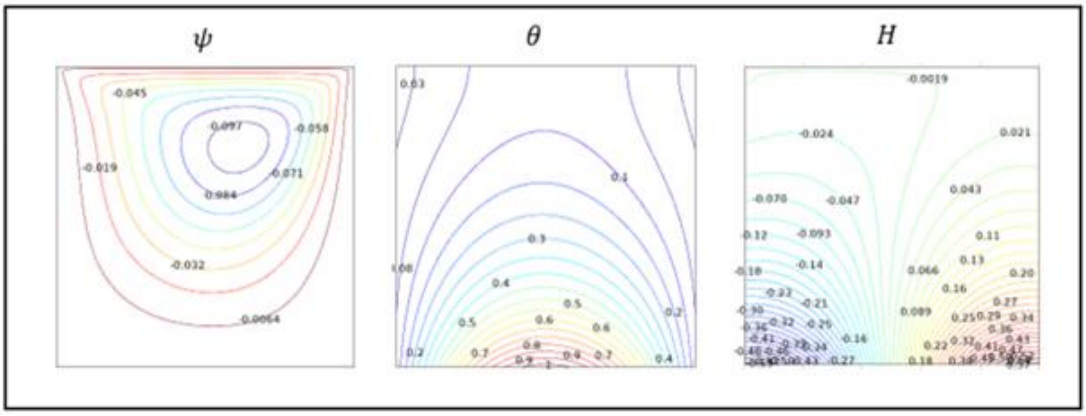
(b)

Figure 4 : Streamlines,ψ , isotherms,θ and heatlines,H contours with for $Re = 10$, $Pr = 0.7$, $Gr = 10^3$ and $A = 0.1$ (a) Ramakrishna [5], (b) present

From streamline plots of Figure 4(a-b), it shows that the main loop occupies most of the cavity at top of the right, whereas the secondary cells occupying small area close to the bottom of the left wall. When *Re* and *Pr* increases, the isotherms do not show symmetrical patterns. This is because viscous force (affected by *Re*) and thermal diffusion (affected by *Pr*) decrease compared to Figure 3(a-b). Heatline plot in previous research has slightly different pattern flows with our study.



(a)



(b)

Figure 5: Streamlines,ψ , isotherms,θ and heatlines,H contours with for Re=100, Pr=0.015, Gr=$10^3$and A=0.1 (a) Ramakrishna [5], (b) present

From Figure 5(a-b), the isotherms distribution patterns also symmetrical like Re=1 due to have effect of low velocity (same low value Pr). However, the streamlines have the significant difference since situation at high Reynolds number have low viscous force. So, it has enough time to transfer the heat and streamline does not have two flow circulations occupy most of the cavity. Heatline plot in previous research has slightly different pattern flows with our study.

4. Conclusion

In this thesis, streamlines, isotherms and heatlines plot was successfully plotted using the COMSOL Multiphysics software. Furthermore, it is showing that the part of theoretical methods can solved easily in COMSOL rather than solved manually using the Partial Difference Equation for Finite Element. Meanwhile, Poisson's equation and laminar flow were used to solve the problem in COMSOL Multiphysics 5.5 to solve the problem. Navier-Stokes equations and heatfunction equations are solved by using Poisson's equation and the mixed convections is solved by laminar flow. Three of the models are related to the problem used in this study, so the equations and boundary conditions of the problem is inserted in COMSOL to create streamline, isotherm and heatlines plot. In this study, the guideline for producing streamlines, isotherms and heatlines plot in COMSOL version 5.5 are showed step by step. To contrast with earlier study, the COMSOL results only displayed the dominant part of the streamlines, isotherms and heatlines plot. Components with small values are not shown in the graph. The result from each software may differ slightly because of upgrade version of COMSOL or the source input for each programmer differs.

In conclusion, the mixed convection problems in square lid driven cavity are solved via finite element method with the assist of COMSOL Multiphysics. Compare with the results of previous study, there are show that the streamlines and isotherms flow patterns are almost equal but heatlines may have slightly different. Based on the results compute from COMSOL Multiphysics, it can conclude that, the isotherms and heatlines have shown symmetrical pattern in the mixed convection of heat transfer at Re=1 with Gr=10^3 when it does not depend on the value of thermal aspect value (A) and Prandtl number (Pr) because of higher value of Richadson number. Besides, heatlines and streamlines vary identically near the core for convection dominant flow at higher Prandtl (Pr=0.7) numbers with Gr= 10^3 irrespective of Re. The streamlines, isotherms and heatlines different values or situations provided in this study can be a reference for future studies.

For the recommendation, COMSOL Multiphysics software should be taught to all university students in relevant courses. This is because technical education can evolve with the technology nowadays in the whole world. Besides, future studies can compare different values or situations in the streamline, isotherm and heatlines plot. This is because it can enhance our understanding of the heatline flows in mixed convection lid driven cavity.

Acknowledgement

The authors would like to thank the Faculty of Applied Sciences and Technology, Universiti Tun Hussein Onn Malaysia for its support.

References

[1] Basak, T., Ramakrishna, D., Roy, S., Matta, A. & Pop, I. (2011). A comprehensive heatline based approach for natural convection flows in trapezoidal enclosures: Effect of various walls heating. *International Journal of Thermal Sciences*, pp.1385-1404.

[2] Kimura S. & Bejan A. (1983). The “Heatline” Visualization of Convective Heat Transfer. *Journal of Heat Transfer*, 105.

[3] Ramakrishna, D., Basak, T., Roy, S. & Pop, I. (2012). Numerical study of mixed convection within porous square cavities using Bejan’s heatlines: Effects of thermal aspect ratio and thermal boundary conditions. *International Journal of Heat and Mass Transfer*, 55(21–22), pp.5436-5448.

[4] Basak, T., Anandalakshmi, R. & Singh, A.K. (2013). Heatline analysis on thermal management with conjugate natural convection in a square cavity. *Chemical Engineering Science*, 93, pp. 67-90.

[5] Biswal, P., Roy, M., Roy, S. & Basak, T. (2017). Analysis of heatline based visualization for thermal management during mixed convection of hot/coldfluids within entrapped triangular cavities. *Journal of the Taiwan Institute of Chemical Engineers*, 77, pp.122-141.

[6] Bondarenko, D.S., Sheremet, M.A., Oztop, H.F. & Ali, M.E. (2019). Natural convection of Al2O3/H2O nanofluid in a cavity with a heat-generating element heatline visualization. *International Journal of Heat and Mass Transfer*, 130, pp.564-574.

[7] Basak, T., Roy, S. & Pop, I. (2009). Heat flow analysis for natural convection within trapezoidal enclosures based on heatline concept. *International Journal of Heat and Mass Transfer*, 52, pp.471-2483.

[8] Ramakrishna, D., Basak, T., Roy, S. & Pop, I. (2012). A complete heatline analysis on mixed convection within a square cavity: Effects of thermal boundary conditions via thermal aspect ratio. *International Journal of Thermal Sciences*, 57, pp.98-111.