Enhanced Knowledge in Sciences and Technology Vol. 3 No. 1 (2023) 080-087

EKST

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

Simulation of Incompressible Fluid in Lid-Driven Cavity using COMSOL

Syahirah Mohammad Rafee[1], Muhamad Ghazali Kamardan[2]*

[1]Kumpulan Wang Simpanan Pekerja, JC 538, Jalan Bestari 5, Bandar Jasin, Bestari Seksyen 2, 77200 Bemban, Melaka, MALAYSIA

[2]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.2023.03.01.010
Received 15 January 2023; Accepted 20 March 2023; Available online 3 August 2023

Abstract: The simulation of incompressible fluid in lid-driven cavity has been carried out. In this study, COMSOL Multiphysics 6.0 was used as a tool to obtain the solution with approximate result. The aim of this study is to apply COMSOL in solving the problem regarding fluid flow in lid-driven cavity and to analyse and compare the streamline provided with the existing study. The dimensionless equations of the flow were modelled in COMSOL to visualize the streamlines and velocity magnitudes for various values of Reynolds number. This study shows that the center of primary vortex tends to move towards the center of the cavity as the Reynolds number increases due to the increased inertia in the flow. Also, the velocity of primary vortex increases as the Reynolds number increase.

Keywords: COMSOL, Incompressible Fluid, Lid-Driven Cavity

1. Introduction

An incompressible fluid is a fluid that maintains its volume over a wide range of pressures and has a constant density throughout the fluid, with no divergence in the flow velocity [1]. The top lid was applied with a tangential velocity to drive the fluid flow in the cavity, while the other three walls had zero-velocity with no-slip conditions. The closed cavity was bound to the zero-pressure point constraint [2]. When the velocity of the lid of a square cavity filled with fluid was fixed to a certain constant value, a spiral flow pattern and two distinct pressure zones became visible in the upper corners of the lid [3].

A multi-grid technique to solve the Navier-Stokes equations and provided solutions for Reynolds numbers ranging from 100 to 10,000 by [4]. In the existing study, which was numerically conducted by [5], a square, deep lid-driven cavity was used, and the finite difference method was applied. According to the results, when the Reynolds number increases, the primary vortex's center moves downward in relation to the top lid. However, when the Reynolds number is more than 1,000, the primary vortex's

*Corresponding author: mghazali@uthm.edu.my

center stays the same below the top lid. This indicates how the primary vortex's movement and structure are significantly influenced by the Reynolds number. To determine whether a fluid is turbulent or laminar, a dimensionless value called the Reynolds number have been used. It represents the proportion of inertial and viscous forces [6].

COMSOL Multiphysics is a powerful interactive environment for modelling and solving all kinds of scientific and engineering problems based on partial differential equations (PDE) [7]. Lid-driven cavities have been a fluid dynamics research area for years due to the simple geometry and well-defined boundary conditions [8]. Therefore, this study was conducted for two-dimensional, incompressible, and laminar fluid flow, similar as the existing study. The objective of this study is to solve a problem involving incompressible fluid flow in a lid-driven cavity using COMSOL Multiphysics version 6.0. Apart from that, this study aims to provide streamline of incompressible fluid in lid-driven cavity and to analyse compare the streamline from the existing study by [5].

2. Methodology

Based on Figure 1, x and y are the coordinate directions and variables u and v are the velocity in direction x and y, respectively. ρ is the density while μ is the dynamic velocity of the fluid. The boundary conditions are:

Top wall: $u = 1, v = 0$

Bottom wall: $u = 0, v = 0$

Right vertical wall: $u = 0, v = 0$

Left vertical wall: $u = 0, v = 0$

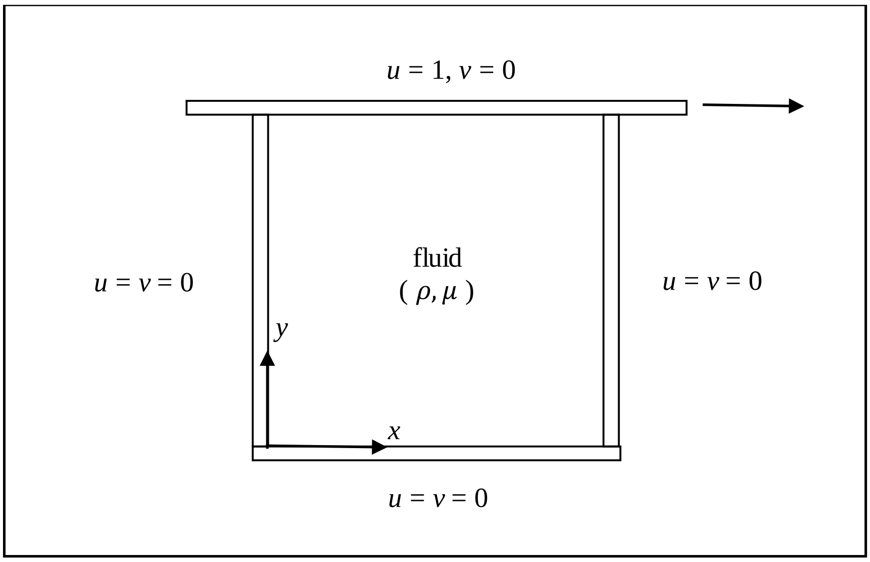


Figure 1: Diagram of lid-driven cavity flow [9]

2.1 Governing Equations

All the equations subjected to the boundary conditions were used in this study since the research of an incompressible fluid in lid-driven cavity was the same problem as Munir *et al.* [5]. The equation of incompressible, steady-state, two-dimensional streamline flow in x and y directions, as well as the absence of other effects are considered in this study, resulting in the continuity and combination of mass and momentum equations (the Navier-Stokes equations) of the mathematical model as follows:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad Eq.1$$

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\left(\frac{\partial p}{\partial x}\right)+w\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \qquad Eq.2$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\left(\frac{\partial p}{\partial y}\right)+w\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \qquad Eq.3$$

where v is velocity in direction y, p is pressure, ρ is density and w is kinematic viscosity.

By eliminating the pressure term from $Eq.2$ and $Eq.3$, the equation is rewritten in terms of vorticity function ω, gives

$$\frac{\partial \omega}{\partial t}+u\frac{\partial \omega}{\partial x}+v\frac{\partial \omega}{\partial y}=w\left(\frac{\partial^2 \omega}{\partial x^2}+\frac{\partial^2 \omega}{\partial y^2}\right) \qquad Eq.4$$

where vorticity function $\omega=\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}$

In terms of stream function (ψ), $u=\frac{\partial \psi}{\partial y}$ and $v=-\frac{\partial \psi}{\partial x}$, the equation defining the vorticity becomes

$$\omega=-\left(\frac{\partial^2 \psi}{\partial x^2}+\frac{\partial^2 \psi}{\partial y^2}\right) \qquad Eq.5$$

The dimensionless variables are transformed using the following variables

$$\Psi=\frac{\psi}{u_\infty H},\Omega=\frac{\omega H}{u_\infty}$$
$$U=\frac{u}{u_\infty},V=\frac{v}{u_\infty} \qquad Eq.6$$
$$X=\frac{x}{H},Y=\frac{y}{H},\tau=\frac{t u_\infty}{H}$$

where u_∞ is lid velocity and H is height of the cavity.

By substituting the dimensionless variables in $Eq.6$, $Eq.4$ and $Eq.5$ become

$$\frac{\partial \Omega}{\partial \tau}+U\frac{\partial \Omega}{\partial X}+V\frac{\partial \Omega}{\partial Y}=\frac{1}{Re}\left(\frac{\partial^2 \Omega}{\partial X^2}+\frac{\partial^2 \Omega}{\partial Y^2}\right) \qquad Eq.7$$

and

$$\Omega=-\left(\frac{\partial^2 \Psi}{\partial X^2}+\frac{\partial^2 \Psi}{\partial Y^2}\right) \qquad Eq.8$$

where the dimensionless parameter of Reynolds number, Re is defined as

$$Re=\frac{u_\infty H}{w} \qquad Eq.9$$

2.2 COMSOL Multiphysics Simulation

In this study, two-dimensional space dimensions were generated using square geometry for cavity shape, no-slip wall conditions, and zero velocity at the top lid. The models in COMSOL that have been used are Laminar Flow and Poisson's Equation. The models are based on the dimensionless equations from this study. As the models are related to this study, the equations and boundary conditions are inserted into COMSOL. Also, the compressibility was changed to incompressible, and the parameter values were added with $Re = 100, 400, 1{,}000, 5{,}000$ and $10{,}000$ as in the existing study. Hence, the streamline and velocity magnitude of each value of Reynolds number were computed.

3. Results and Discussion

3.1 Streamline

In the previous research, the streamlines were computed using FORTRAN language with three values of Reynolds number, 100, 400 and 1,000 by defining the height of cavity and constant velocity of the top lid of the cavity. However, in this study two more values of Reynolds number, 5,000 and 10,000 are added.

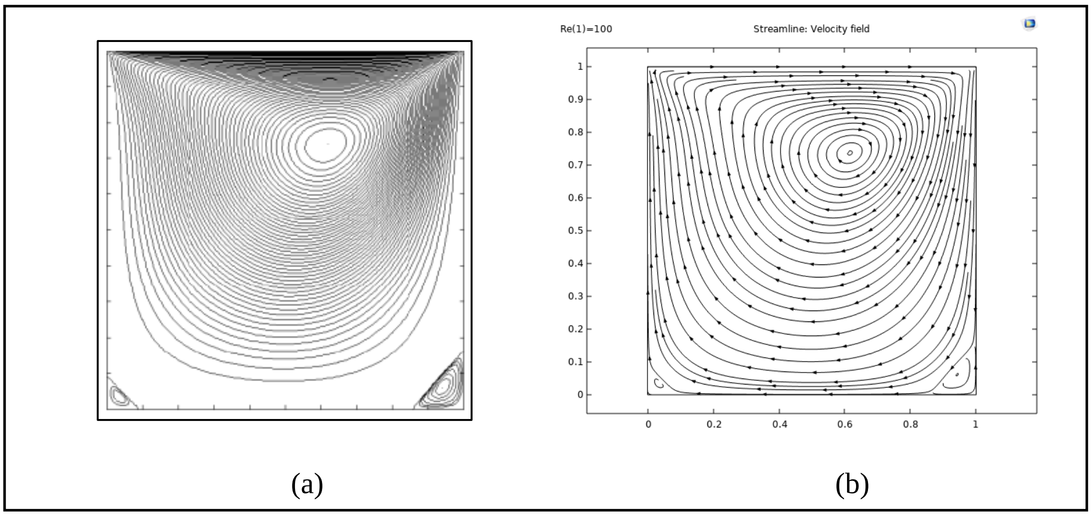
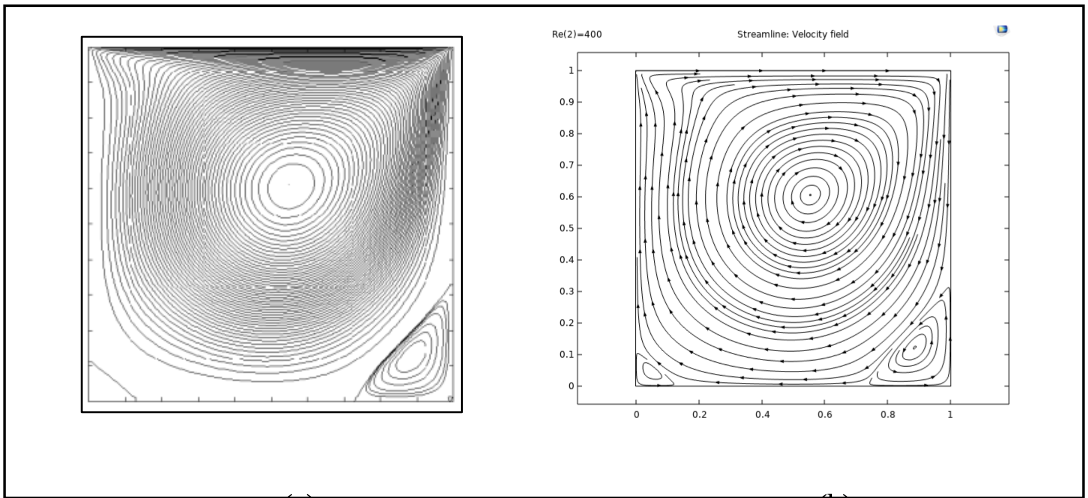
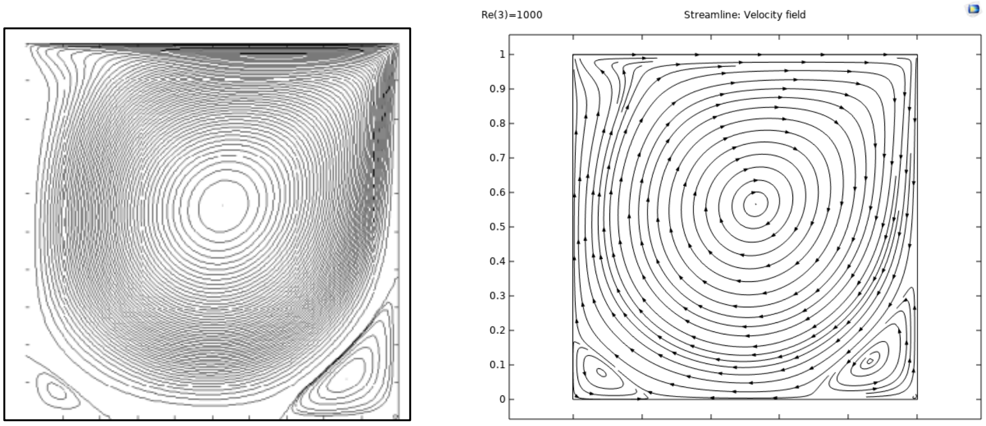


Figure 2: Streamline for $Re = 100$ (a) from previous study by Munir *et al.* [5] and (b) using COMSOL



(a) (b)

Figure 3: Streamline for $Re = 400$ (a) from previous study by Munir *et al.* [5] and (b) using COMSOL



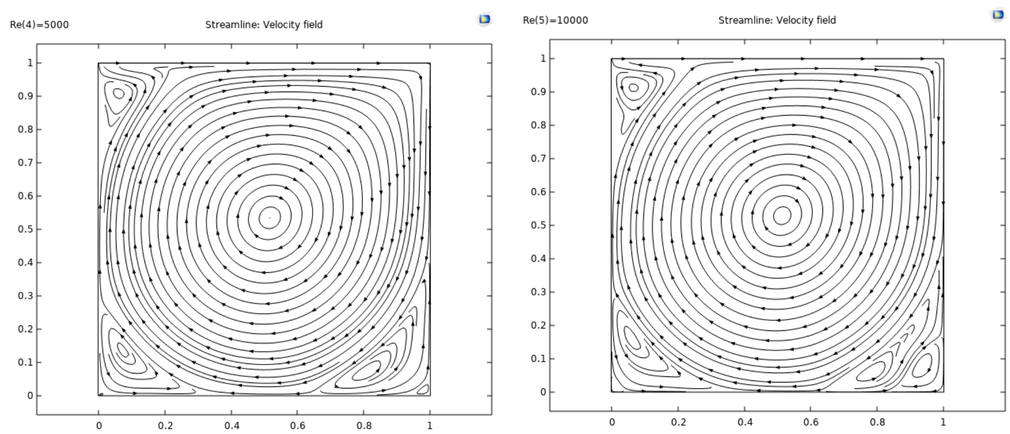
(a) (b)

Figure 4: Streamline for $Re = 1,000$ (a) from previous study by Munir *et al.* [5] and (b) using COMSOL

Figure 2 to Figure 4 shows the streamline of previous research by Munir *et al.* [5] and the streamline computed by COMSOL Multiphysics 6.0. For $Re = 100$ from both studies, there are the formation of primary vortex at about one-third of the cavity from the top and two smaller corner vortices at the bottom of cavity. Due to separation at the corners, the two smaller corner vortices that rotate anticlockwise are formed when the primary vortex rotates in a clockwise direction.

As the Reynolds number increase, the center of primary vortex tends to move towards the center of the square cavity. Also, the corner vortices gradually appear at the bottom corner of the cavity and become larger.

Figure 5 shows the streamline for $Re = 5{,}000$ and $Re = 10{,}000$. The primary vortex remains constant as the Reynolds number increase beyond 1,000. An induced corner vortex in upper left corner of the cavity appears and become clearer. Also, a tertiary vortex appears at the bottom left of the cavity for $Re = 10{,}000$.



(a) (b)

Figure 5: Streamline for (a) $Re = 5,000$ and (b) $Re = 10,000$ by COMSOL

3.2 Velocity Magnitude

Apart from streamline, this study also computes the velocity magnitude to demonstrate the speed of the primary vortex for each Reynolds number which are 100, 400, 1,000, 5,000 and 10,000.

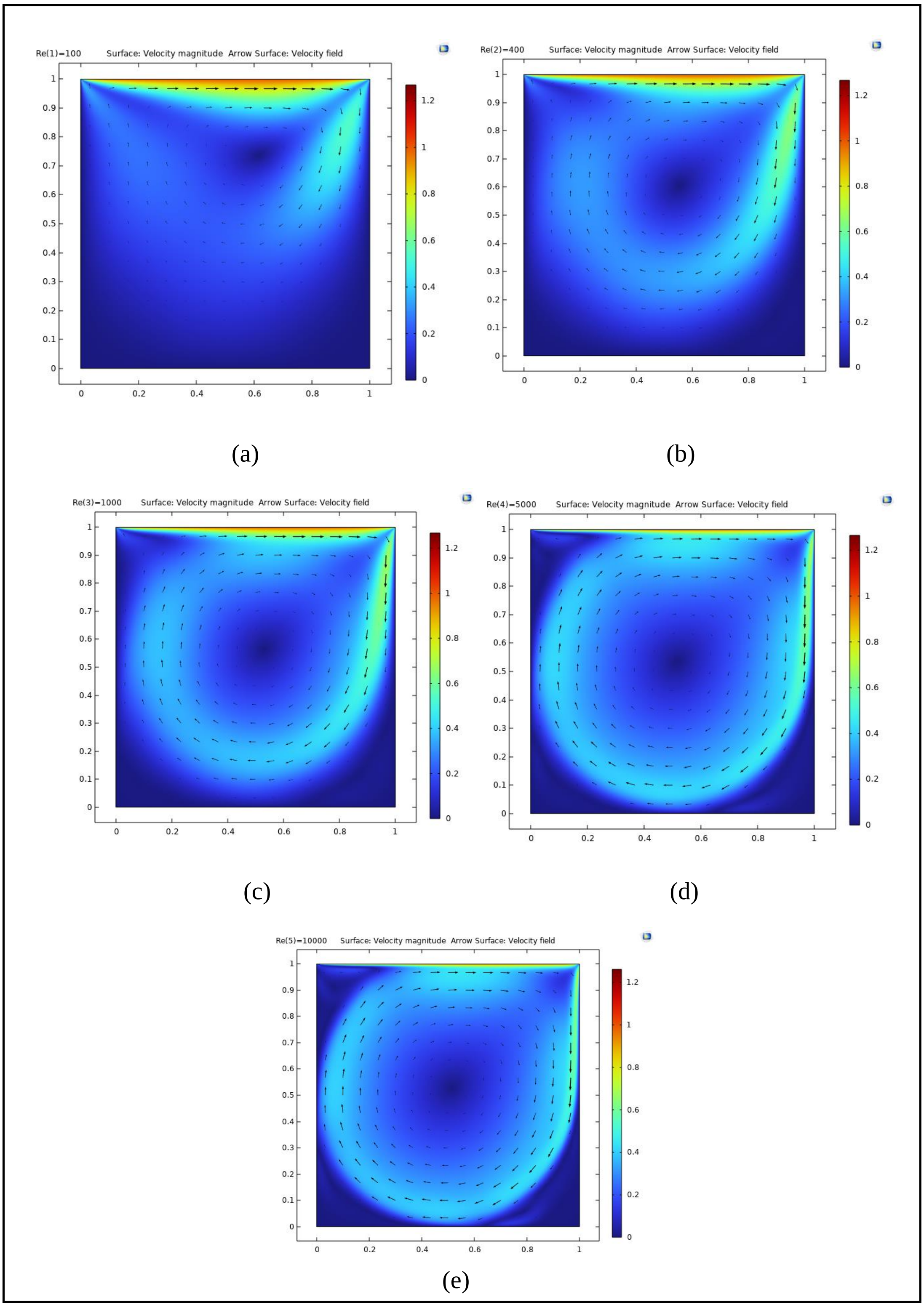


Figure 6: Velocity magnitude for (a) $Re = 100$, (b) $Re = 400$, (c) $Re = 1,000$, (d) $Re = 5,000$ and (e) $Re = 10,000$

Figure 6 shows the velocity magnitude for $Re = 100, 400, 1{,}000, 5{,}000$, and $10{,}000$ computed by COMSOL. In all cases, the fluid velocity approaches one near the top moving wall and zero near the no slip side and bottom walls. For the higher Reynolds number flow ($Re = 10{,}000$), the vortex extends more significantly into the cavity due to the increased inertia. The central vortex rotates faster for $Re = 10{,}000$ than for $Re = 100$ due to the increased inertia in the flow for the higher Reynolds number. Lower velocity regions appear in the bottom and left corners of the cavity where the secondary vortices are located.

4. Conclusion

In summary, COMSOL Multiphysics 6.0 have been used to solve the problem of incompressible fluid in lid-driven cavity. In contrast to the FORTRAN language that was applied in the previous studies by Munir *et al.* [5], it is easier to compute the solution using COMSOL because it contains a limited amount of input which makes it easy to set up the analysis. In FORTRAN language, the coding was required to predict the fluid flow behaviour. The generated streamlines were compared with the previous study by Munir *et al.* [5]. It was found that the result computed by COMSOL Multiphysics 6.0 are the same as previous study. The results show that the center of primary vortex tends to move towards the center of the cavity when the Reynolds number increase. Also, the central vortex's velocity of higher Reynolds number rotates faster than the lower Reynolds number. In conclusion, the position and velocity of primary vortex are significantly affected by the Reynolds number.

Acknowledgement

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

References

[1] Sambavaram, S. R., & Sarin, V. (2002). A High Performance Algorithm for Incompressible Flows Using Local Solenoidal Functions. doi:10.1109/IPDPS.2002.1015525

[2] Straccia, A. (2018, May 8). How to Solve a Classic CFD Benchmark: The Lid-Driven Cavity Problem. Retrieved from COMSOL: https://www.comsol.com/blogs/how-to-solve-a-classic-cfd-benchmark-the-lid-driven-cavity-problem/#References

[3] Lorena A. Barba, N. C. (2017). Essential skills for reproducable research computing. Retrieved from https://barbagroup.github.io/essential_skills_RRC/

[4] Ghia, U., Ghia, K. N., & Shin, C. T. (1982). High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method. Journal Of Computational Physics , 387-411.

[5] Munir, F. A., Zin, M. R., Sidik, N. A., & Salim, M. A. (2012). Accurate Numerical Prediction of Incompressible Fluid Flow in Lid-Driven Cavities. Applied Mechanics and Materials Vols. 110-116, 4365-4372. doi:10.4028/www.scientific.net/AMM.110-116.4365

[6] Rehm, B., Haghshenas, A., Paknejad, A. S., & Schubert, J. (2008). Chapter Two - Situational Problems in MPD. Managed Pressure Drilling, 39-80.

[7] Venkataraman, D. P. (2013). *An Introduction to COMSOL*. Retrieved from https://people.rit.edu/pnveme/personal/VenkatCOMSOL42/IntroductionCOMSOL/index.html

[8] Shankar, P. (1993). The eddy structure in Stokes flow in a cavity. Journal of Fluid Mechanics, 371-383.

[9] Marchi, C. H., Suero, R., & Araki, L. K. (2009). The Lid-Driven Square Cavity Flow: Numerical Solution with a 1024 x 1024 Grid. 186-198.