Knowledge-based Engineering and Sciences (2023), **4**, (2), 51–59
doi:10.51526/kbes.2023.4.2.51-59

RESEARCH PAPER

Navier-Stokes Equations and High-Resolutions: Advancements in Accurate Incompressible Flow Simulations

Hussein Togun,*,[1] Raad Z. Homod,[2] Emad Sadeghinezhad,[3] and Salim Newaz Kazi[4]

[1]Department of Biomedical Engineering, College of Engineering, University of Thi-Qar, Thi-Qar 64001, Iraq
[2]Department of Oil and Gas Engineering, Basrah University for Oil and Gas, Iraq
[3]School of Minerals and Energy Resources Engineering, UNSW, Sydney, NSW 2052, Australia
[4]Department of Mechanical Engineering, Faculty of Engineering, Universiti Malaya, 50603 Kula Lumpur, Malaysia
*Corresponding author. Email: htokan_2004@yahoo.com,hussein-tokan@utq.edu.iq

(Received 23 June 2023; revised 13 August 2023; accepted 17 August 2023; first published online 31 August 2023)

Abstract

The lid-driven cavity serves as a standard test case for validating computational methods in fluid dynamics. This problem involves a 2D square cavity with a tangentially moving upper wall, resulting in a flow pattern characterized by a central vortex and smaller vortices in the corners. The Reynolds number significantly influences the size and number of vortices in the flow. This model showcases the process of defining appropriate boundary conditions for the lid-driven cavity problem using COMSOL Multiphysics. Furthermore, the model's results for the velocity profile, vortex size, and location were compared to a previously published study providing a basis for validating the accuracy and reliability of the computational approach. Increasing Reynolds numbers induced stronger vortices, enhanced flow mobility, and turbulence, resulting in the shifting positions of vortices within the lid-driven cavity, highlighting the significant influence of Reynolds number on fluid flow behaviour.

Keywords: Lid-driven cavity; Laminar flow; COMSOL multiphysics; Reynolds number.

1. Introduction

Natural convection in cavities has garnered significant attention due to its effectiveness and wide range of applications across various sectors and industries [1-3]. These applications span building and construction supplies [4], electronics, thermal conversion of solar and renewable energy, refrigeration, cooling of nuclear reactors, and many more. The study of natural convection has been ongoing for over a century, with increasing interest among researchers since the 1980s. However, a complete understanding of this phenomenon remains a constant challenge.

Previous studies have predominantly utilized numerical methods to investigate natural convection phenomena [5-12]. These studies have provided valuable experimental data that are crucial for understanding the underlying mechanisms of natural convection and validating numerical models. Khalifa extensively reviewed natural convection in enclosures, considering various thermal conditions and incorporating spatial and temporal variations [13, 14]. Karatas and Derbentli conducted experimental research on rectangular enclosed cavities filled with air, focusing on one active cavity. They examined variations in temperature profiles and local Nusselt numbers along the height of the cavity [15], [16].

In analyzing axially symmetric flows within rectangular cavities, a two-dimensional approach can be employed by considering a limited number of boundary conditions, such as the Rayleigh number and the direction of the gravity vector. This case has been extensively studied, exploring various configurations, including partial heating or cooling of one wall and, in some cases, varying inclination angles of the cavities [17, 18]. These studies have contributed significant insights into understanding this highly complex phenomenon and paved the way for effectively utilizing this heat transfer mode. However, due to inherent challenges in measuring relevant variables, limited experimental data is available, leading to several unresolved issues within the field.

This study analyses a 2D square enclosure with a unique flow pattern derived from the tangential velocity applied to the upper wall. The resulting flow has a prominent vortex at the centre of the cavity, accompanied by smaller eddies in the corners. By varying the Reynolds number, the size and quantity of these vortices can be altered. This model aims to illustrate the method for defining the boundary conditions specific to this issue within the COMSOL Multiphysics software. In conclusion, the study has implications for flow control, mixing procedures, and comprehending fluid dynamics. Its viability is supported by its controlled parameters, simplified shape, well-proven techniques, robust simulation tools, and visually perceptible phenomena. These elements influence the significance and viability of the research.

2. Methodology

The lid-driven cavity problem is most elegantly modeled using a non-dimensional form of the Navier-Stokes equations (See Figure 1). Laminar Flow physics in COMSOL Multiphysics solves the traditional Navier-Stokes equations. For an incompressible stationary flow with no body forces, they are defined as:

$$\rho(u.\nabla)u = -\nabla p + u\nabla^2 u \tag{1}$$

By non-dimensionalization the velocity $\left(u^* = \frac{u}{v}\right)$, pressure $\left(p^* = \frac{p}{(\rho v)^2}\right)$, and length scale $\left(r^* = \frac{r}{L}, \nabla^* = L\nabla\right)$, the non-dimensional Navier-Stokes equations can be written as:

$$\left(u^* \cdot \nabla^*\right) u^* = -\nabla p^* + \frac{1}{Re}\nabla^{*2} u^* \tag{2}$$

Where, $Re = \left(\frac{\rho v L}{\mu}\right)$ is the Reynolds number.

Solving the non-dimensional form of the Navier-Stokes equations offers the benefit of characterizing flow based solely on the Reynolds number. COMSOL Multiphysics can effectively solve the non-dimensional form by appropriately selecting density and viscosity values. The flow domain consists of a square cavity with a side length of 1, serving as the characteristic length scale. The fluid density is set to 1, and the viscosity is defined as one divided by the Reynolds number $\left(\frac{1}{Re}\right)$. The upper wall has a prescribed horizontal velocity of 1, while the other walls are considered no-slip boundaries with zero velocity.

To improve the resolution of boundary layers and corner vortices in the flow, a mapped mesh is utilized with distributions prioritizing a higher concentration of elements near the walls (See Fig 1). This technique is precious when solving for flows with higher Reynolds numbers. This meshing approach allows four-sided geometries to be discretized efficiently while effectively resolving the boundary layer.

The study employs an auxiliary sweep method to efficiently solve for a range of Reynolds numbers (100 to 10,000). This technique involves solving for each parameter in sequence, utilizing the solution of each parameter as the initial condition for the next one. By doing so, the computation is accelerated.

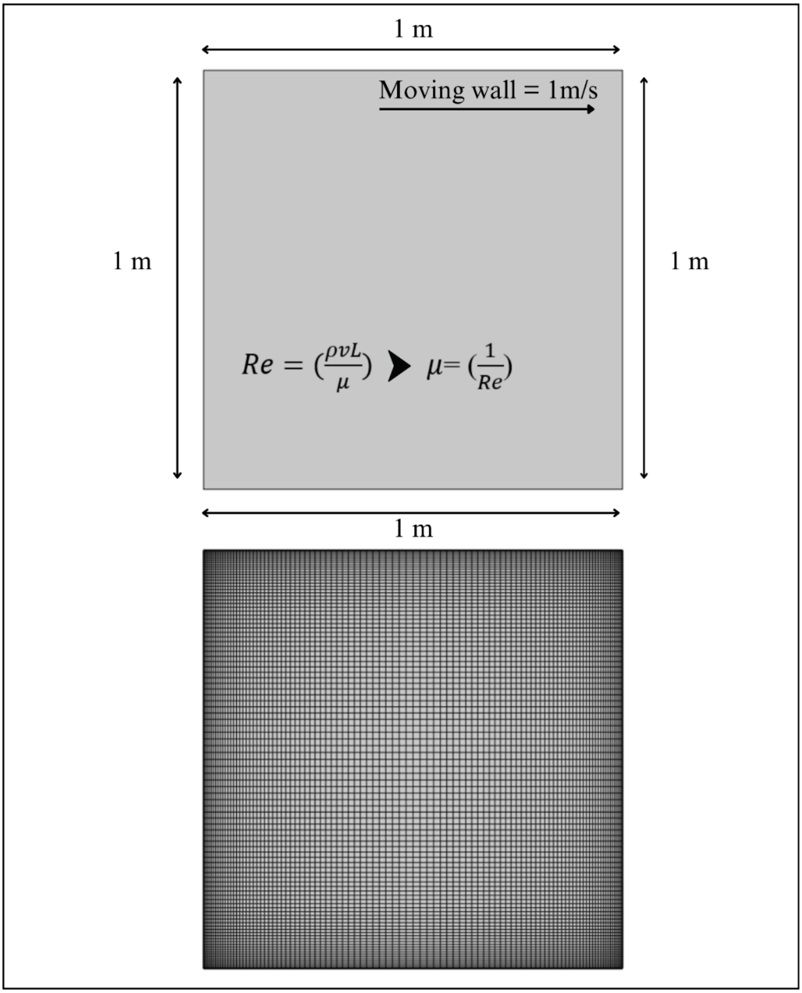

Figure 1: *Geometry and computational domain of the current problem.*

This approach, known as nonlinearity ramping, is beneficial for enhancing convergence in highly nonlinear models.

3. Results and Discussion

3.1 Validation and Verification

As per Figs 2 - 3, the current results were validated and verified with the previous benchmark study [19]. The results unequivocally demonstrate a commendable concurrence between the present computational model and the preceding investigation, as evident from the conformity of the contour plots and streamlined distributions of the velocity magnitude. This striking coherence not only attests to the accuracy and reliability of the current computational framework but also substantiates its suitability for further exploration and in-depth analysis in the field.

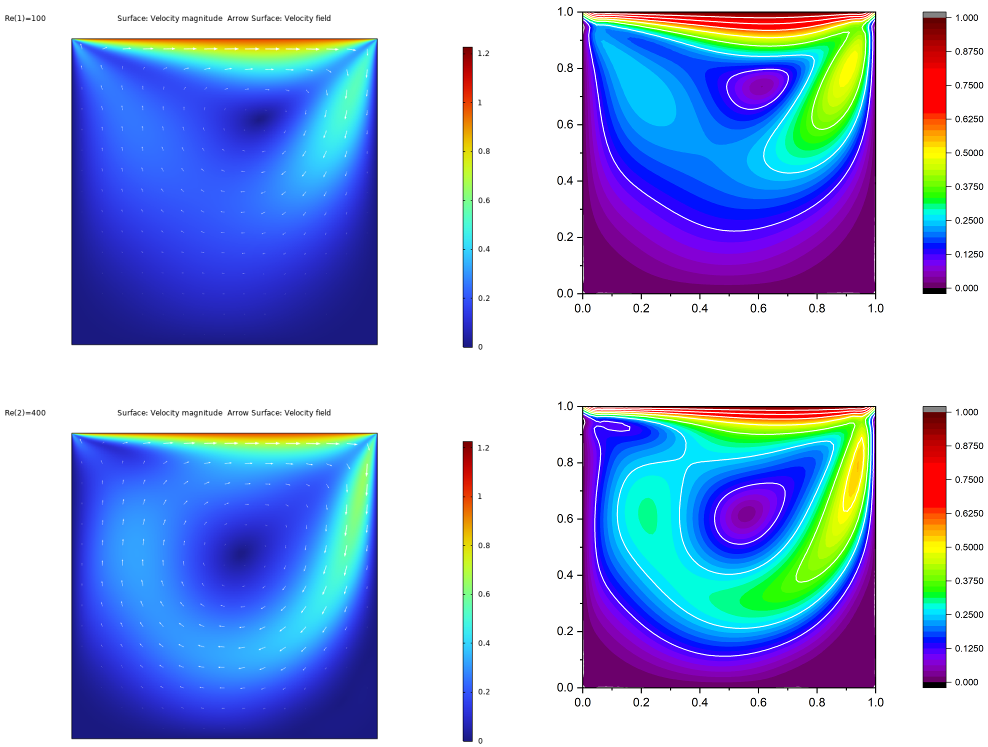

Figure 2: *Comparisons of velocity surfaces at Re = 100 and Re = 400.*

3.2 Contours and Streamlines of Velocity

Figures 4 – 5 depict the Contours and Streamlines of Velocity at different Reynolds numbers (Re) ranging from 100 to 10,000. Increasing the Reynolds number causes noticeable changes in the positions of the center, right, and left vortices within the lid-driven cavity. The flow remains relatively steady at lower Reynolds numbers (e.g., Re = 100, 400), and the vortices maintain consistent locations. As the Reynolds number increases, the fluid velocity intensifies, resulting in stronger vortices. Consequently, the vortices start to shift and reorganize within the cavity. At higher Reynolds numbers (e.g., Re = 1,000, 3,200, 5,000), eddies undergo further displacement due to increased flow mobility and turbulence. The intensified flow forces disrupt the stability of the vortices, causing them to relocate within the cavity. For even higher Reynolds numbers (e.g., Re = 7,500, 10,000), eddies experience displacement as flow turbulence becomes more pronounced. The intricate interactions between fluid particles and flow forces determine the specific positions of the vortices.

3.3 Velocity Profiles

Figures 6 and 7 present the Velocity profiles at the vertical and horizontal centrelines, specifically the x-component and y-component velocities. In Figure 6, the velocity profiles at various Reynolds numbers are distributed within the range of 0–1. On the other hand, Figure 7 displays velocity profiles at different Reynolds numbers that are distributed between 0 and 0. These velocity profiles provide valuable insights into the flow characteristics and behaviour at different Reynolds numbers. Figure 6 indicates that as the Reynolds number increases, the velocity values in the x-component

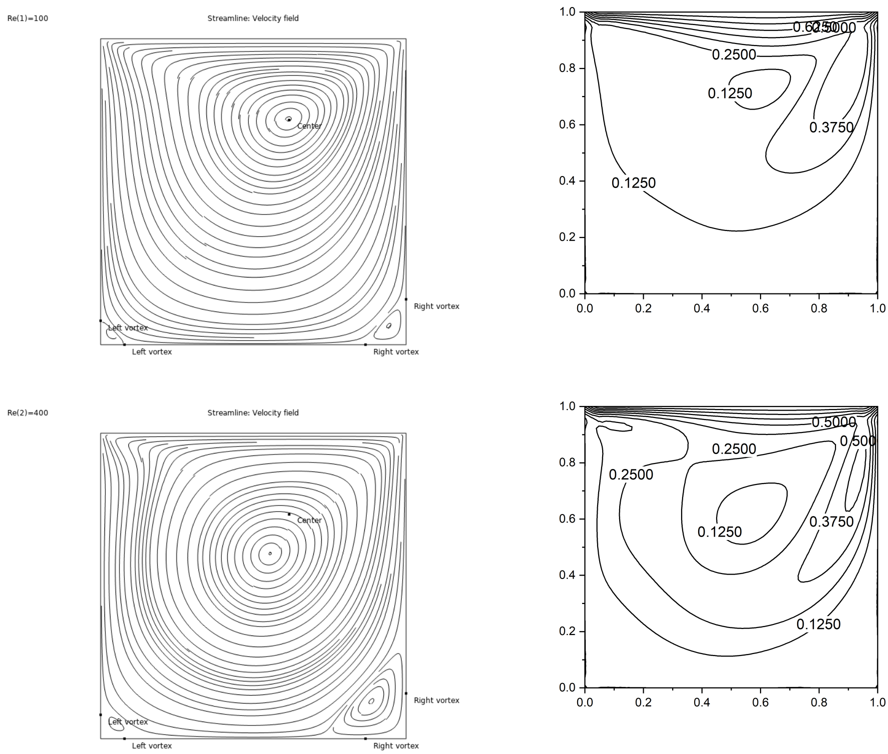

Figure 3: *Comparisons of velocity streamlines at Re = 100 and Re = 400.*

and y-component profiles tend to approach the upper limit of 1. This suggests that higher Reynolds numbers result in faster flow velocities along vertical and horizontal centrelines. In contrast, Figure 7 shows velocity profiles distributed between 0 and 1 across the range of Reynolds numbers. This suggests that the velocities in the y-component profiles remain relatively low or negligible. These profiles indicate that the x-component velocities primarily dominate the flow, while the y-component velocities contribute less significantly to the overall flow behaviour.

4. Conclusions and Recommendations

The problem involves a 2D square cavity with a tangentially moving upper wall, which generates a flow pattern characterized by a prominent central vortex and smaller vortices in the corners. The behaviour of the vortices is influenced by the Reynolds number, which determines their size and number within the flow. Key observations from the study are as follows:

The validation and verification of the current computational model with a previous benchmark study [4] exhibited strong agreement in contour plots and streamline distributions, affirming the model's accuracy, reliability, and suitability for further investigations.

Increasing Reynolds numbers induced stronger vortices, enhanced flow mobility, and turbulence, resulting in the shifting positions of vortices within the lid-driven cavity, highlighting the significant influence of Reynolds number on fluid flow behaviour.

Reynolds number-dependent Velocity profiles at the vertical and horizontal centrelines illustrated the impact of Reynolds number on flow velocities, with higher Reynolds numbers driving increased x-component velocities and minimal influence from the y-component velocities.

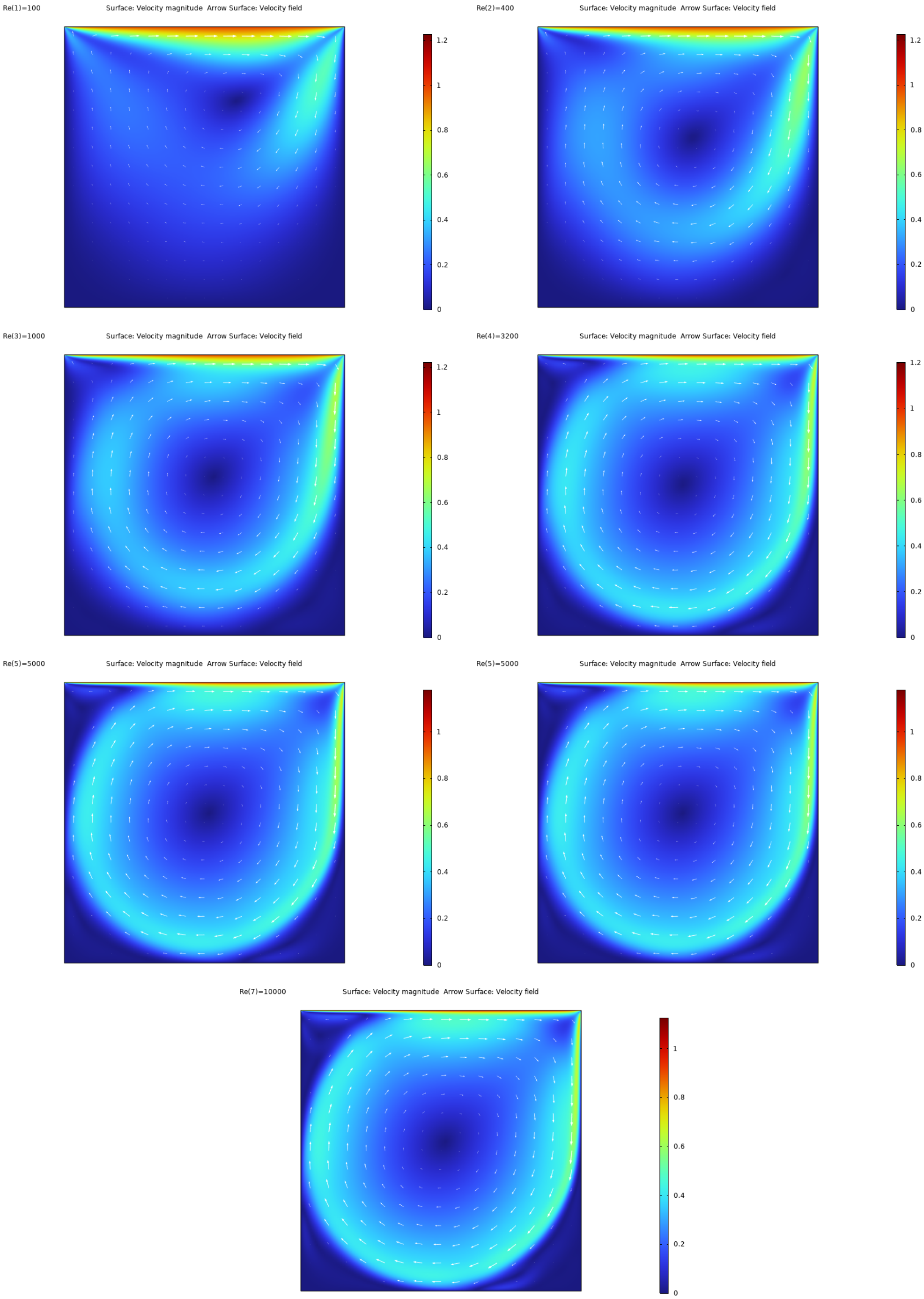

Figure 4: *Velocity surface at different Reynolds numbers.*

It is advised to investigate vortex interactions and how they affect future heat and mass transmission inside the cavity. Investigating diverse geometrical elements like cavity aspect ratios might help us

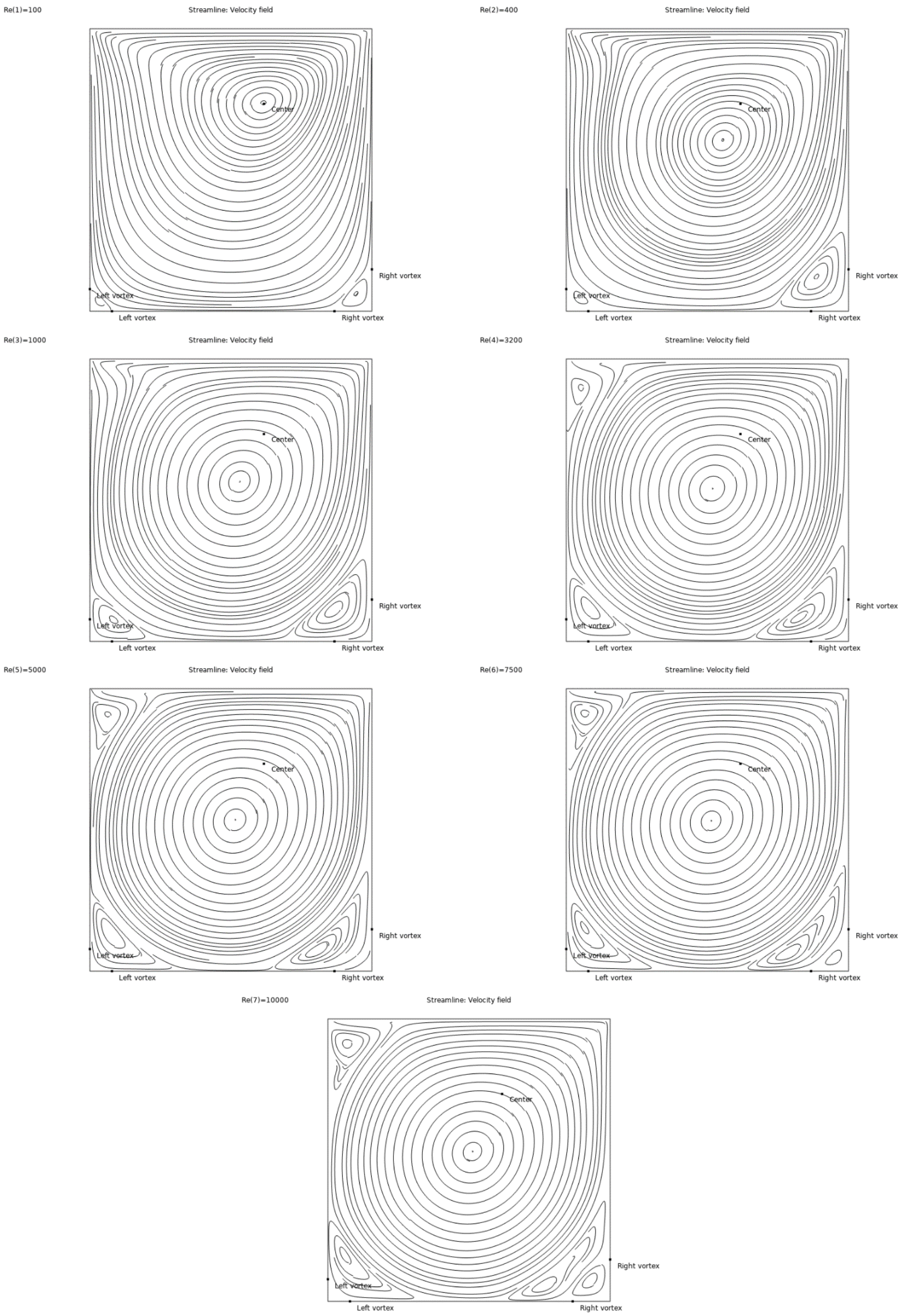

Figure 5: *Velocity streamlines at different Reynolds numbers.*

better understand vortex dynamics. Additional difficulties might be discovered by using sophisticated turbulence models and considering three-dimensional simulations. The practical applications of the study's conclusions would be strengthened by experimental validation.

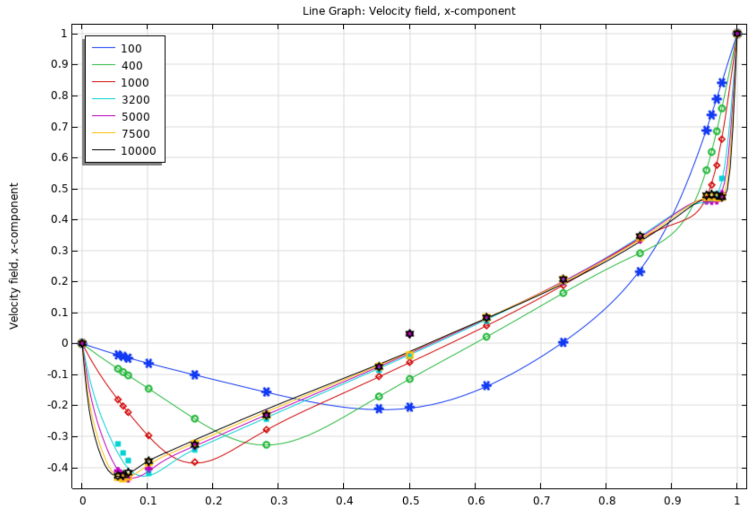

Figure 6: *Velocity profiles at the vertical centreline.*

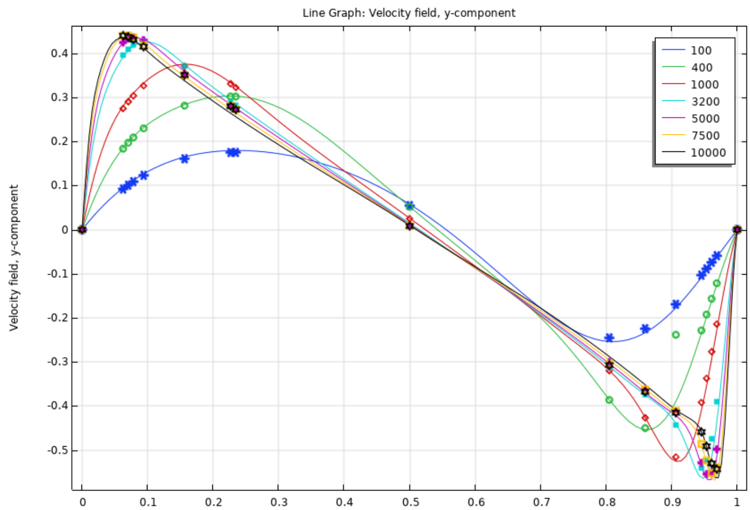

Figure 7: *Velocity profile at the horizontal centreline.*

Funding: There is no funds received.
Conflicts of Interest: The authors declare no conflict of interest.

References

[1] T. Scott, D. Ewim, and A. Eloka-Eboka, "Experimental study on the influence of volume concentration on natural convection heat transfer with Al2O3-MWCNT/water hybrid nanofluids," Materials Today: Proceedings, 2023.
[2] T. A. Franco et al., "The effects of discrete conductive blocks on the natural convection in side-heated open cavities," Applied Thermal Engineering, p. 119613, 2022.
[3] A. I. Alsabery et al., "Convection heat transfer in enclosures with inner bodies: A review on

single and two-phase nanofluid models," Renewable and Sustainable Energy Reviews, vol. 183, p. 113424, 2023.
[4] M. Leporini, F. Corvaro, B. Marchetti, F. Polonara, and M. Benucci, "Experimental and numerical investigation of natural convection in tilted square cavity filled with air," Experimental Thermal and Fluid Science, vol. 99, pp. 572-583, 2018.
[5] J. S. Yang, J. K. Min, C. Yang, and K. Jung, "Numerical studies of natural convection phenomena for a vertical cylinder with multiple lateral baffles in triangular and hexagonal enclosures," Case Studies in Thermal Engineering, vol. 45, p. 102971, 2023.
[6] J. Alzabut, S. Nadeem, S. Noor, and S. M. Eldin, "Numerical analysis of Magnetohydrodynamic convection heat flow in an enclosure," Results in Physics, p. 106618, 2023.
[7] K. U. Rehman, W. Shatanawi, H. M. Bahaidarah, S. Abbas, and A. Khan, "Thermal case study of nanofluid flow in partially heated rectangular enclosure rooted with sinusoidal heated rods and inclined magnetic field," Case Studies in Thermal Engineering, vol. 45, p. 102982, 2023.
[8] T. Aziz, R. U. Haq, M. A. Sadiq, and H. Bahaidarah, "Thermal performance of MHD natural convection flow in a concentric semi-circle porous enclosure having corrugated radius," International Communications in Heat and Mass Transfer, vol. 146, p. 106905, 2023.
[9] K. U. Rehman, A. Khan, S. Abbas, and W. Shatanawi, "Thermal analysis of micropolar nanofluid in partially heated rectangular enclosure rooted with wavy heated rods," Case Studies in Thermal Engineering, vol. 42, p. 102701, 2023.
[10] S. Alqaed, J. Mustafa, F. A. Almehmadi, and M. Sharifpur, "Study of heat transfer and the nanofluid flow inside a rectangular enclosure with five rectangular baffles in the middle affected by a magnetic field," Engineering Analysis with Boundary Elements, vol. 148, pp. 126-136, 2023.
[11] D. Taloub, A. Bouras, A. J. Chamkha, and M. Djezzar, "Numerical simulation of the natural double-diffusive convection in an elliptical cylinder-Impact of the buoyancy force," International Communications in Heat and Mass Transfer, vol. 144, p. 106790, 2023.
[12] Q. R. Al-Amir et al., "Investigation of Natural Convection and Entropy Generation in a Porous Titled Z-Staggered Cavity Saturated by TiO2-Water Nanouid," International Journal of Thermofluids, p. 100395, 2023.
[13] A.-J. N. Khalifa, "Natural convective heat transfer coefficient–a review: I. Isolated vertical and horizontal surfaces," Energy conversion and management, vol. 42, no. 4, pp. 491-504, 2001.
[14] A.-J. N. Khalifa, "Natural convective heat transfer coefficient–a review: II. Surfaces in two-and three-dimensional enclosures," Energy conversion and management, vol. 42, no. 4, pp. 505-517, 2001.
[15] H. Karatas and T. Derbentli, "Natural convection in rectangular cavities with one active vertical wall," International Journal of Heat and Mass Transfer, vol. 105, pp. 305-315, 2017.
[16] H. Karatas and T. Derbentli, "Natural convection and radiation in rectangular cavities with one active vertical wall," International Journal of Thermal Sciences, vol. 123, pp. 129-139, 2018.
[17] M. Saglam, B. Sarper, and O. Aydin, "Natural convection in an enclosure with a discretely heated sidewall: Heatlines and flow visualization," Journal of Applied Fluid Mechanics, vol. 11, no. 1, pp. 271-284, 2018.
[18] O. Laguerre, S. Benamara, D. Remy, and D. Flick, "Experimental and numerical study of heat and moisture transfers by natural convection in a cavity filled with solid obstacles," International Journal of Heat and Mass Transfer, vol. 52, no. 25-26, pp. 5691-5700, 2009.
[19] U. Ghia, K. N. Ghia, and C. Shin, "High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method," Journal of computational physics, vol. 48, no. 3, pp. 387-411, 1982.