Journal of Information Systems Engineering and Management
2024, 9(3)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article

Numerical Simulation of Two-Dimensional Lid-Driven Cavity Flow

Gamit Jigar C[1], Parimal Chaudhari[2], Jiteshkumar Vijaybhai Mistry[3], Uday Dineshbhai Rathod[3]
[1]*Department of Mechanical Engineering, Government Engineering College, Modasa, Gujarat, India.*
[2]*Department of Mechanical Engineering, Shri K. J. Polytechnic, Bharuch, Gujarat, India.*
[3]*Dr. S. S. S. Ghandhy College of Engineering and Technology Surat, Gujarat, India.*

ARTICLE INFO

Received: 10 May 2024

Accepted: 20 July 2024

ABSTRACT

The present work explores the behavior of two-dimensional, incompressible laminar flow within a square cavity driven by a moving lid, using ANSYS Fluent as the simulation tool. To examine how the flow develops within the laminar regime, simulations were carried out at Reynolds numbers of 100, 400, and 1000. A structured quadrilateral mesh was employed, and the flow was solved using a coupled pressure–velocity formulation with third-order accurate spatial discretization. The results of velocity profiles and stream function were compared with the data provided by Ghia et al. [1]. Overall, the simulations captured both primary and secondary flow structures with high accuracy, reinforcing the validity of the numerical approach and laying the foundation for future investigations involving more complex geometries or transitional flow regimes.

Keywords: Lid-Driven Cavity Flow, Laminar CFD Simulation, Stream Function.

INTRODUCTION

Computational Fluid Dynamics (CFD) has become a foundational tool for studying fluid flow, offering a powerful alternative to both experimental setups and theoretical analyses. What makes CFD particularly valuable is its ability to handle intricate geometries, simulate harsh or untestable physical conditions, and provide rich visual insights into flow behavior. Its role has only grown with the growth of high-performance computing, making it an essential part of modern engineering workflows from aerospace design to automotive aerodynamics and thermal system optimization.

One of the most recognized test cases in CFD is the lid-driven cavity problem. Despite its simple geometry, a square cavity with a single moving lid and three stationary walls; it exhibits surprisingly complex internal dynamics. As the lid moves, it generates a primary vortex, along with smaller recirculation zones and boundary-layer separations, challenging even advanced numerical schemes. The work of Ghia et al. [1] remains the standard for benchmarking such flows, and has been extended by others, such as Sahin & Owens [2] and Botella & Peyret [3], who employed refined approaches like non-uniform meshing and spectral methods to capture finer flow details.

Beyond its role as a validation benchmark, the cavity flow scenario also mirrors several real-world systems. For instance, shear-driven flows in lubrication devices, recirculation patterns in compact heat exchangers, and pollutant transport in street canyons all resemble this configuration.

In this study, we simulate two-dimensional, incompressible, laminar flow in a lid-driven cavity using ANSYS Fluent. Simulations are performed at Reynolds numbers of 100, 400, and 1000 to explore how flow characteristics evolve within the laminar regime. The Finite Volume Method is used to solve the Navier–Stokes equations, and the analysis focuses on velocity distributions, stream function contours, and vorticity fields. Results are validated against the benchmark data from Ghia et al. [1] to assess the accuracy and reliability of the numerical approach.

MATHEMATICAL FORMULATION AND PROBLEM SETUP

Accurate modelling of lid-driven cavity flow begins with a sound mathematical foundation and carefully applied boundary conditions. For this study, the fluid behavior is governed by the incompressible Navier–Stokes equations, which, together with the continuity equation, describe how momentum and mass are conserved throughout the domain.

Journal of Information Systems Engineering and Management
2024, 9(3)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article

These equations are discretized using a finite volume approach and is solved numerically to capture the internal flow patterns with both physical accuracy and computational stability.

The flow takes place inside a square cavity, where each side measures one meter in length. A Cartesian coordinate system is used to define the domain, with the origin positioned at the bottom-left corner. The x-axis runs horizontally along the base of the cavity, while the y-axis extends vertically along the left wall. This setup establishes a clear spatial reference for analyzing the development of flow structures within the cavity.

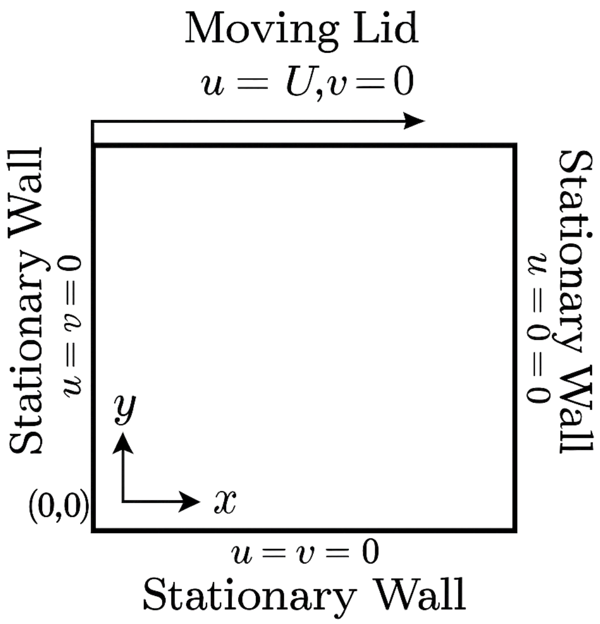


Figure 1: Schematic Diagram of the Lid-Driven Cavity

The accuracy of CFD simulations hinges on how well boundary conditions are defined. This is especially true for problems like the lid-driven cavity. In this setup, the cavity is square, and all four walls enforce the no-slip condition. While the bottom, left, and right walls remain fixed, the top wall commonly referred to as the "lid" moves horizontally to the right at a constant velocity. This movement introduces shear at the upper boundary, setting the entire fluid domain into motion and generating complex patterns of circulation and vortex formation within the cavity. The motion of the lid generates shear at the top boundary, which initiates fluid motion within the cavity, resulting in complex recirculation zones and vortex structures. The Reynolds number, a key parameter in determining flow dynamics, is varied here by modifying the horizontal velocity of the moving lid. Simulations are carried out at Reynolds numbers of 100, 400, and 1000, covering a range of laminar flow regimes. This classical problem is widely regarded as a benchmark in CFD research due to its simplicity in geometry and complexity in internal flow behavior.

The motion of fluid within the cavity is described by the two-dimensional, incompressible Navier–Stokes equations, which account for the conservation of both mass and momentum in Newtonian fluids. To ensure incompressibility, the continuity equation is applied, and it takes the following form:

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

The momentum conservation equations in the x- and y-directions are:

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)$$

Journal of Information Systems Engineering and Management
2024, 9(3)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}\right) = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right)$$

In this study, the two-dimensional lid-driven cavity flow was simulated using ANSYS Fluent, a widely used commercial CFD tool that applies the Finite Volume Method (FVM). Fluent is particularly effective for modelling incompressible, laminar flows thanks to its reliable solvers, robust pressure–velocity coupling algorithms, and powerful visualization features such as streamlines and stream function.

The simulation process followed a regular workflow:

1. Geometry Definition: A square cavity domain was defined as the computational geometry.
2. Meshing: A structured mesh was generated across the domain to ensure orthogonality and better control over element quality.
3. Boundary Conditions and Fluid Properties: The fluid was assumed to be Newtonian and incompressible. No-slip boundary conditions were applied to the bottom, left, and right walls, while the top wall (lid) moved horizontally at a set velocity. This velocity was adjusted to simulate Reynolds numbers of 100, 400, and 1000.
4. Solving: The discretized form of the governing equations was solved iteratively until residuals fell below defined convergence thresholds.
5. Post-Processing: Simulation results were analyzed using streamline plots, velocity field visualizations, and vortex identification to examine flow behavior at each Reynolds number.

ANSYS Fluent has been consistently used in previous studies to model lid-driven cavity problems, and its reliability in reproducing benchmark solutions is well-documented [6].

To achieve accurate and stable results, solver settings and numerical schemes were selected with care. A steady-state, pressure-based solver was employed—well-suited for incompressible laminar flows. Pressure–velocity coupling was handled using the coupled algorithm, which improves convergence speed and stability. For spatial discretization, third-order upwind schemes were applied to minimize numerical diffusion and resolve features like sharp shear layers and secondary vortices. The governing equations were integrated over control volumes using the Finite Volume Method, ensuring mass and momentum conservation throughout the domain and offering a robust framework for simulation.

MESH GENERATION AND GRID INDEPENDENCE STUDY

Mesh quality and resolution play a critical role in determining the accuracy and numerical stability of CFD simulations. For this study, a structured quadrilateral mesh was created using ANSYS Meshing, which is particularly well-suited for the square cavity geometry due to its orthogonal layout and precise control over cell aspect ratios.

To verify that the numerical results are not sensitive to mesh density, a grid independence study was performed. Meshes with progressively smaller element sizes—ranging from 0.08 to 0.01 were tested by tracking the average velocity within the cavity as a key flow parameter. The results indicated that when the mesh resolution reached 100×100 cells, the variation in average velocity fell below 0.4%, suggesting that further refinement yielded minimal accuracy gains. This confirmed that the chosen mesh offers a suitable compromise between computational efficiency and solution fidelity.

The mesh refinement study follows established practices in benchmark simulations of lid-driven cavity flow and ensures confidence in the numerical predictions derived from the selected mesh configuration.

Table 1: Grid Independence Study

No.	Mesh Element size	Computational Elements	Average Velocity
1	0.08	169	0.000265417

Journal of Information Systems Engineering and Management
2024, 9(3)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article

2	0.04	625	0.000275655
3	0.02	2500	0.000281214
4	0.01	10000	0.000282294

RESULTS AND DISCUSSION

The numerical simulations of the two-dimensional lid-driven cavity flow were conducted for three distinct Reynolds numbers: 100, 400, and 1000. The results, obtained from simulation, are presented and discussed in terms of velocity profiles, streamline contours, and vorticity distributions, highlighting the evolving flow characteristics with increasing Reynolds number. These findings were compared against established results by Ghia et al. [1].

The velocity field within the square cavity is predominantly driven by the motion of the top lid, which induces internal recirculation. As shown in Figure 2, the horizontal velocity profiles extracted at the vertical midplane of the cavity display a characteristic parabolic-like shape. The velocity magnitude peaks near the top lid due to the imposed motion and diminishes toward the bottom wall, where it approaches zero owing to the no-slip boundary condition.

- At Reynolds number Re = 100, the velocity contours in Figure 2a demonstrate a smooth and symmetric flow structure. The maximum velocity is concentrated in a thin layer adjacent to the moving lid, with gradual decay into the cavity. Flow remains gentle, dominated by viscous forces, and the gradients across the cavity are mild.
- At Re = 400, as seen in Figure 2b, velocity gradients become steeper near the lid and vertical sidewalls. The primary recirculation zone expands laterally, and higher velocities persist deeper into the cavity. Flow separation begins to emerge near the upper corners, and signs of shear intensification appear along the side boundaries.
- At Re = 1000, the velocity contours in Figure 2c, reveal a highly energized internal flow. The core vortex becomes more compact and shifts slightly upward, while thin boundary layers form near the walls. High-velocity streaks are directed toward the lower sidewalls, and distinct low-velocity zones emerge, correlating with secondary vortex development.

The stream function contours shown in Figure 3 illustrate the internal circulation patterns and the vortex structure induced by the lid motion.

- Figure 3a (Re = 100) displays a single dominant clockwise vortex centered slightly above the geometric center. The contours are relatively smooth and circular, with no evidence of secondary flow structures.
- Figure 3b (Re = 400) shows increased vortex strength and a slight downward shift of the vortex center. The streamlines are more densely packed near the top wall and corners, indicating stronger rotational motion.
- In Figure 3c (Re = 1000), the stream function contours illustrate a significantly intensified flow field. The primary vortex is clearly defined and centered toward the middle of the cavity. The streamlines near the corners show distortions suggestive of fully developed secondary vortices, a sign of stronger inertial forces shaping the internal flow topology.

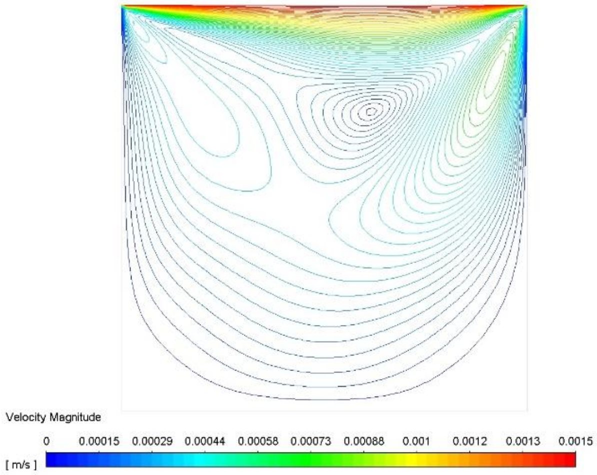
The collective analysis of velocity and stream function reveals the growing complexity of lid-driven cavity flow as Reynolds number increases.

- At Re = 100, viscous forces dominate, leading to a stable, symmetric recirculation with a single vortex and smooth velocity gradients. Vorticity is diffused and mild.

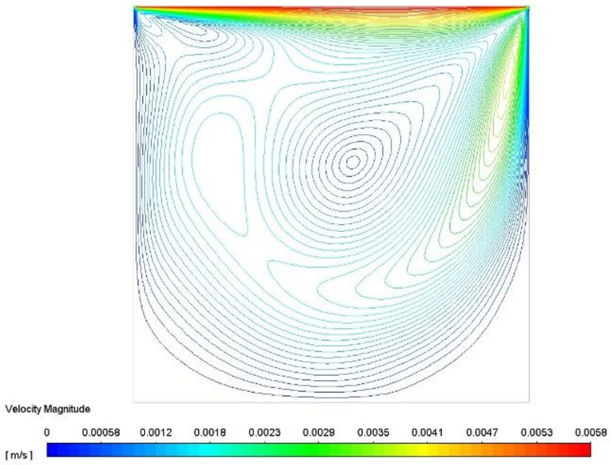
Journal of Information Systems Engineering and Management
2024, 9(3)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article

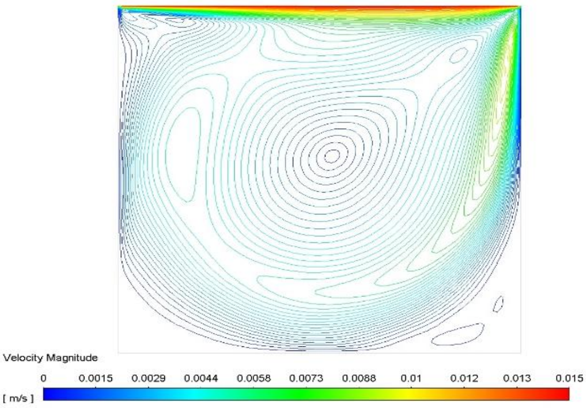
- At Re = 400, inertial effects become more influential, producing sharper velocity transitions, greater streamline curvature, and the initiation of secondary flow regions. Local shear increases near boundaries and corners.
- At Re = 1000, the flow transitions into a highly sheared, inertia-dominated regime still within laminar bounds led to thin boundary layers, sharp vorticity gradients, and vortical structure develop. These features are precursors to unsteady, and potentially turbulent flows expected at higher Reynolds numbers.



a.



b.

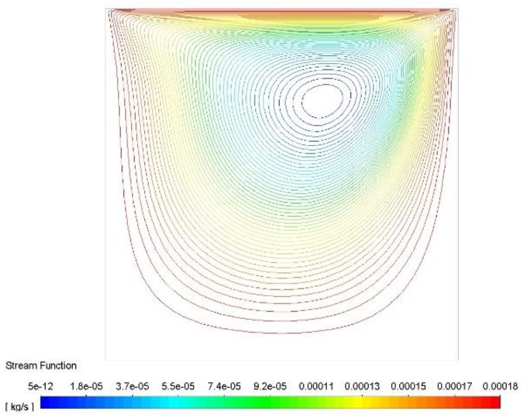


c.

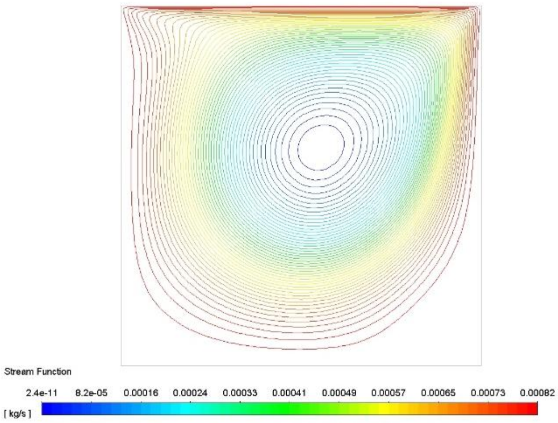
Figure 2: Comparison of Velocity Contours at Re=100, 400, 1000 respectively

Journal of Information Systems Engineering and Management
2024, 9(3)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

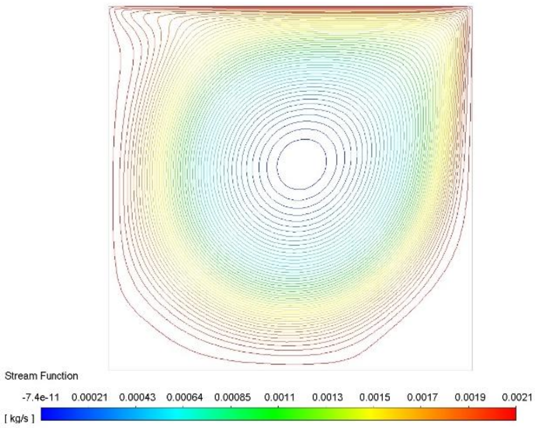
Research Article



a.



b.



c.

Figure 3: Comparison of Stream Function at Re=100, 400, 1000 respectively

CONCLUSION

In this study, a numerical investigation of two-dimensional, incompressible, laminar lid-driven cavity flow was conducted using ANSYS Fluent. Simulations were carried out for Reynolds numbers 100, 400, and 1000 to analyze flow behavior under varying inertial conditions. The governing Navier–Stokes and continuity equations were solved using the Finite Volume Method with a structured quadrilateral mesh.

Key flow features including velocity contours and stream functions were examined in detail. As the Reynolds number increased, the flow exhibited more pronounced vortical structures and increased asymmetry, highlighting the growing influence of inertia over viscosity. The results showed good agreement with benchmark data from Ghia et al.[1], validating the accuracy of the simulation methodology.

The study confirms that ANSYS Fluent provides reliable predictions for laminar cavity flow and serves as an effective tool for benchmarking numerical solvers. The outcomes of this work offer a solid foundation for extending the analysis to more complex scenarios, such as transient behavior, turbulent regimes, or non-Newtonian fluid effects.

REFERENCES

[1]. U. Ghia, K. N. Ghia, and C. T. Shin (1982), High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method, Journal of Computational Physics, 48(3), 387-411.

Journal of Information Systems Engineering and Management
2024, 9(3)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article

[2]. Sahin, M., & Owens, R. G. (2003), A novel fully implicit finite volume method applied to the lid-driven cavity problem — Part I: High Reynolds number flow calculations. International Journal for Numerical Methods in Fluids, 43, 57–68.
[3]. Botella, O., & Peyret, R. (1998), Benchmark spectral results on the lid-driven cavity flow. Computers & Fluids, 27(4), 421–433.
[4] Soulhac, L., et al. (2020), A review from theoretical, experimental and numerical perspectives. Renewable and Sustainable Energy Reviews, 133, 110295.
[5] Alves, M. A., et al. (2016), Lid-driven cavity flow of viscoelastic liquids. Journal of Non-Newtonian Fluid Mechanics, 234, 15-30.
[6] Markovic, J., Lukic, N., Ilić, J. D., Nikolovski, B., Sovilj, M. N., & Sijacki, I. M. (2012), Using the ANSYS FLUENT for simulation of two-sided lid-driven flow in a staggered cavity. Acta Periodica Technologica, 43, 169–178.