



Case Study

CFD Investigations of Lid Driven Cavity Flow using Ansys Fluent Software

Developed and curated by the Ansys Academic Development Team

Ravindra A. Shirsath

education@ansys.com

Ansys Software Used

This resource uses Ansys Fluent®, a fluid simulation software.

Summary

A lid driven cavity is a well-studied fluid dynamics problem involving a square or rectangular enclosure with a moving lid (top wall) that induces a flow within the cavity. It's a benchmark problem used to test and validate numerical methods for solving Navier-Stokes equations, especially for incompressible flows. The flow pattern within the cavity, characterized by a primary vortex and potentially secondary vortices at higher Reynolds numbers, is highly dependent on the Reynolds number, which represents the ratio of inertial to viscous forces. The present work explores the behavior of flow under varying Reynolds number over a wide range, highlighting the primary and secondary vortex regions with the help of flow visualization using streamlines and velocity contours along with velocity components and velocity vector plots for each case.

Table of Contents

1. Introduction.....	3
2. Problem Statement	3
3. Geometry and Mesh.....	3
4. Solution Methodology.....	4
5. Results and Discussion	4
5.1 Low Reynolds Number Flow Re=100	5
5.2 Moderate Reynolds Number Flow Re=1000.....	6
5.3 High Reynolds Number Flow Re=5000	8
6. Conclusion and Further Steps.....	9
7. References.....	9

1. Introduction

The lid driven cavity problem has long been used as a test or validation case for new codes or new solution methods. The problem geometry is simple and two-dimensional, and the boundary conditions are also simple. The standard case is fluid contained in a square domain with Dirichlet boundary condition on all sides, with three stationary sides and one moving side with velocity tangent to the side. Owing to the simplicity of its setup the lid driven cavity has been investigated quite extensively. It has been employed as a numerical benchmark problem and as a test bed for studying particular physical effects.

Because of its geometric simplicity and flow diagnostic method, this phenomenon is now routinely used for numerical validation of computational fluid dynamics (CFD) investigations. This case study presents numerical investigations of fluid flow inside a lid driven square cavity with dimensions as $1\text{m} \times 1\text{m}$. The numerical simulations have been performed using the fluids simulation software Ansys Fluent. To obtain the true flow behavior, several flow characteristics such as streamline plots, components of velocity vector, contours of velocity components in the respective direction and the plots of these components along the centerline of the cavity at various Reynolds numbers in the range of, $100 \leq \text{Re} \leq 5000$ are investigated.

2. Problem Statement

The lid driven cavity is a benchmark problem in computational fluid dynamics, where a fluid filled square cavity has one wall, i.e. the lid moving at a constant velocity. Flow behavior inside the cavity is investigated computationally. The investigations are performed over three different Reynolds numbers ($\text{Re}=100, 1000$ and 5000), to explore and visualize different aspects of flow such as primary and secondary vortices and how that is influenced by Reynolds number. The distinct aspects of flow are shown through visualization using velocity contours, additionally streamline and velocity vector plots are also shown for enhanced understanding of the underlying physics.

3. Geometry and Mesh

The computational domain used in the present work is shown schematically in Fig. 1(a), to highlight the important features of the same. The height of the cavity is Y , ($Y=1\text{ m}$) and the width of the cavity is X , ($X=1\text{ m}$) thus resulting in the square cavity. The domain nomenclature is also as shown in Fig. 1(a) consisting of side walls on the either side and bottom wall at the lower end. To have clear differentiation, the top side is named as top lid that will be used subsequently to apply the moving wall boundary condition. The zoomed view of the mesh used for present work is shown in Fig. 1(b), resulting from ≈ 251005 number of nodes resulting in ≈ 250000 number of elements that are mostly quadrilateral elements due to the simplicity of the geometry. Fairly refined mesh is used throughout the domain using Face meshing with the element size of $2 \times 10^{-3}\text{ m}$ to ensure that the near wall phenomenon is captured correctly, and specific requirements of the selected turbulence models are met. Given that the elements are mostly quadrilaterals, the best possible mesh quality is obtained with average element quality ≈ 1 , aspect ratio ≈ 1 , orthogonal quality ≈ 1 and skewness ≈ 0 .

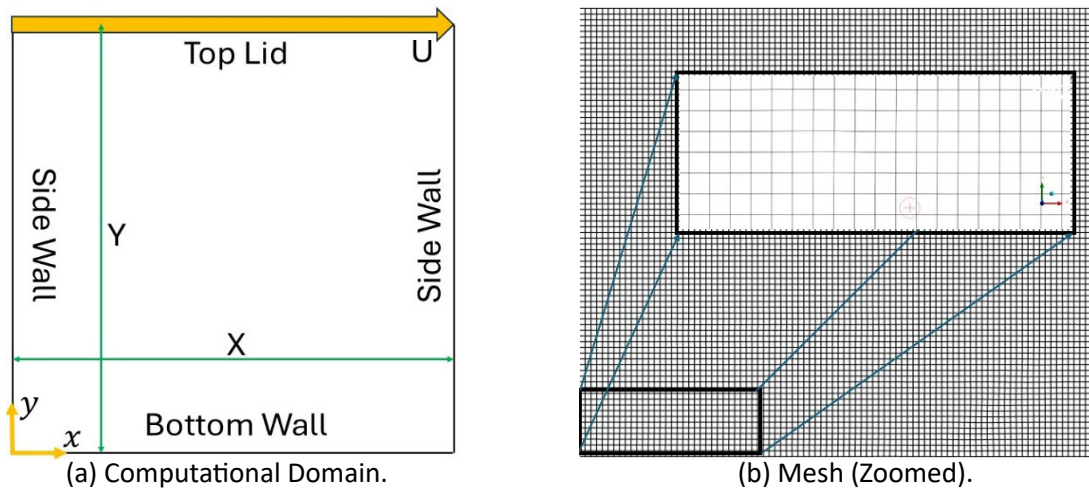


Figure1. Computational domain, geometry and mesh for Lid Driven Cavity (LDC).

4. Solution Methodology

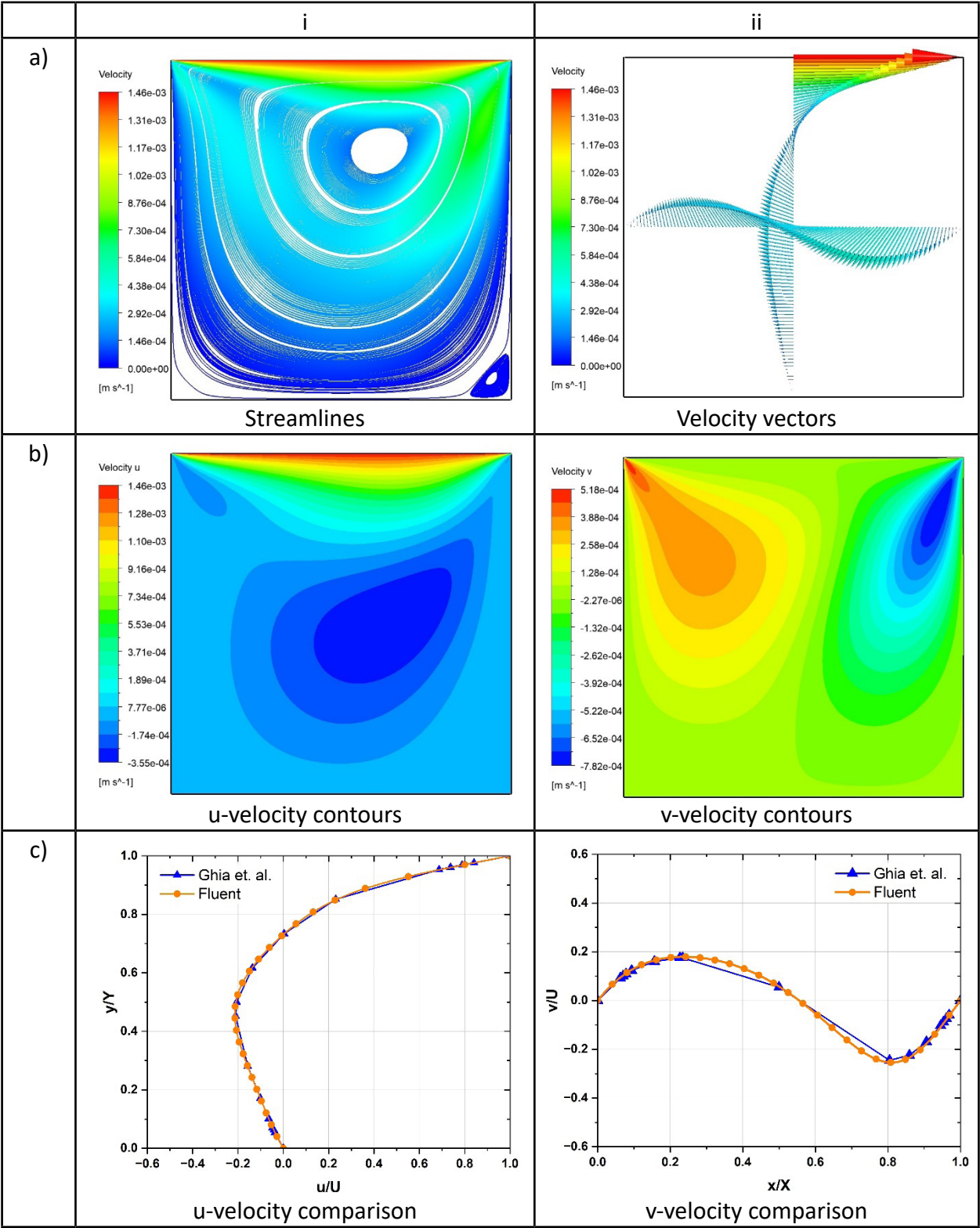
Using Ansys Fluent software, the following methodology is used. Numerical solver is set up as 2D space – planar, with pressure-based type, absolute velocity formulation and steady state simulation. The flow is assumed to be viscous turbulent hence different turbulence models are tried before finalizing $k-\omega$ SST with low Re corrections. The working fluid is chosen to be air with density, $\rho=1.225 \text{ kg/m}^3$ and the coefficient of dynamic viscosity, $\mu=1.7894 \times 10^{-5} \text{ kg/ms}$. Pressure-velocity coupling is dealt with coupled algorithm while least square cell-based method is employed for estimating the gradients and second order upwind scheme is used for discretizing the momentum equations. The lid is specified to be a moving wall with the required magnitude of velocity as necessary. The bottom and side walls are specified to be stationary walls with no-slip condition. Zero-gauge pressure is initialized throughout. The criterion for convergence is set to be 10^{-5} and the simulations are run for enough iterations.

5. Results and Discussion

This section explains the key results obtained by increasing the Reynolds number and validation of the results is done with benchmark results from Ghia et. al. [1]. The effect of gradually increasing Reynolds number on the vortex formation is shown with the help of streamline and velocity contours additionally supported by the corresponding velocity component contours and vector plots. The range of Reynolds numbers is divided into low $Re=100$, moderate $Re=1000$ and high $Re=5000$, as follows.

5.1 Low Reynolds Number Flow Re=100

Table 1: Streamlines, velocity components at Re=100



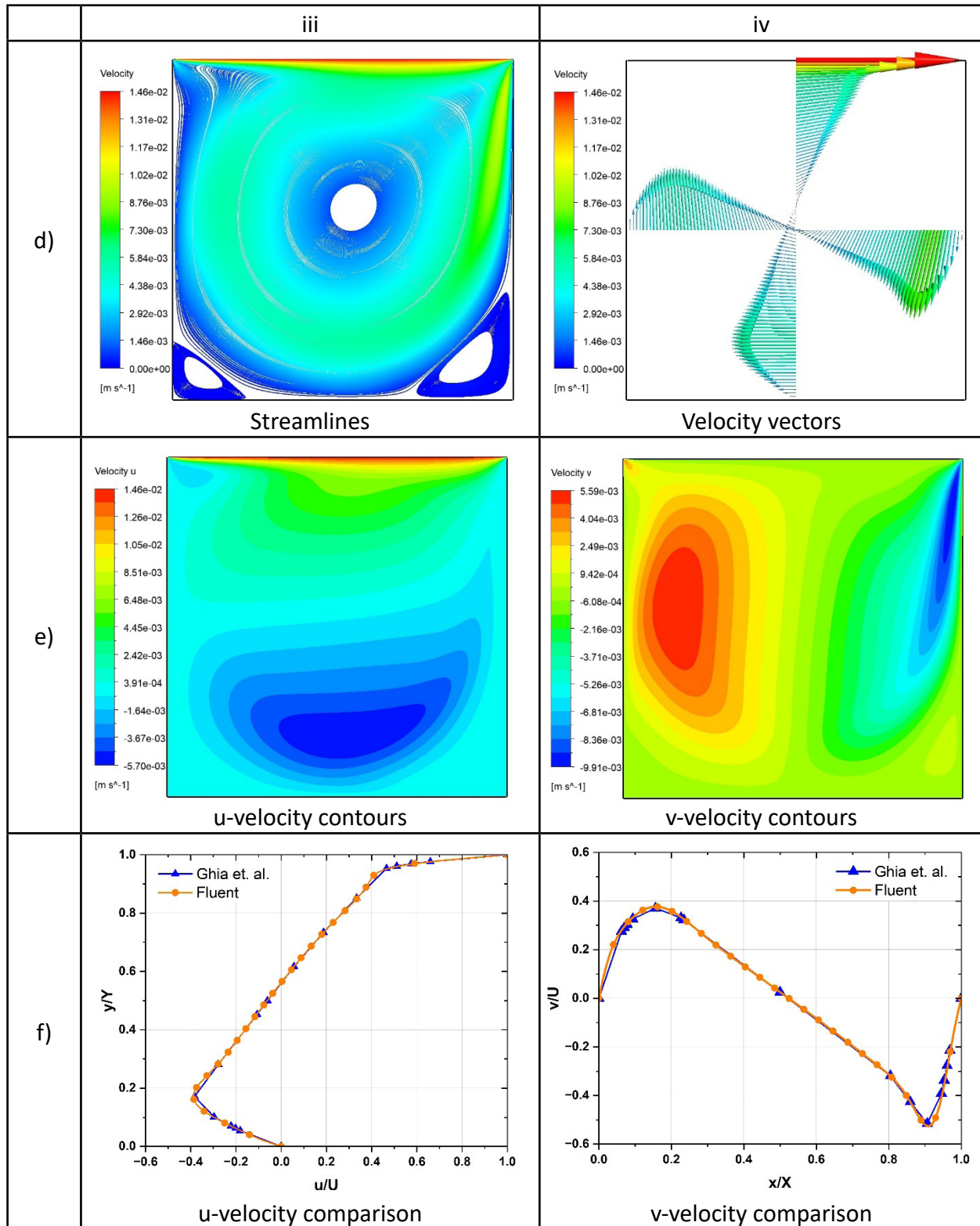
The simulation results from present study at Re=100 are shown in Table 1. The streamline plot observed from a(i) shows a large primary vortex that mostly dominates the flow with the onset of secondary

vortex at the bottom right corner. The velocity vectors from a(ii) clearly indicating forward flow confined to less than half portion of the cavity closer to the lid while rest is a backward flow created due to primary vortex effect, however in vertical direction the flow seems to be equally divided with right half having downward flow while the same is upward in the left half. The contour plot for u-velocity component as seen from b(i) further justifies that this velocity is maximum and equal to the lid velocity in its vicinity and gradually decreases away from it creating the minimum velocity region in the central part of the cavity. Similarly, the contours for v-velocity component indicate downward flow in the right half of the cavity through a negative velocity magnitude according to the present sign convention while that is upward in the left half thus resulting in positive velocity component as is visible from b(ii). The non-dimensional velocity components in respective directions across the centerline of the cavity are further extracted to quantify and make a qualitative comparison with the benchmark results. As observed from c(i) the u-velocity component closely matches the results from Ghia et. al.[1] clearly showing zero velocity on the lower wall and varying nearly parabolically and reaching that of the lid on the top. Additionally, the v-velocity component as observed from c(ii) is also in close agreement with the literature showing nearly sinusoidal variation across the centerline of the cavity. Overall, a smooth variation of respective velocity components is observed in both the directions along the centerlines of the cavity.

5.2 Moderate Reynolds Number Flow $Re=1000$

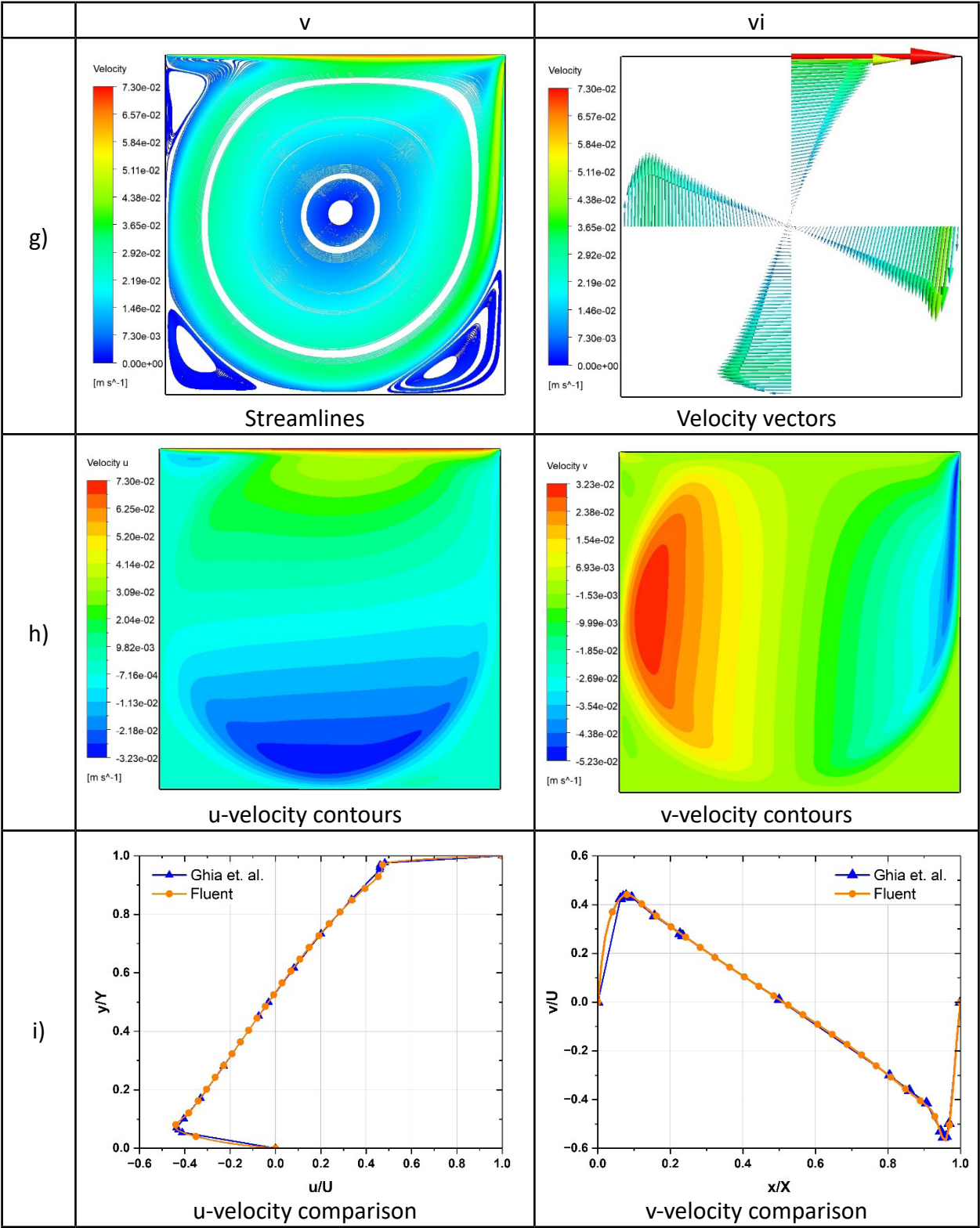
The effect of increasing the Reynolds number to $Re=1000$ on the respective results is shown in Table 2. The flow in this case as observed from d(iii) through the streamline plot continues to be dominated with the primary vortex at the center, however the formation of secondary vortices at both the corners at the bottom and also the onset of formation of additional vortex at top left corner differentiates it from the previous case clearly indicating the effect of increased Reynolds number achieved by increasing the lid velocity while everything else remains identical. More importantly, the core of primary vortex is observed to be shifting more towards the cavity center that is clearly evident from the velocity vectors plot as seen from d(iv). The u-velocity component clearly shows forward flow almost exactly in the upper half of the cavity while it is backwards in the lower half with no gradual variation. The v-velocity component also differentiates downward flow in the right half from the upward flow in the left half, but the variation is no more gradual or sinusoidal in this case that marks major deviation from the earlier observations. The horizontal split of the cavity consisting of forward flow in upper half and backward flow in the lower is further evident from u-velocity contours as seen from e(iii) while the similar in vertical direction is seen from e(iv). The right half of the cavity is seen to have a downward flow while that is upward on the left half. More importantly, the effect of increasing Reynolds number is better quantified by plotting respective velocity components at the centerlines of the cavity and comparing them with the literature as shown in f(iii) for u-velocity component and f(iv) for v-velocity component. With increasing lid velocity, more flow is seen to be dragged along the lid at top right corner but the presence of impermeable wall on the right is seen to push the flow downwards thereby causing a larger negative peak in the right half. The cumulative effect further with the impermeable bottom wall can only make the flow in the lower half to travel in opposite direction to the lid thus creating another negative peak for u-velocity component. The increased Reynolds number through increasing lid velocity is the sole reason for the corresponding peaks observed in both u and v-velocity components in addition to the corner vortices observed that are distinct from the previous observations. This Reynolds number however is reported to be a crucial in the literature as more complicated flow physics including Hopf bifurcation occurring around this range is reported by some researchers. However, these observations should suffice for the beginners' understanding.

Table 2: Streamlines, velocity components at Re=1000



5.3 High Reynolds Number Flow Re=5000

Table 3: Streamlines, velocity components at Re=5000



The obvious extension here is to see if this trend in overall results and observations continues with increasing the Reynolds number further to even a larger value and thus the next Reynolds number considered is $Re=5000$ and Table 3 shows all the corresponding results. In accordance to the observation made for previous cases, the flow remains dominated by the primary vortex originated very closely to the center of the cavity with clearly formed corner vortices at all the three corners except the top right as seen from $g(v)$. The increased size of corner vortices is clearly evident thereby shrinking the primary vortex marginally. The velocity vectors show similar trend to the previous cases just with slightly higher peaks and that are observed to move closer to the respective walls from $g(v)$. The u and v -velocity contours corroborate well with having forward/backward flow in horizontal halves and upward/downward flow in vertical halves of the cavity as seen from $h(v)$ and $h(vi)$ respectively. The increased negative peak in magnitude of u -velocity component and its shift closer to the bottom wall is observed from $i(v)$ while the similar ones for v -velocity component, negative peak in the right half and positive peak in the left half with their proximity to right and left walls respectively is seen from $i(vi)$.

Thus, the lid driven cavity flow simulated through CFD and the important flow characteristics made visible through streamline plots, velocity vector plots, contours of respective velocity components and quantitative comparison with existing literature is found to be in good agreement and thus provides a good starting point with invaluable data set for learning and validating the results obtained through various means.

6. Conclusion and Further Steps

Given the lid driven cavity flow to be the most fundamental problem for beginners to understand the concepts of fluid mechanics and CFD. The present investigations focus mostly on the flow visualization using contours of velocity components to give an idea about underlying physics involved. The effect of increasing Reynolds number incorporated by increasing the lid velocity is explored through these contours and are additionally supported by streamline and velocity vector plots. This approach is expected to enhance the learner's understanding about primary and secondary vortex formation along with the basic principles of fluid mechanics and CFD.

Given the simplicity of geometry, mesh and minimal simulation time, this case study is expected to be ideal to be demonstrated for beginners CFD class or partially as an assignment. Nonetheless, several suggestions for further investigations can be made, and there is certainly an opportunity for improvements. The first being the grid convergence study to investigate and optimize finer grids that will ensure all the important physical phenomenon is captured while keeping the computational time and efforts minimal. The efforts can be made to explore the influence of the aspect ratio of the cavity in case of non-square cavities by varying either the width or the height. The sizes of primary and secondary vortices can be quantified and their strong dependence on Reynolds number should be documented. In the end, more thorough flow field assessments should be carried out, which should include various designs of fins and baffles if turbulent mixing behavior is of prime interest.

7. References

- [1] Ghia U., Ghia K. N., and Shin C. T., High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method, *Journal of Computational Physics*, Vol – 48(3), pp. 387-411, 1982.
- [2] Bruneau C. H., and Saad M., The 2D lid driven cavity problem revisited, *Computers and Fluids*, Vol – 35(3), pp. 326-348, 2006.

© 2025 ANSYS, Inc. All rights reserved.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited. The full Academic Terms & Conditions can be found [using this link](#).

Document Information

This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

Ansyes Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

[Please click the link here](#) out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit www.ansys.com for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2025 ANSYS, Inc. All Rights Reserved.