

Lid-Driven Cavity Flow - using OpenFOAM

R Mythreyi

April 18, 2017

Contents

1	Introduction	1
2	Problem Definition	1
3	The Solution	2
4	Discussion	2
4.1	Case 1 - Cavity	2
4.2	Case 2 - Clipped Cavity	4
5	Limitations	6
6	Conclusion	6

1 Introduction

The aim of this report is to visualise how the lid-driven flow of an incompressible fluid inside a cavity is affected by changing the velocity of the lid as well as the geometry of the cavity.

Lid-driven cavity problem is often used as the test case for any new codes or new solution methods. The problem has a simple geometry to start with, the boundary conditions are also simple. There exists a lot of literature to compare results with and laminar case has a steady solution.

2 Problem Definition

The definition of the problem is straightforward : There is a two-dimensional domain (the standard case is a square), well represented by the rectangular co-ordinate system, inside which the cavity exists containing a fluid. On all the four boundaries of the planar domain, there is a Dirichlet boundary condition. Three sides are stationary and one side moves with a constant velocity. This is shown in figure 1. [3]

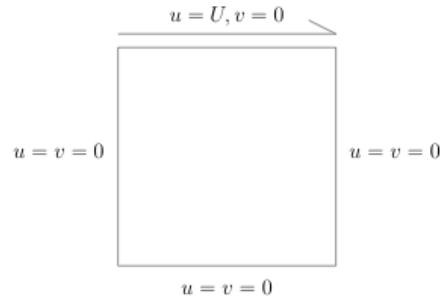


Figure 1: In this figure, u and v denote the x component and y component of velocity respectively.

While remaining a simple problem, a lot of variety is present. In the Reynolds number expression $Re = \frac{\rho U L_c}{\mu}$ for this case, ρ and μ are the density and viscosity of the fluid respectively, U is the velocity of the lid (the x component) and L_c is the length of the domain (along x axis). Thus, the variety is got by either keeping Re constant and varying either L_c or U , or by changing Re while keeping one of its parameters constant.

In this report,

1. The first case is the standard one. The domain is a square of length 0.1 m . The velocity of the lid is 1 ms^{-1} . The viscosity of the fluid is taken as 0.01 m^2s^{-1} . The Reynolds number Re is 10.
2. The second case considered is a "clipped" cavity, where a square of 0.04 m length is removed from the bottom right of the cavity. The velocity of the lid is set to 1 ms^{-1} . The other parameters are kept constant.

3 The Solution

The fluid flow in open lid driven cavity can be simulated by a set of mass and momentum conservation equations. The flow is assumed to be two-dimensional, laminar, incompressible and Newtonian. [1] In OpenFOAM, the solver used for all cases in the report is called `icoFoam`. The `icoFoam` solver solves two dimensional Navier-Stokes Equations using `PISO Foam` algorithm.[4] According to the description in the solver file, `icoFoam` is a "Transient solver for incompressible, laminar flow of Newtonian fluids."

The equations to be solved in this problem are :

1.

$$\frac{\partial}{\partial x_1}u_1 + \frac{\partial}{\partial x_2}u_2 = 0$$

which is what the continuity equation $\nabla_i u_i = 0$ reduces to for a two-dimensional case.

2.

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\nu \frac{\partial u_i}{\partial x_j})$$

for i and j running from 1 to 2 (since the domain is two dimensional). This is the Navier-Stokes equation without the force term. Here p is the Kinematic Pressure (pressure divided by density).

The solver uses the `PISO` algorithm to solve these equations. We get two fields - the Velocity field (ms^{-1}) and Kinematic Pressure field(m^2s^{-2}). The `PISO` algorithm working ideas are as follows : Since there are two complex coupling terms in the pressure-velocity systems - A u - u coupling in the non-linear advection term $u_j \frac{\partial u_i}{\partial x_j}$ and a linear pressure-velocity coupling in the equation as a whole, there is a necessity to keep the Courant number small so that one of these couplings dominate the other. Courant number (Co) is defined as

$$Co \equiv \frac{u\delta t}{\delta x}$$

where u is the characteristic velocity of the problem (in this case, it's the magnitude of the velocity in a cell), δt is the time-step of the numerical model and δx is the grid spacing of the numerical model. It can be defined in words as *a measure of how much information traverses (u) a computational grid cell (δx) in a given time-step (δt)*. When Courant number is small, the pressure-velocity coupling is much stronger than the other. This allows one to iterate over different u for the same p to first establish a velocity field and later the pressure distribution.

4 Discussion

4.1 Case 1 - Cavity

The mesh used for this case is shown in figure 2. The Courant number Co was evaluated at the regions of maximum velocity (near the moving lid, where $|U| = 1$) and set to be 1 to get time-step δt as 0.005s since $\delta x = 0.005m$ in the mesh in figure 2.

As seen in figure 3, the pressure is low at the left top corner and high at the right top corner. The sudden transfer of momentum from the moving lid to the fluid at the left end leads to the motion of the fluid away from the left top corner. This leads to a low pressure region. The fluid near the moving lid gets imparted with momentum due to the motion of the wall. From figure 4, we see that the x component of the velocity, influenced by the moving wall, is high in the top layers. Because of the presence of the stationary right wall, the x component of velocity must go to zero at the wall. This leads to a stagnation at the right corner, leading to high pressure in the region.

From the two-dimensional continuity equation, we see that if the x component of velocity is decreasing, the y component must increase. This is seen in figure 4. As we go further down the right boundary, we see that the

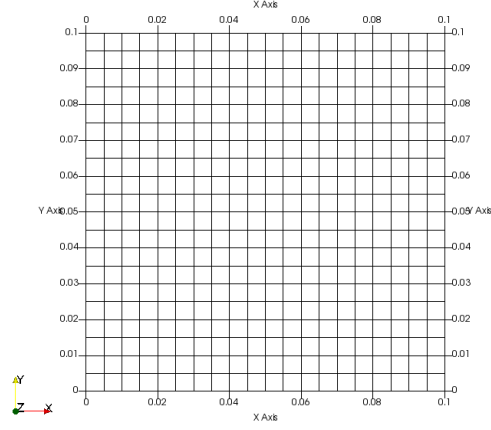


Figure 2: The mesh used for square geometry in case 1. It is a 20x20x1 cuboid and each of the cells within constitute the finite elemental control volumes.

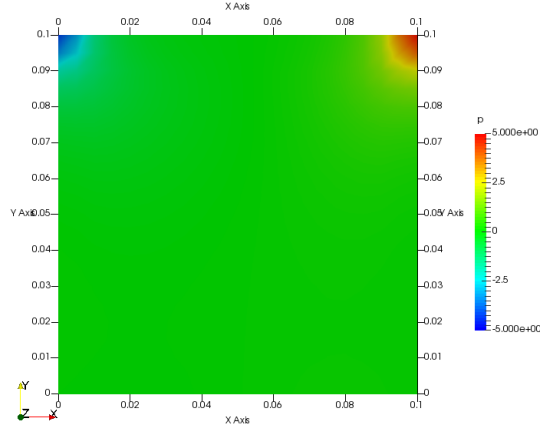


Figure 3: The pressure distribution in this case is given in this plot. There are regions of very high pressures towards the right top corner of the domain and regions of very low pressures in the left top corner. This build up pressure at the right corner can be attributed to the stagnation of the X component of velocity.

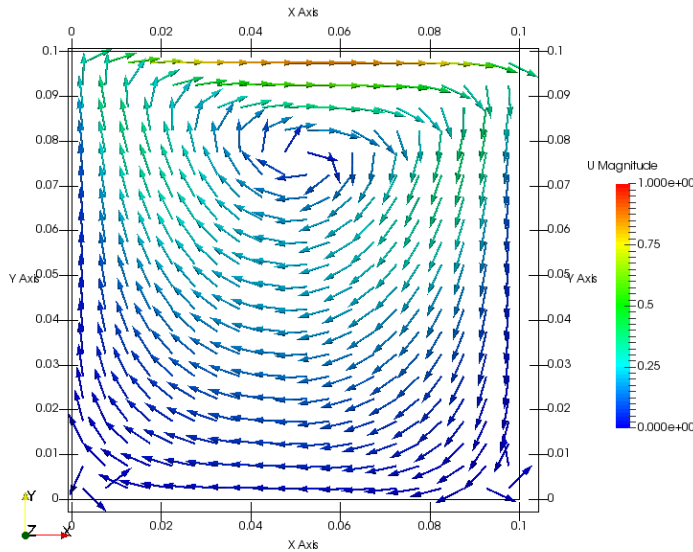


Figure 4: Velocity Vector Field Plot in case 1. The red colour at the top layers is due to high velocity. We can see that the colours become predominantly blue in the bottom half region. This is because of the transfer of momentum by the fluid at the top to the previously stationary fluid at the bottom.

y component is decreasing gradually, this is because its momentum is getting transferred to the layers below. As we reach the bottom right corner, we see that the velocity is not high enough for it to penetrate till the bottom wall. What we instead see is a dead zone and recirculation within it. The presence of recirculation is seen from the figure 4 right bottom corner velocity directions. A similar recirculation zone is seen at the bottom left corner.

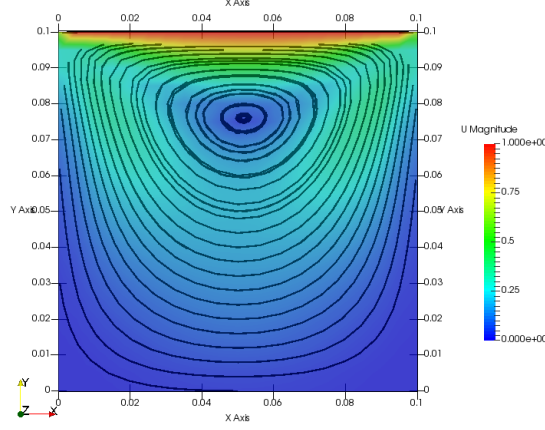


Figure 5: Streamlines in case 1. These streamlines are closed loops a little below the top layers, resembling a vortex formation. This is because of the viscosity of the fluid and absence of external forces. The streamlines indicate that there's more curvature at the bottom of the vortex than the top, this is due to the velocity differences while inertia (viscosity) remains the same.

From the streamlines plotted in figure 5, we see a vortex behavior at the centre. This is because of the viscosity of the fluid and the absence of external forces. We can also see dead zones (without the visualisation of the recirculation) at the bottom corners of the cavity. What we see are regions which aren't affected directly by the central vortex, which because of shear due to nearby regions moves in the opposite direction. These loops of fluid flow opposite to the mainstream are the recirculation loops.

4.2 Case 2 - Clipped Cavity

The mesh used for this case is shown in figure 6. The Courant number Co was evaluated, similar to case 1. The regions of maximum velocity are near the moving lid, where $|U| = 1$, and if Co is set to be 1, we get time-step δt as $0.005s$ since $\delta x = 0.005m$ in the mesh in figure 6.

To solve for this clipped case, the starting point is the final fields of case 1. This is because there are only

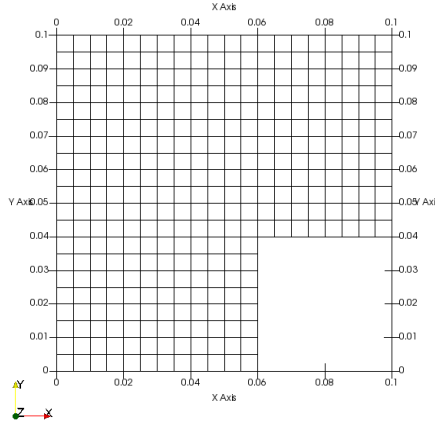


Figure 6: The mesh used for square geometry in case 1 has been clipped. The clip is a square of length $0.4m$. This is a combination of three cuboids - $12 \times 12 \times 1$ at left top, $12 \times 8 \times 1$ at right top, $8 \times 12 \times 1$ at left bottom. Each of the cells within constitute the finite elemental control volumes.

minor changes in geometry. This mapping of fields will save computational time and resources.[2]

As seen in figure 7, the pressure is mapped as such. The solved pressure distribution is given in figure 8. The characteristic length (the length along x axis) hasn't changed in this part of the domain. Thus, the effect of change in geometry is not felt.

Since the characteristic length has changed in the bottom region, there is a visible change in the velocity profile as seen in figure 10 compared to the mapped field in figure 9. Because the bottom layer is higher up

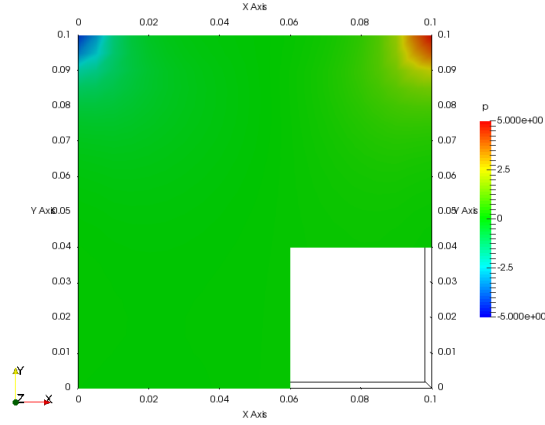


Figure 7: The pressure distribution given here is the one mapped from case 1, as it is, without solving for this case.

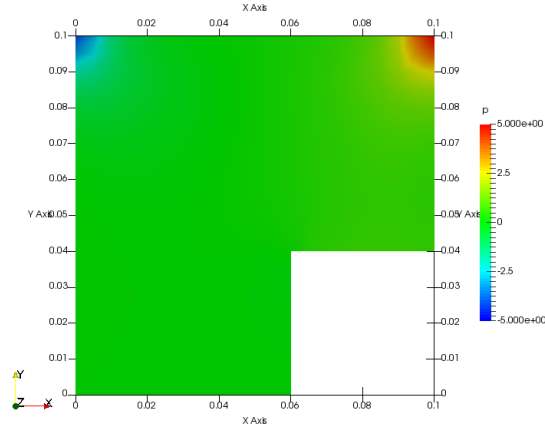


Figure 8: The pressure distribution given here is the one solved for this clipped case. On comparing with the previous case, we do not see much differences. This could be because there is no change in the characteristic length in this part of the domain.

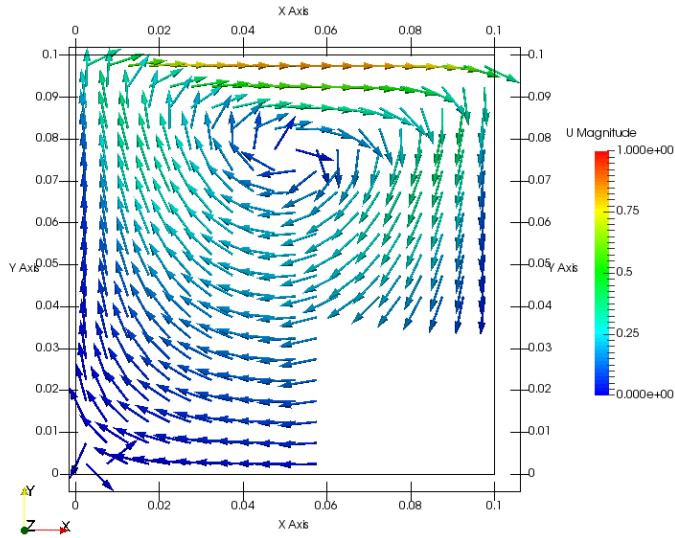


Figure 9: Velocity Vector Field Plot in case 1 has been mapped to the new changed domain. We see that near vertical clipped region, the x component of velocity is not zero at this stage (without solving for the case).

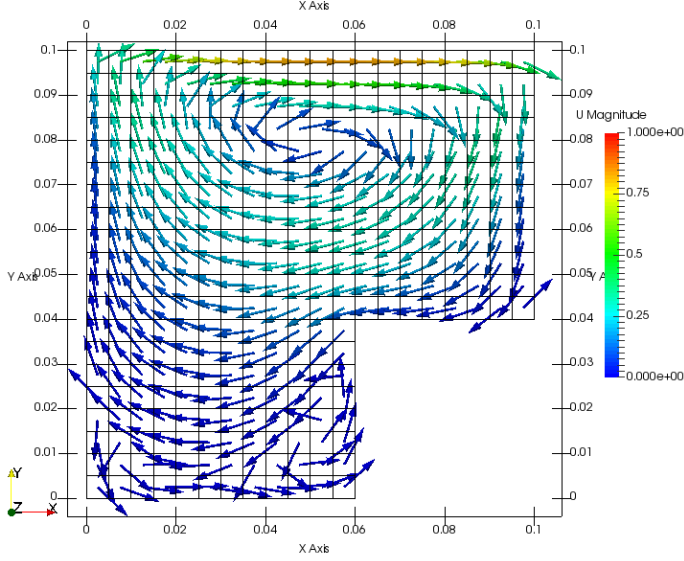


Figure 10: After solving for the case, we see a much larger recirculation zone. The profound changes in the velocity profile from before in this region can be treated to be induced by the change in the characteristic length itself.

in the clipped region, the x component of the velocity is high enough to not move along vertical clipped wall beyond the sharp point in the clip at (0.06,0.4). This leads to a larger dead zone which leads recirculation. The recirculation loops are evident from the streamlines plotted in figure 11.

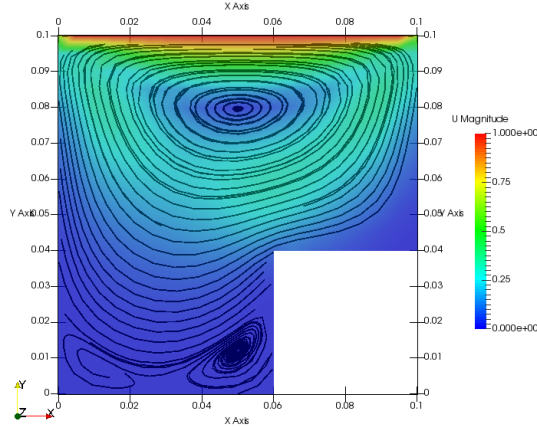


Figure 11: Streamlines in case 2. The presence of recirculation loops is much more evident in the plot at both the left and right bottom corners in the clipped region. The recirculation loops are much larger on the right compared to left because of the higher velocity in the right than the left.

From the streamlines plotted in figure 11, we see a vortex behavior at the centre as before in case 1. We see recirculation loops at the bottom corners. The loops are larger on the right corner because of higher velocity. The velocity is high enough to not allow momentum transfer to the lower layers near the boundary wall, thus the streamlines almost run diagonally down.

5 Limitations

There are certain limitations to the PISO algorithm and the finite element approach taken. One such limitation is the sensitivity of the algorithm to the mesh chosen for the problem. Also, difficulties may arise at the corner intersections of the fixed and moving walls in finite element approaches [3].

6 Conclusion

From the two cases taken, the dependence of the velocity and pressure fields for a simple case was discussed. The basics of the Lid-driven cavity problem was thereby explored and hints for further analyses can be obtained

from the limitations.

References

- [1] CFD Direct. *OpenFOAM User Guide - Lid-driven cavity flow*. URL: <https://cfd.direct/openfoam/user-guide/cavity/> (visited on 04/16/2017).
- [2] Francis X Giralgo. *Time Integrators*. URL: http://faculty.nps.edu/fxgiraldo/projects/nseam/nps/new_section4.pdf (visited on 04/18/2017).
- [3] CFD-Online Wiki. *Lid-driven cavity problem*. URL: https://www.cfd-online.com/Wiki/Lid-driven_cavity_problem (visited on 04/17/2017).
- [4] OpenFOAM Wiki. *IcoFoam*. URL: <https://openfoamwiki.net/index.php/IcoFoam> (visited on 04/17/2017).