



## LID- DRIVEN CAVITY FLOW

Manyam Srivallabh  
MM23B049

### Abstract

This report deals with fluid simulations based on a popular simulation problem called lid-driven cavity flow. The simulation involves a cavity full of fluid top of which is a lid moving at constant speed. The simulation involves a lot of interesting fluid phenomenon like vortices, laminar or turbulent flow with an unsteady nature and many fundamental concepts like Navier-Stokes equation, continuity in incompressible flow, planar flows etc based on the content of the transport phenomenon course.

**Keywords:** Openfoam, simulation, Reynold's number(Re)

### Introduction and Setup

The setup for lid-driven cavity flow involves a rectangular or square-shaped vessel filled with fluid to the brim. On top of the vessel is a uni-directional moving lid which is translating at a constant velocity parallel to the base of the vessel while making sure the lid is in contact with fluid along the entirety of the the open end of the vessel and also for the entire duration of the experiment. On the other hand, the other three faces(walls and base) on the vessel follow a no slip condition ,i.e, fluid particles in

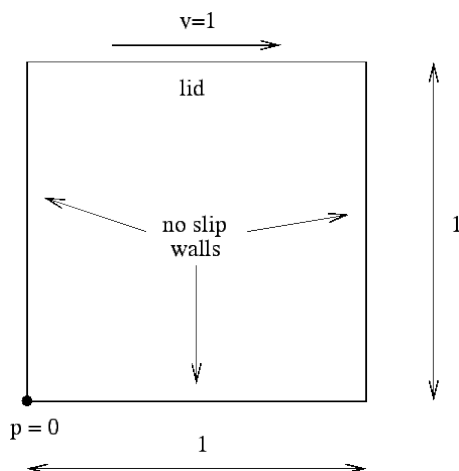


Figure 1: Schematic of an ideal setup for the basecase.

contact with these faces have no motion. The properties

of the fluid are such that the reynolds number is low enough for a laminar flow.

In a standard configuration, the fluid flow is driven by the moving lid and the boundary conditions are set at the walls. This is a classics example for a simulation involving laminar planar flow and incompressible fluids where there are no external forces that take effect other than the shear force at the lid.

### Why lid-driven cavity flow?

#### *Simplicity*

The Cavity flow is a well-defined case with clear and simple boundary conditions, which makes numerical setup easy and straightforward. The cavity's simple geometry also helps along with the need of only fundamental concepts from fluid mechanics.

#### *Rich fluid phenomenon*

The conditions in this setup are just simple and sufficient enough to find complex phenomenons like:

- *Vortex Formation* : Vortices form in this case because the translating lid creates shear stress that causes the fluid to rotate. The lid-driven cavity case always contains a primary vortex and a secondary vortex at higher Reynold's number regimes.
- *Non-uniform fluid-flow distribution* : The fluid moves at relatively high velocities near the lid and decreases as we go down. The pressure also varies with position.

#### *Relevance*

The case involves fluid in a confined environment which is quite common to find. The case involves simple concepts from the course content like:

- Continuity equation in incompressible fluids
- Computation using Navier-stokes equation
- Creeping flow
- Planar flow
- Reynold's number

### Objectives

- To gain experience in setting up, running, and post-processing CFD simulations using OpenFOAM.

- To simulate the 2D lid-driven cavity flow problem in OpenFOAM.
- To interpret streamline patterns through post-processing techniques.
- To understand the impact of Reynold's number on the formation of primary and secondary vortices.
- To observe and analyze velocity and pressure distributions within the cavity at different Reynold's number and cavity dimensions.

## Openfoam Execution Overview

### Folder Structure

```
lidDrivenCavity/  <-- (Case Folder)
├── 0/            <-- (Initial conditions)
│   ├── u        <-- (Velocity field)
│   └── p        <-- (Pressure field)
├── constant/    <-- (Physical properties and mesh)
│   ├── transportProperties <-- (Fluid properties like viscosity)
│   └── polyMesh/
│       └── blockMeshDict  <-- (Mesh definition)
├── system/      <-- (Simulation control settings)
│   ├── controlDict      <-- (Start/end times, deltaT, etc.)
│   ├── fvSchemes        <-- (Discretization schemes)
│   └── fvSolution        <-- (Solver settings)
```

### Pre-processing

- **Mesh Description** : First leap involves describing the domain involved in the case to the software. This done by visualizing the domain as a conglomerate of faces which can be described through their vertices in a particular fashion such as clockwise direction. These observations need to be feeded into 'system/blockMeshDict'.
- **Mesh Generation** : Once domain is described, next step involves how refined the mesh needs to be, i.e, at what scale of units of space do you want the calculations to be done to optimize desired accuracy? Ex: I used (20x20x1) for Re=100 and (135x135x1) for Re=4000.
- **Execution in code** : Once parameters are set as desired, run the command "blockMesh". For complex geometries, further use the "snappyHexMesh" utility to generate a fully-functional mesh. Personally, I didnt need to use snappyHexMesh because the geometry of a cavity is simple.

### Processing

- **Solver Selection** : a solver of choice like icoFoam, pisoFoam etc could be choosen. Since mine were simple cases, I used the utility 'foamRun' which automated this process.
- **Control Parameters**: Control parameters involves time properties to control duration of simulation,

frequency of carrying out simulations etc. This frequency is very important as simulations become more static like in low Re regimes, a smaller frequency is required to capture motion. For example, values such as 0.005 were perfect for  $Re > 1000$  but created problems due Courant number restrictions for Re less than that. Time duration is important since for  $Re < 2100$ , the domain reaches steady-state within seconds(2-10 sec), high time-durations are extra work for the computer.

- Execution in code: run 'foamRun' or the solver of choice for openfoam to compile results of simulation.

### Post-Processing

- **Visualization** : Simulation results can then be seen visually using ParaFoam(using "paraFoam" command). It helps see velocity and pressure distributions. I used slice tool to get a 2D section for further processing.
- **Streamlines** : The tool "StreamTracer" could be used to trace streamlines ,i.e, how fluid moves over time. This is important as this is needed to see primary and secondary vortices.
- **Velocity and pressure Profiles Along Lines** : The tool "PlotOverLine" could be trace velocities and pressures at different segments of a line. I did that at  $y=l/2$  line.

## Numerical Setup

Considering multiple cases based on different sets of parameters:

### Problem description

- Planar, In-compressible, Laminar lid-driven cavity flow
- Cavity-Dimensions: (0.1 X 0.1) or (0.2 X 0.1) or (0.1 X 0.2) m<sup>2</sup> for each reynold's number

### Boundary Conditions

- **Lid** :  $U(1,0,0)$  m/s,  $P(\text{zero gradient})$
- **Walls and Base** :  $U(0,0,0)$  m/s,  $P(\text{zero gradient})$

### Simulation Settings

- **Solver** : icoFoam or pisoFoam
- **Utility** : foamRun

### Deciding Parameters

- **Mesh Resolution** : Ranging from (20x20x1) to (135x135x1)
- **$\nu$**  (changes Reynolds number indirectly): (0.1, 0.025, 0.01, 0.005, 0.0025) m<sup>2</sup>/s  $\approx$  (100, 400, 1000, 2000, 4000) units
- **$\Delta T$**  : 0.005 for  $Re \geq 1000$  else 0.001
- **EndTime** : ranging from 2 to 15 s

## Results

Case 1 ( $Re=100$ ):

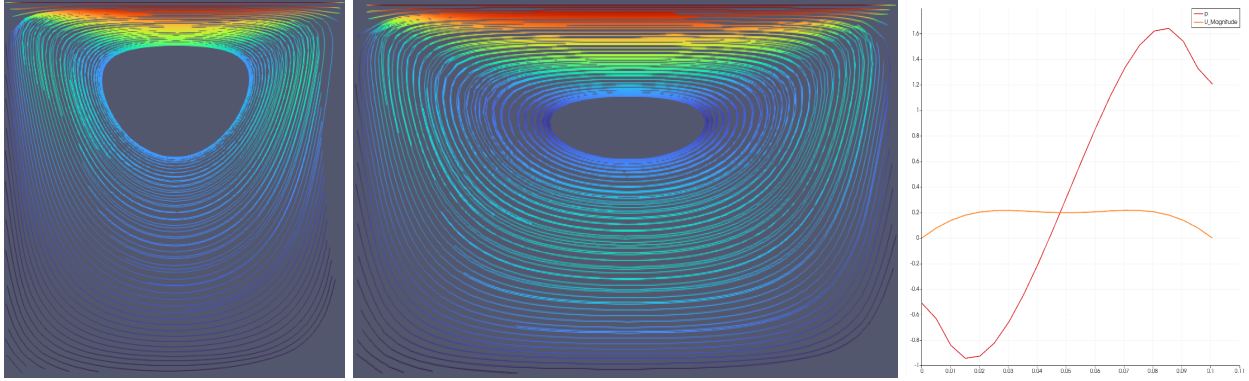


Figure 2: At Steady-state (a) Streamline of 1:1 cavity flow (b) Streamline of 2:1 cavity flow (c) Common plot of  $p$  and  $U$  vs distance along  $y=L/2$  line

- *Parameters Used* : Mesh(20x20x1); nu(0.1); deltaT(0.001); EndTime(2s)
- *stead-state timestamp* : instant
- viscous forces dominate leading to a very stable, laminar flow.
- Symmetric and centered vortex structure.

Case 2 ( $Re=400$ ):

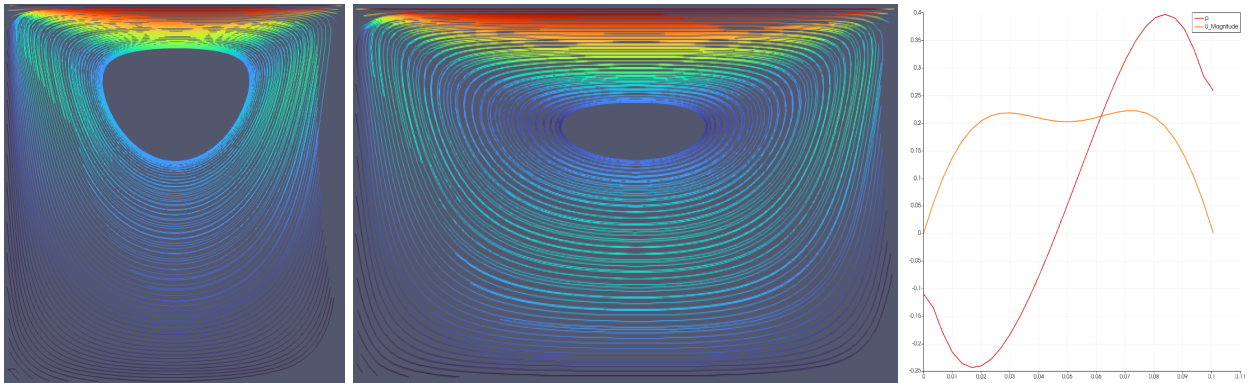


Figure 3: At Steady-state(or End-time) (a) Streamline of 1:1 cavity flow (b) Streamline of 2:1 cavity flow (c) Common plot of  $p$  and  $U$  vs distance along  $y=L/2$  line

- *Parameters Used* : Mesh(30x30x1); nu(0.025); deltaT(0.001); EndTime(2s)
- *stead-state timestamp* : instant
- viscous forces dominate leading to a very stable, laminar flow.
- vortex is losing its symmetric and centered nature.

Case 3 ( $Re=1000$ ):

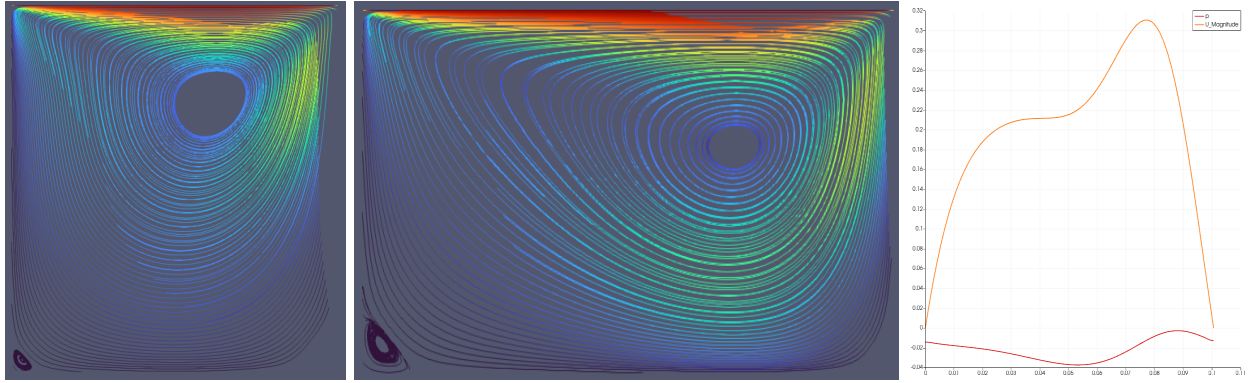


Figure 4: At Steady-state(or End-time) (a) Streamline of 1:1 cavity flow (b) Streamline of 2:1 cavity flow (c) Common plot of  $p$  and  $U$  vs distance along  $y=L/2$  line

- *Parameters Used* : Mesh(75x75x1); nu(0.01); deltaT(0.005); EndTime(2s)
- *stead-state timestamp* :  $\approx 5s$
- Transition between laminar and turbulent characteristics.
- Asymmetric and Uncentered vortex structure but Secondary vortices start to appear.

Case 4 ( $Re=2000$ ):

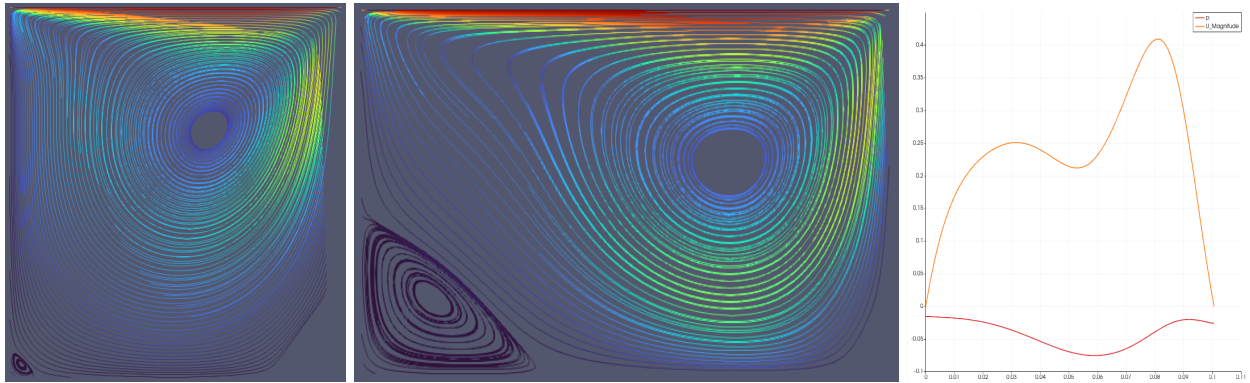


Figure 5: At Steady-state(or End-time) (a) Streamline of 1:1 cavity flow (b) Streamline of 2:1 cavity flow (c) Common plot of  $p$  and  $U$  vs distance along  $y=L/2$  line

- *Parameters Used* : Mesh(75x75x1); nu(0.005); deltaT(0.005); EndTime(2s)
- *stead-state timestamp* :  $\approx 7s$
- Early Turbulent regime
- Asymmetric and Uncentered vortex structure and Secondary vortices seem to get bigger.



Case 5 ( $Re=4000$ ):

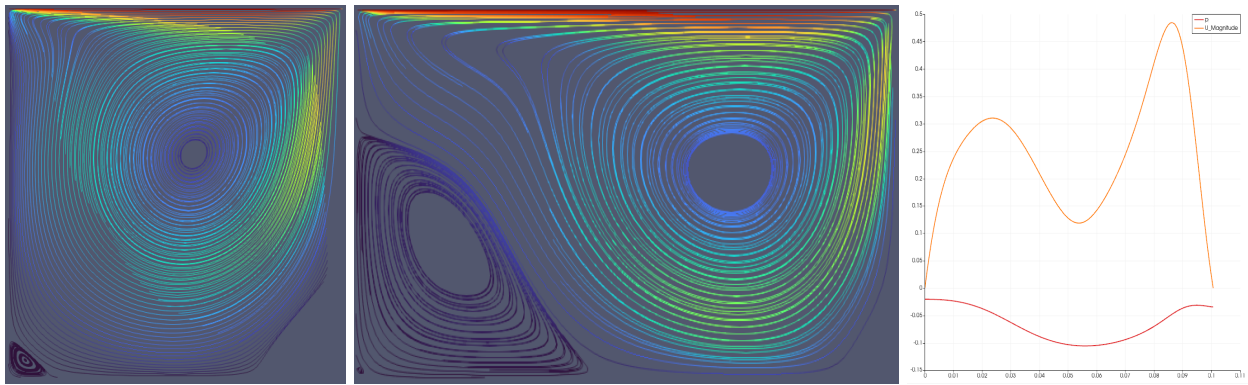
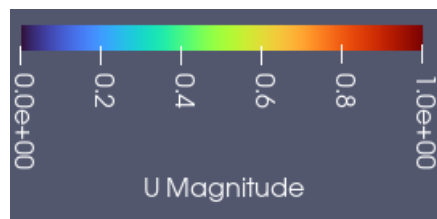


Figure 6: At Steady-state(or End-time) (a) Streamline of 1:1 cavity flow (b) Streamline of 2:1 cavity flow (c) Common plot of  $p$  and  $U$  vs distance along  $y=L/2$  line

- *Parameters Used* : Mesh(135x135x1); nu(0.0025); deltaT(0.005); EndTime(10s)
- *stead-state timestamp* : never
- Fully turbulent regime
- massive vortex structure in 2:1 cavity.



## Observations

- 2:1 cavity has a higher tendency to contain secondary vortices since shifting of primary vortex creates more space for the secondary one to form. Also, more kinetics energy is transferred from the boundary liquid to fluid internally.
- Velocity plot flattens from a rather 2 peak one as  $Re$  decreases. This is because as Reynold's number decreases, viscous forces dominate favouring smooth transistions.
- Pressure tends to drop to negative values as  $Re$  increases,since, Higher the rotational velocity lower the pressure in the region.
- The velocity of fluid in secondary vortices is near zero because very little kinetic energy seeps to the bottom region.

## Takeaways

- Learnt about the overall procedure to simulate a fluid case.
- Learnt about flow structures and how important are boundary conditions.
- Learnt about how reynolds number influences fluid behaviour.
- Learnt how can dimensions of a cavity influence fluid behaviour
- Learnt the importance of control parameters and how powerful they are in controlling simulations.

## References

1. OpenFoam Guide. Available from: <https://doc.cfd.direct/openfoam/user-guide-v12/>
2. MIT Resource. Available from: [https://web.mit.edu/calculix\\_v2.7/CalculiX/ccx\\_2.7/doc/ccx/node14.html](https://web.mit.edu/calculix_v2.7/CalculiX/ccx_2.7/doc/ccx/node14.html)

See GitHub Link -OPENFOAM execution