MAE 560

Final Project

Fall 2018

Solution of the Incompressible Navier-Stokes Equation on a general two dimensional domain of quadrilateral cells using the finite-volume method

Sujal Dave

MAE

Problem 1: 2-D Taylor Green Problem

1.1 - Introduction

The problem describes the decay in time of two stationary counter-rotating vortices in a periodic domain. The problem has an exact closed form solution of the incompressible Navier–Stokes equations in the Cartesian coordinates.

The exact solution in 2-D is given by:

$$u(x, y) = -\cos(x)\sin(y)\, e^{-2\nu t}$$

$$v(x, y) = \sin(x)\cos(y)\, e^{-2\nu t}$$

$$p(x, y) = p_0 - \frac{1}{4}(cos(2x) + \cos(2y))\, e^{-4\nu t}$$

$$k(t) = \frac{1}{4} e^{-4\nu t}$$

1.2 Method of Solution

The Taylor Green problem is the most rudimentary problem in solving the Incompressible Navier-Stokes equation in a domain. The following are the steps taken in order to define the problem computationally and solve it with a good accuracy.

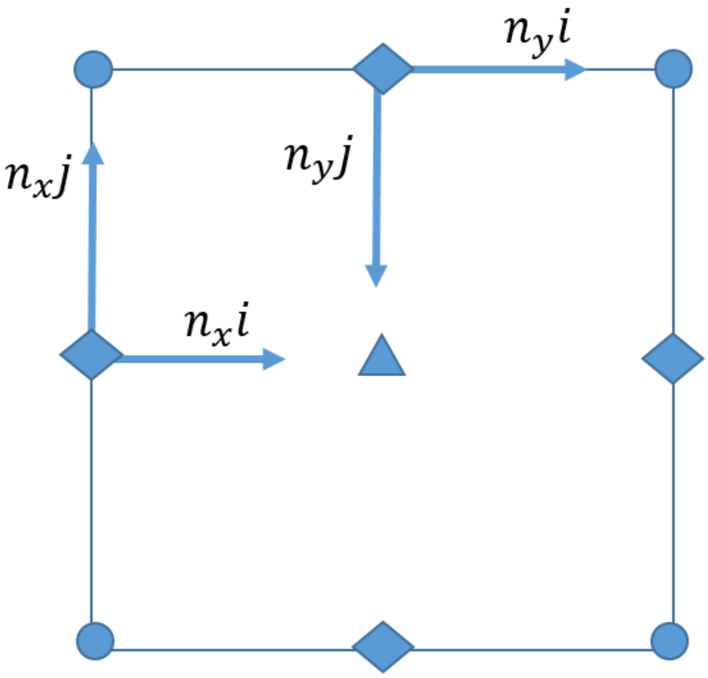
1.2.1 Domain and Mesh definition

A periodic domain is used to tap the transient counter rotating vortices.

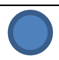
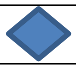
Length of domain in x-direction	2π
Length of domain in y-direction	2π
Number of cells in x-direction	100
Number of cells in y-direction	100

The finite volume method is used, and hence the properties are assumed to be concentrated at the cell centres. Definition of normal, surface areas are also made in the similar way along the edges. A structured square mesh is used in the domain.

Nomenclature of the cell parameters in a typical cell of the domain is as shown:



The following table represents the values which are computed as an approximation at the various points.

	Nodes
	Normals and Surface Areas
	Cell centres, Velocities, Volume, Pressures, Non-linear terms, Viscous terms

1.2.2 Boundary conditions

As the domain is periodic, the ghost cells on each side need to be initialized accordingly. Considering two additional rows and columns for the ghost cell, the boundary condition applied to the ghost cell is as shown;

```
u(1,:)=u(nx+1,:);
u(nx+2,:)=u(2,:);
u(:,1)=u(:,ny+1);
u(:,ny+2)=u(:,2);
```

Here, the ghost cells exist as the 1st and (nx+1) row and 1st and (ny+1) column.
The domain exists from [2 to nx+1] and [2 to ny+1] cells.

1.2.3 Initialization

Since the exact solution is provided, we incorporate $t = 0$ and initialize the domain with the counter rotating vortices.

Initial Condition:

$$u(x,y) = -\cos(x)\sin(y)$$

$$v(x,y) = \sin(x)\cos(y)$$

The initialization of pressure is not really required because we can guess the pressure points to be zero for the SOR method and it would converge to the solution accordingly.

1.2.4 Mahesh Algorithm

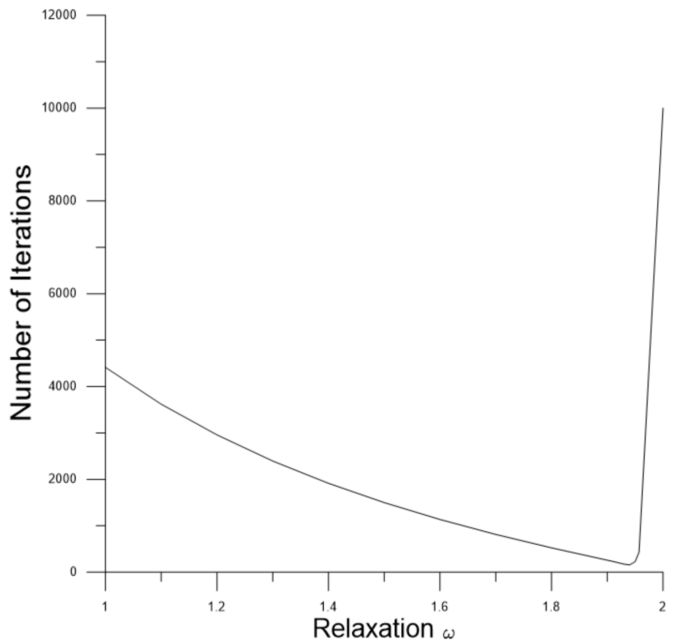
The finite volume version of the fractional step method from the paper by Mahesh, Constantinescu and Moin (2014) is used. The method uses non-staggered, energy conserving formulation. The following are the brief steps used in incorporating the algorithm:

Step 1: Ignore the pressure and use Adams-Bashforth discretization to get the predicted velocity fields
Step 2: Find the pressure that gives us the divergence-free velocity field.
Step 3: Put the pressure derivatives that we ignored in Step 1, back in.

1.2.5 Input parameters

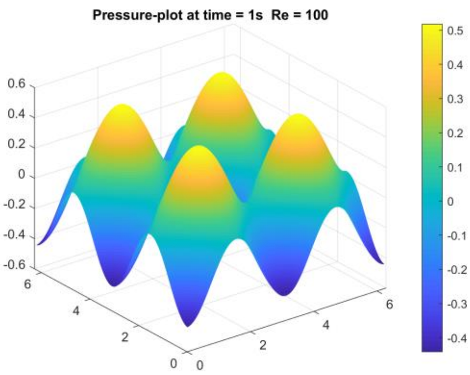
ρ	1
Re $\left(\frac{1}{\nu}\right)$	10;100;1000
Relaxation factor (ω)	1.96
Tolerance (SOR)	10^{-4}
Δt	10^{-3}

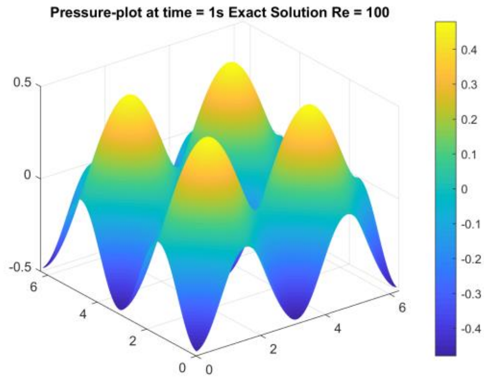
The choice of the most efficient relaxation factor is done by referring the following graph;

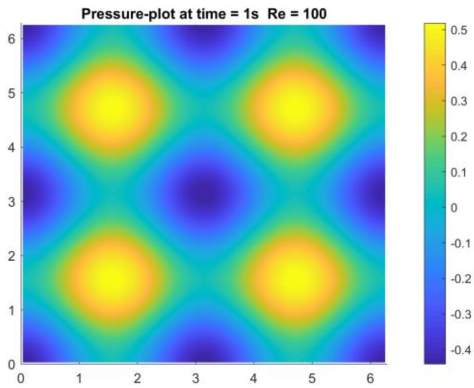


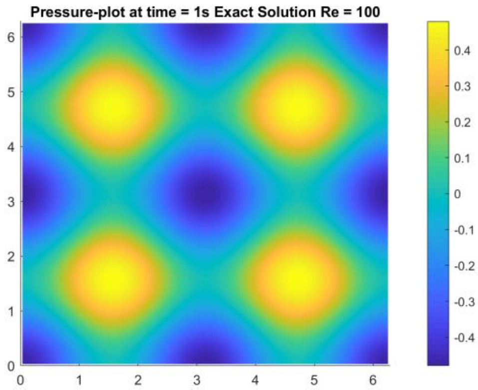
1.3 Results and Discussion

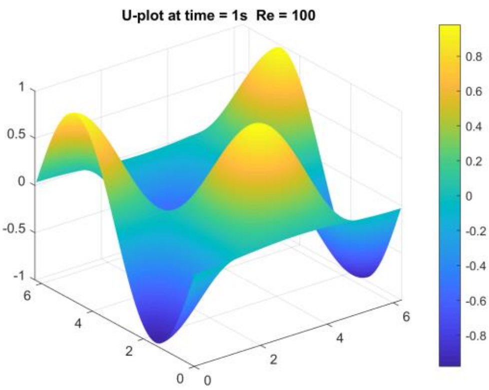
The solution was modelled and code was written in MATLAB which produced the following results:

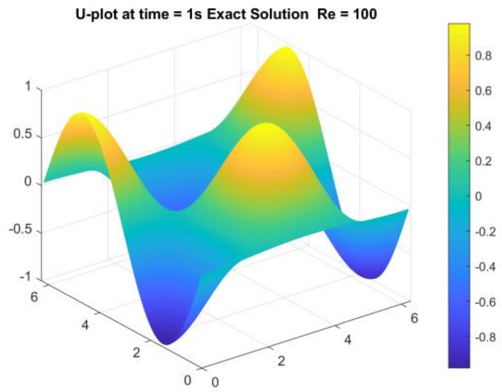


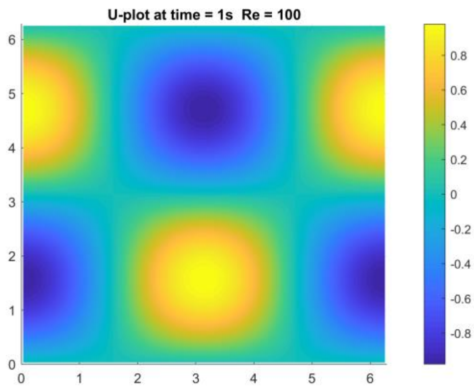


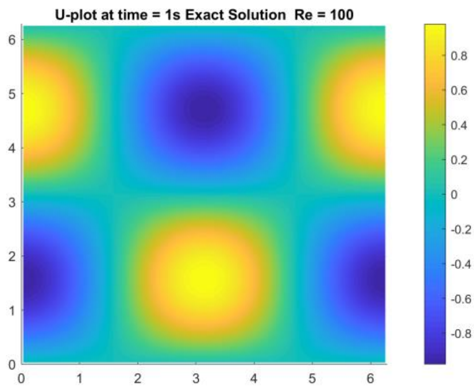


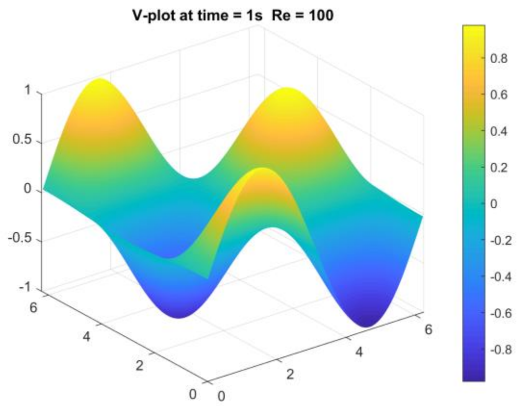


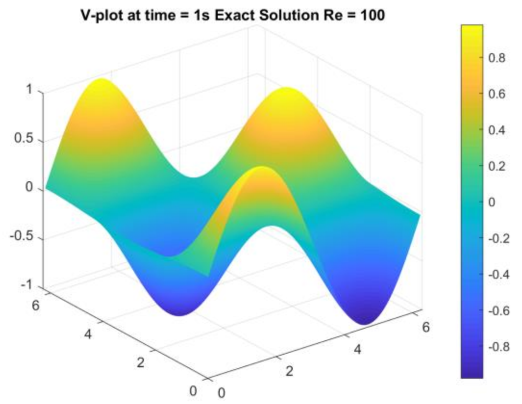


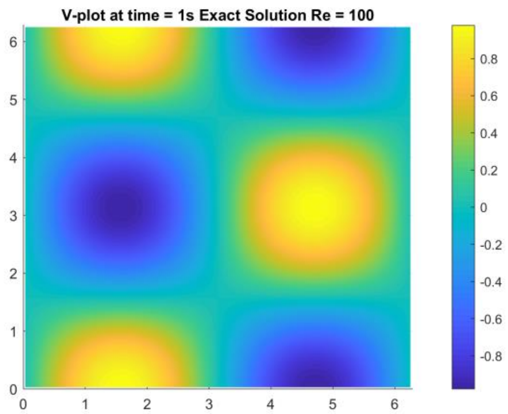


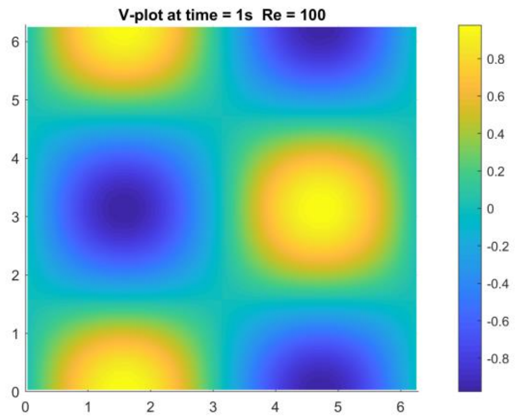






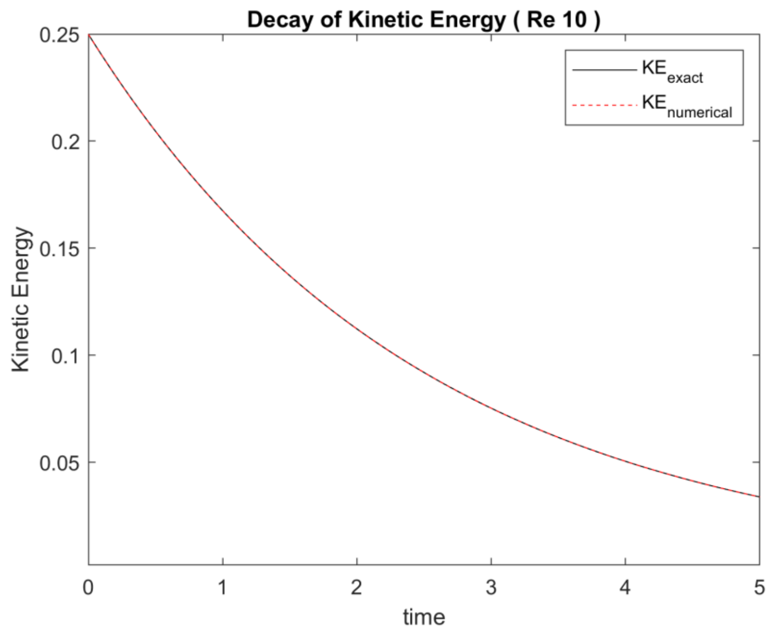


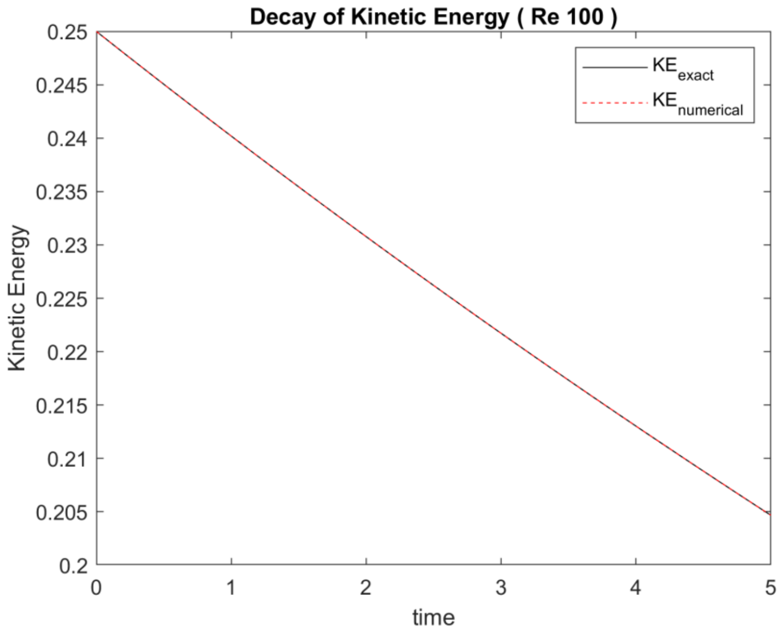


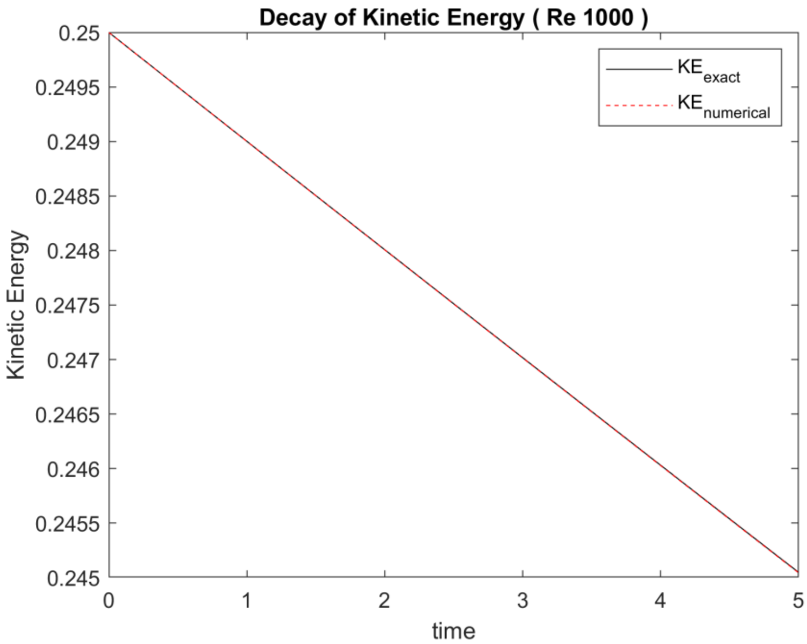


The comparison of the plots obtained from the numerical simulation and the exact solution as provided have been presented and the legend bars on the right of each figure, show that the magnitudes obtained after time=1s are well in line with the exact solution that is obtained. The small time step taken has greatly influenced the accuracy of the results obtained.

The Kinetic energy of the system was measured at each time step and the decay was plotted against time. For comparison the exact Kinetic energy drop is also plotted for reference.







It was observed that as the time stepping was reduced, the graphs of the numerical and exact solution tried merging into one. The Kinetic energy graphs for Re=100 and 1000 seem linear but actually the graphs are exponentially decreasing ones. Here, since the energy dissipated is not too high in 5 seconds, the graph is not able to show the correct curve. However, if we look at the Re=10 graph, we can clearly make out the nature of the decay of Kinetic energy is not linear.

Here in this particular problem,

$$Re \propto \frac{1}{\nu}$$

we could see that as the Reynolds number decreased, there was a greater dissipation of the kinetic energy of the system of vortices.

Since the decay in Kinetic energy computed by the Mahesh algorithm and that provided by the exact solution go hand in hand for the time step considered, the code is written well and hence the Taylor-Green problem is validated.

Problem 2: Lid-driven cavity

2.1 Introduction

The problem is defined as, at time t=0, the lid of a cavity is given a constant velocity in one direction, here in x-direction i.e. u=constant. Due to this, there is a disturbance caused in the cavity which then due to the fixed wall conditions generates rotational circulatory flow in the cavity.

The code, validated on the Taylor-Green problem above, can be modified by making some changes in order to approach the lid-driven cavity problem.

The lid driven cavity is a classical problem in the fluid flow field and a lot of articles have been published regarding the different mesh sizes used, algorithms employed or the Reynolds number considered. For the validation of the problems here, paper by Ghia et al. (1982) will be referred.

2.2 Method of Solution

2.2.1 Domain and Mesh definition

The cell definitions and the calculation of the normals, surface areas and other quantities remain unchanged from the last question. The domain however is altered as follows;

Length of domain in x-direction	1
Length of domain in y-direction	1
Number of cells in x-direction	128
Number of cells in y-direction	128

2.2.2 Boundary conditions

The cavity is defined as having solid walls on the left, right and bottom sides i.e. no slip, and the pressure is conditioned using the Neumann boundary conditions.
Here, the domain exists from [2 to nx+1] and [2 to ny+1] cells.
The boundary conditions for the u, v velocity and pressure is as shown below,

```
u(1,:)=-u(2,:);
u(nx+2,:)=(2*lid_velocity)-u(nx+1,:);
u(:,1)=-u(:,2);
u(:,ny+2)=-u(:,ny+1);

v(1,:)=-v(2,:);
v(nx+2,:)=-v(nx+1,:);
v(:,1)=-v(:,2);
v(:,ny+2)=-v(:,ny+1);

p(1,:)=p(2,:);
p(nx+2,:)=p(nx+1,:);
```

```
p(:,1)=p(:,2);
p(:,ny+2)=p(:,ny+1)
```

2.2.3 Initialization

For t<0, there is nothing going on in the system, hence all the velocities need to be initialized to zero.

2.2.4 Convergence criteria

The problem is that of an unsteady system where at t=0, the lid is given a certain velocity. This develops a circulatory flow in the domain but after one point the flow becomes steady. Hence, here, the fixed time steps used in the last problem can not be used because we don't know when the system might attain the steady state. Hence we use the convergence criteria according to which, if the residual of the new velocity and old velocity are lesser than a certain tolerance, we assume that the steady state is reached. The convergence condition is as shown:

```
criteria=0.0001; %criteria for convergence
converge=0;
.
.
.
while converge==0
. %do Adams Bashforth
.
.
converge=1;
temp1=0;
temp2=0;

for i=2:nx+1
    for j=2:ny+1
        temp1=temp1+(unew(i,j)-uold(i,j))^2;
        temp2=temp2+(vnew(i,j)-vold(i,j))^2;
    end
end
temp1=sqrt(temp1);
temp2=sqrt(temp2);
if temp1<criteria && temp2<criteria
    converge=converge*1;
else
    converge=converge*0;
end
```

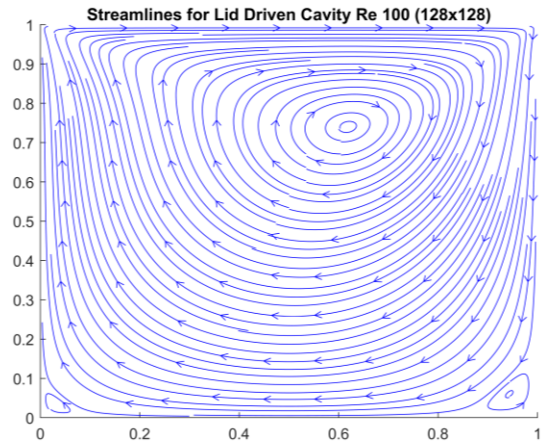
2.2.5 Input parameters

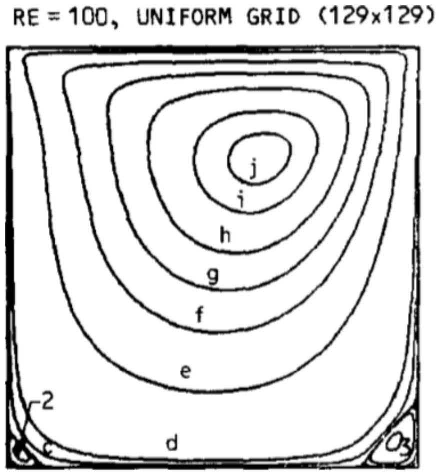
ρ	1
Re $\left(\frac{Lid\_velocity*L}{\nu}\right)$	100;400;1000
Relaxation factor (ω)	1.96
Tolerance (SOR)	10^{-4}
Δt	5x10^{-4}
Convergence criteria	10^{-4}
Lid_velocity	1

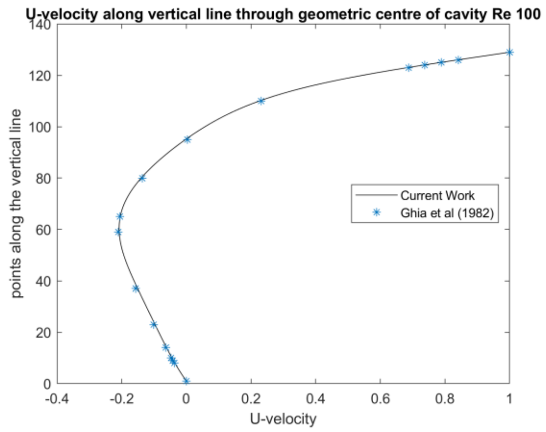
2.3 Results and discussion

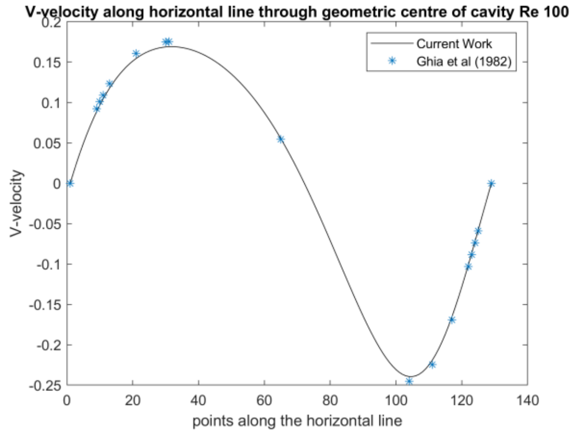
The results obtained by the simulation of the code in MATLAB are compared with the results of simulations performed in Ghia et al. (1982). The simulations have been carried out at three different Reynolds Number, 100, 400 and 1000.

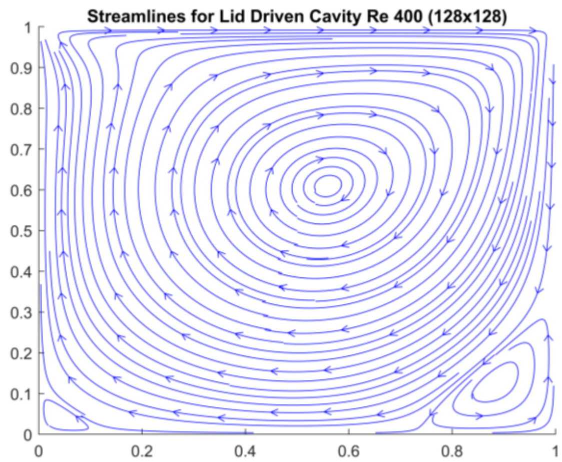
The streamlines obtained by simulation are compared with those obtained by Ghia et al. (1982)

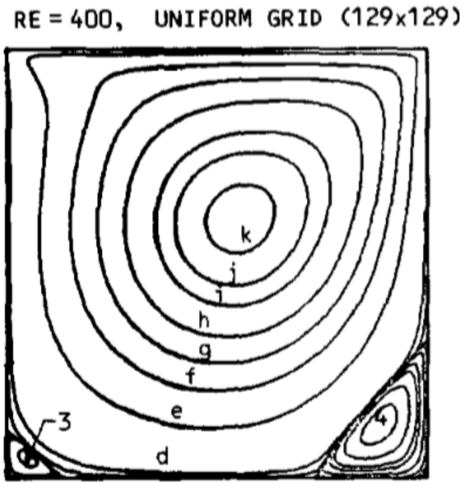


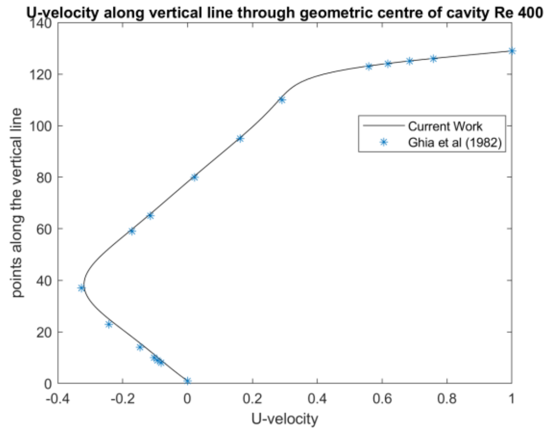


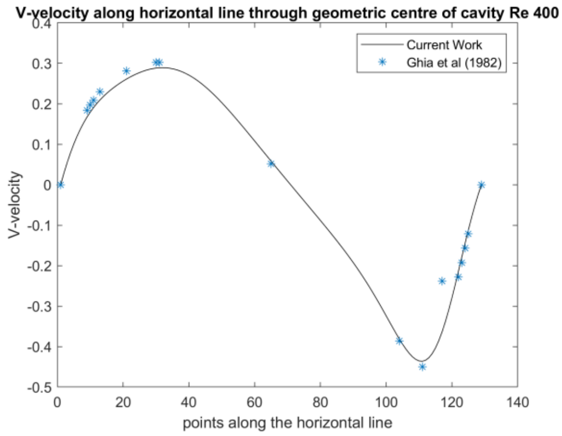


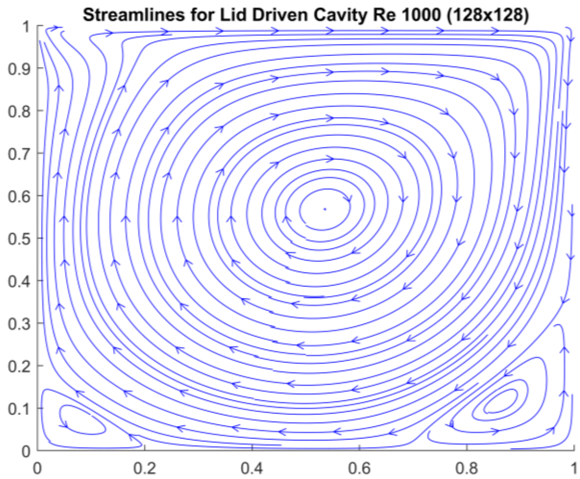


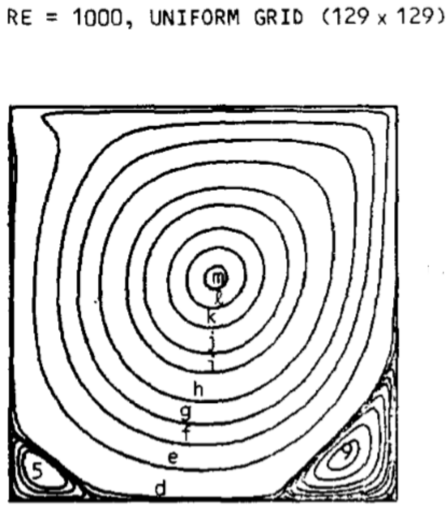


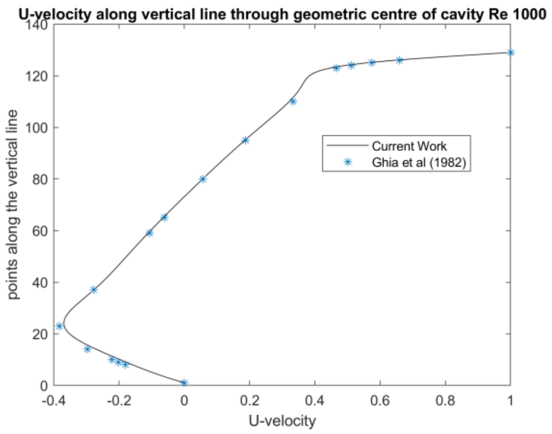


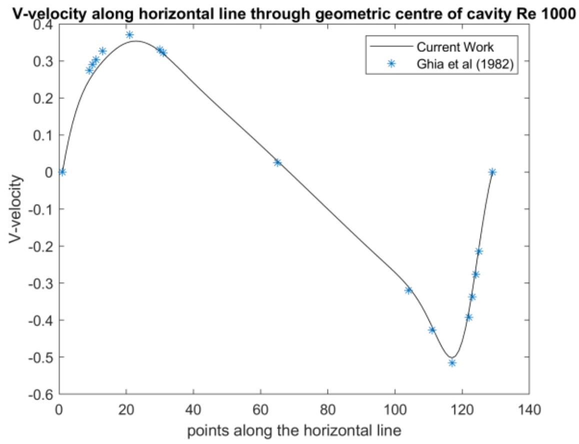












From the above results we can see that using a grid of (128 x 128) elements and using the Adams Bashforth method takes us close to the result obtained by the spectral methods but the accuracy has to be compromised. For Re=100 we can see that the values at the geometric centre are matching to a good extent but for the higher Reynolds number the curves show a considerable error. This error may be due to the incorrect time stepping, compromising on the resolution of the grid in order to speed up the computational time or due to the inefficiency of the numerical method used in order to solve the question. Since, Adams-Bashforth is an explicit method, we see that the time steps we need to take are very small in order to produce a precise result. The main disadvantage of this is the increase in number of steps that we have to consider in order to solve the equation. Moreover, the implicit methods prove to be efficient as they allow a relatively larger time step without compromising the stability of the system.

Other methods in order to get more accurate results can be increasing the resolution of the grid i.e using a 256x256 grid, decreasing the time-step further, using an implicit scheme or using an unstructured grid.

Although the streamlines reveal a picture of how the solution should look when the solution reached a steady state, there are places which could be further taken care of.

References

U. Ghia, N. Ghia, Shin C.T., High-re solutions for incompressible flow using the Navier–Stokes equations and a multigrid method, J. Comput. Phys., 48 (1982), pp. 387-411