Finite Element in Fluids

Incompressible Navier-Stokes equations

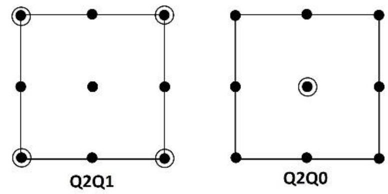
MATLAB Exercise

Student Name

Zahra Rajestari

1. Stokes problem

a) The aim of this part is to study the error obtained from finite element solution for two types of elements: Q2Q1 and Q2Q0 shown in the following:



This can be done using the following formulation

$$e_{\boldsymbol{v}} = \sqrt{\int_{\Omega} (\nabla \boldsymbol{v} - \nabla \boldsymbol{v}^h)^2 d\Omega} \quad \text{and} \quad e_p = \sqrt{\int_{\Omega} (p - p^h)^2 d\Omega}.$$

where v and p correspond to the exact solution for velocity and pressure fields for stokes problem, and v^h and p^h correspond to the FE solution.

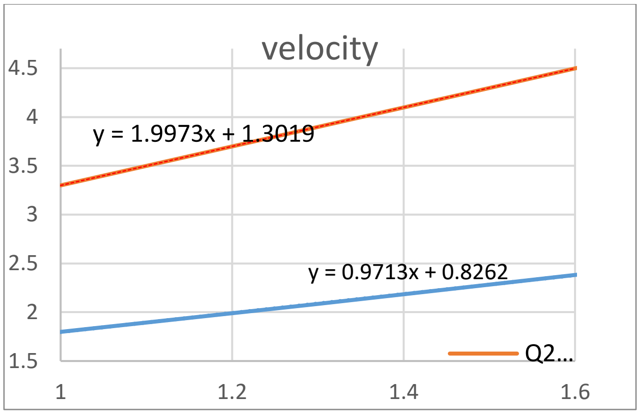
The FE solution is already implemented in MATLAB and the variables corresponding to velocity and pressure are named as "velo" and "pres", respectively. The velocity has two columns which correspond to the two components in x and y direction. The analytical solution is also given for velocity and pressure as functions of x and y in the function called "ExactSol.m".

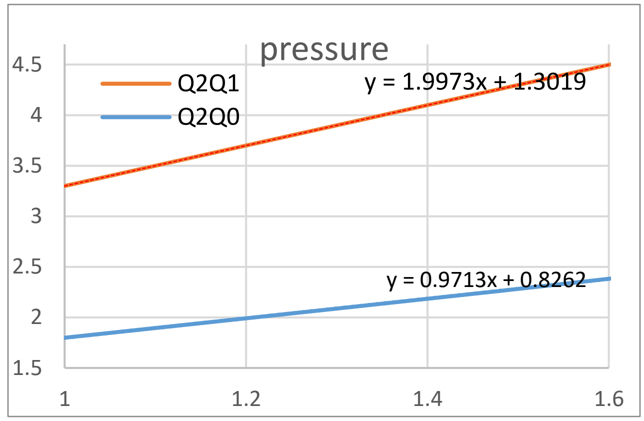
Provided these data, we can compute the error according to the formulation defined previously in which the integration in the domain is done using gauss points. This integration should be calculated for all the nodes in the whole domain which means nx*ny, defining nx and ny to be the number of nodes in x and y direction, respectively. In our case these two numbers are equal. In addition, the derivative of velocity is computed using the shape functions.

After implementing the error, the logarithmic curve for pressure and velocity error is obtained assigning different values for number of nodes (or mesh size). The problem is solved using the following number of nodes and the mesh size is calculated accordingly for square domain of [0,1,0,1]:

Number of nodes	10	20	30	40
Mesh size	1/10	1/20	1/30	1/40

The convergence plot is shown in the following for both Q2Q1 and Q2Q0 elements using uniform mesh of the values previously defined. As it can be seen in the figures, considering Q2Q1 element, both for velocity and pressure the finite element solution converges to the exact solution since the order of the error is 10^-5 for velocity. In terms of the slope of the curve, calculations show that for Q2Q1 element the slope is almost equal to 2 for both velocity and pressure, which confirms the convergence of the error. This is due to the fact that Q2Q1 elements satisfy the LBB condition and therefore give acceptable results for velocity and pressure fields.





The plots also include the results for Q2Q0 elements. As it can be seen, considering the velocity filed, this type of element shows lower convergence to the exact solution compared to that of Q2Q1 element since the error is of order 10^-2 and its slope, according to the linear trend line, is almost equal to 2.

In general, it can be seen from the results that Q2Q1 that satisfies the LBB condition show good results and acceptable value for error and convergence rate both for velocity and pressure. This is while Q2Q0 shows less accuracy and convergence rate for velocity compared to Q2Q1.

b) Before implementing any stabilization method, the solution for pressure has oscillations as expected. The pressure distribution before stabilization is shown below:

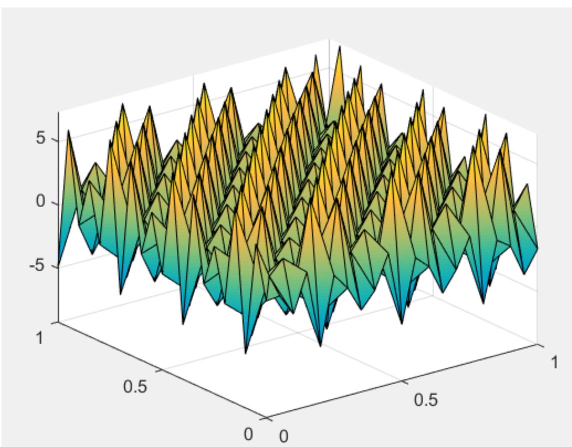
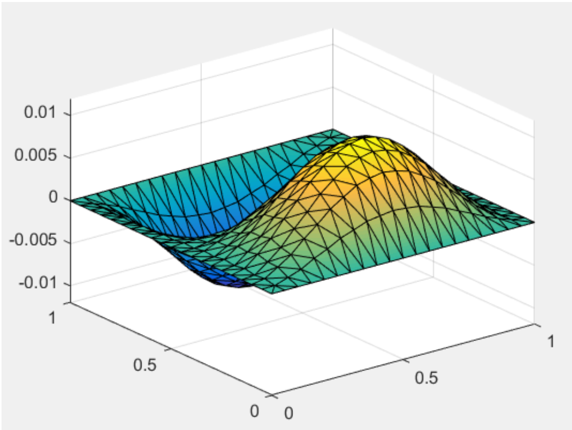


Figure 1 Pressure distribution for P1P1 before stabilization

P1P1 element gives non-oscillatory results for velocity as shown below although no stabilization is applied yet:



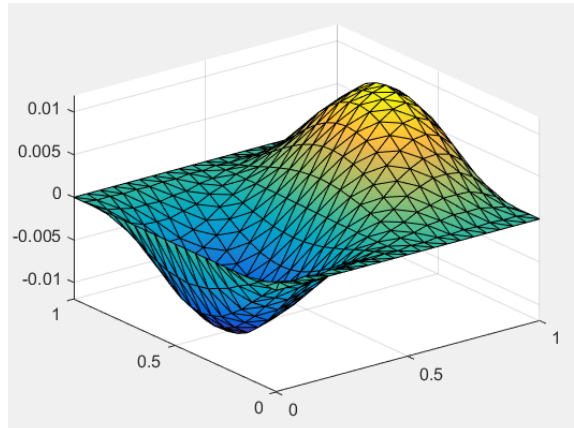


Figure 2 Velocity distribution for P1P1

Stabilization:

In order to implement the stabilization for linear triangular elements for velocity and pressure field discretization, we can use GLS method which deals with addition of a term with a specified stabilization parameter. Since we want to stabilize only the linear element, it is not necessary to deal with the second order derivatives in general formulation of GLS since they are equal to zero.

After applying the stabilization, the result obtained for pressure are not oscillatory anymore as shown in the figure below:

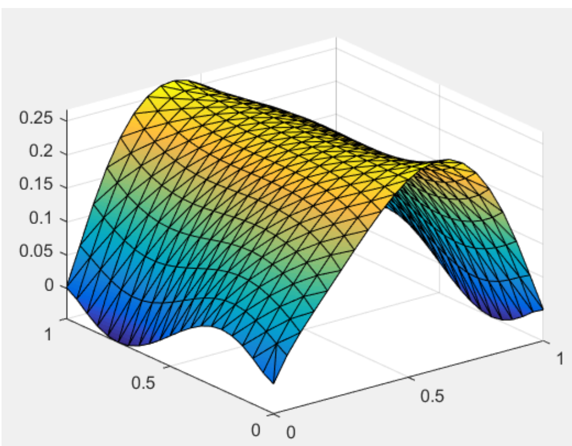
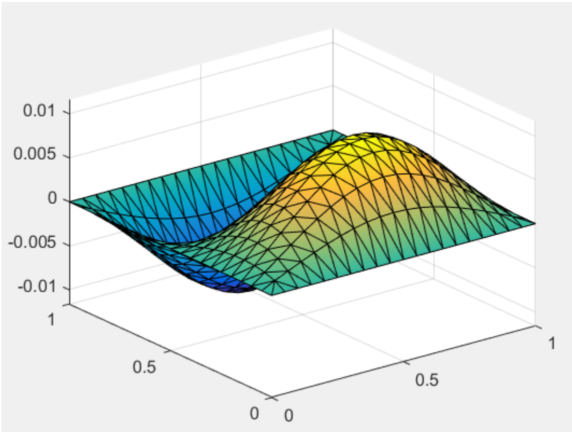


Figure 3 Pressure distribution for P1P1 after stabilization

The velocity keeps to have the same solutions as before since it was already non-oscillatory and stable:



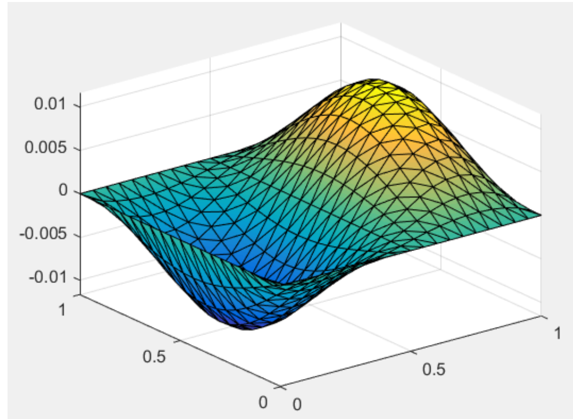
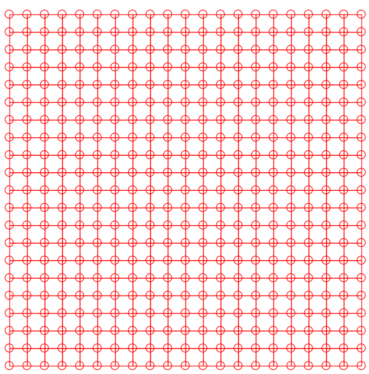
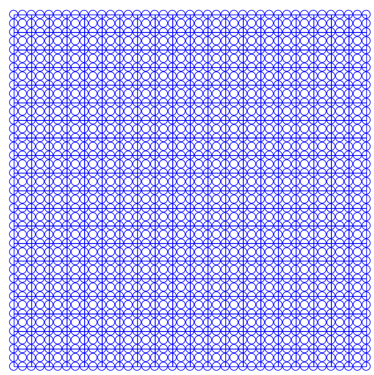


Figure 4 Velocity distribution after stabilization

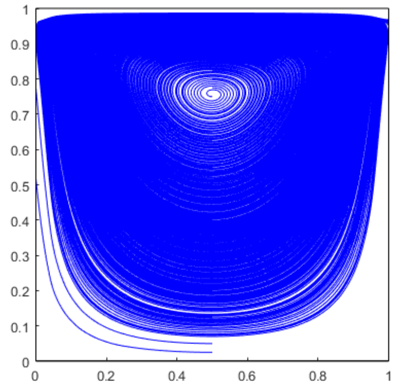
2. This part of the assignment aims at studying the cavity flow problem.

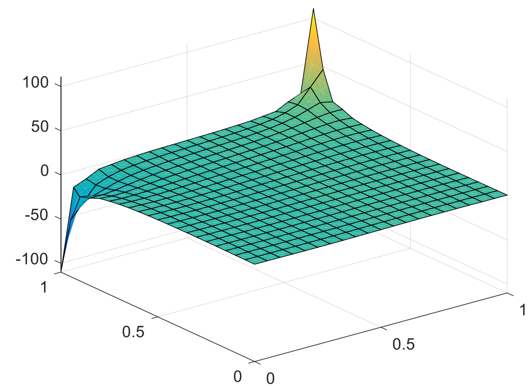
a) We shall use Q2Q1 element for the solution of cavity problem and evaluate the results for two different mesh: 1. Uniform 20*20 mesh, 2. Structured 20*20 mesh refined near the walls. In the code provided, we have the possibility to use to functions to create the mesh called “CreateUniformMesh.m” and “CreateAdaptedMesh.m”.

Using the function that creates uniform mesh, the following results are obtained:

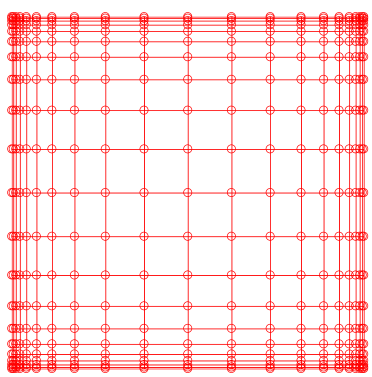
The res mesh represents the mesh for pressure (Q1) and the blue one corresponds to the velocity (Q2). For both, the uniform mesh creation has been chosen.

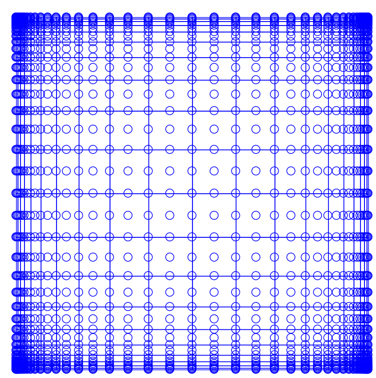




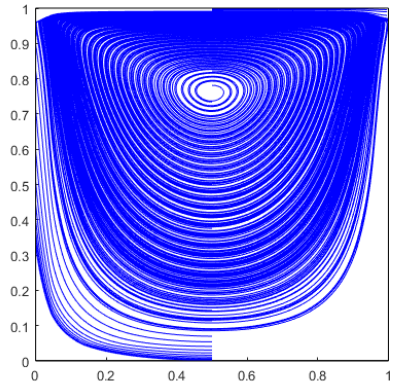
The figure above on the left shows the streamlines and the one on the right shows the pressure. As it can be seen from the figure provided for pressure, the solution is not smooth on the corners where there exist discontinuities in the boundary. Because of this pressure singularity, the mesh in the boundary of the cavity has to be refined since those places are considered to be critical. Not refining the mesh and keeping it uniform as the other parts of the domain results in very sharp changes in pressure and therefore not smooth solution. About the streamline, we can mention its significant feature of being symmetric due to non-existence of shear layers.

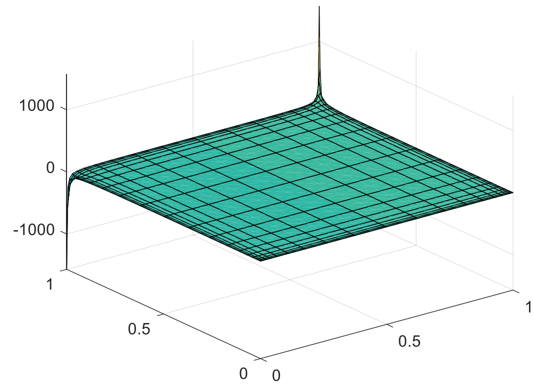
In order to get a better solution for pressure, a non-uniform mesh has to be used keeping the same size for the whole domain but refining it near the boundaries. In order to conduct this, the function that creates adapted mesh has been chosen for boot pressure and velocity discretization. The results are shown below:





The red (Q1) and the blue (Q2) mesh shown above correspond to pressure and velocity discretization, respectively.





The figures for streamline (left) and solution for pressure (right) have been included above. As it can be seen the streamline keeps its symmetry with respect to the vertical axis. about the pressure distribution, we can see that this mesh gives more smooth solution in the boundary where pressure singularities exist. Therefore, it is confirmed that the use of a non-uniform mesh can highly improve the results of the cavity problem.

b) The matrix C arising from discretization of the convective term in Navier-Stokes equation should be implemented in the function called “ConvectionMatrix.m”.

After its implementation, the solution to Navier-Stokes equation has been found assigning the Reynolds number to be Re = 100, 500, 1000 and 2000 using Q2Q1 elements with 20 elements per side. The results for the streamline for different Reylonds are shown below:

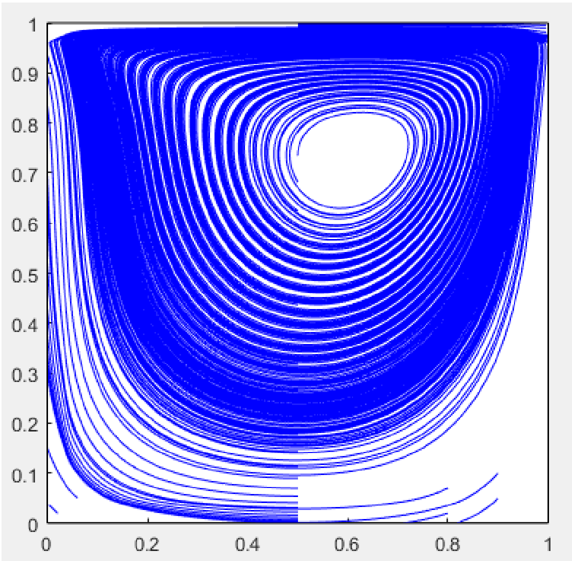


Figure 5 Re=100

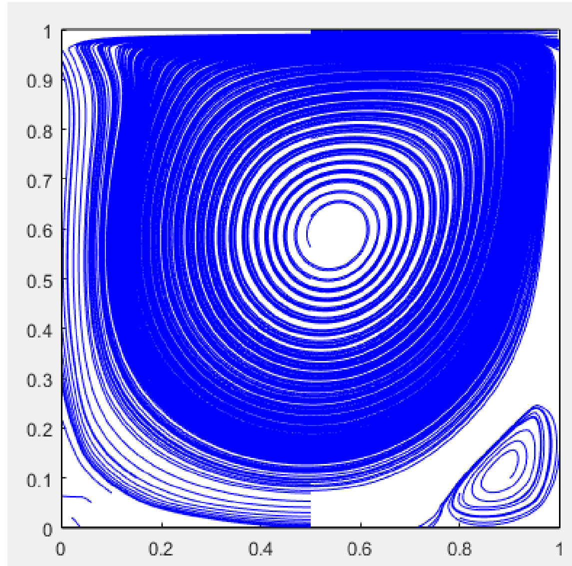


Figure 6 Re=500

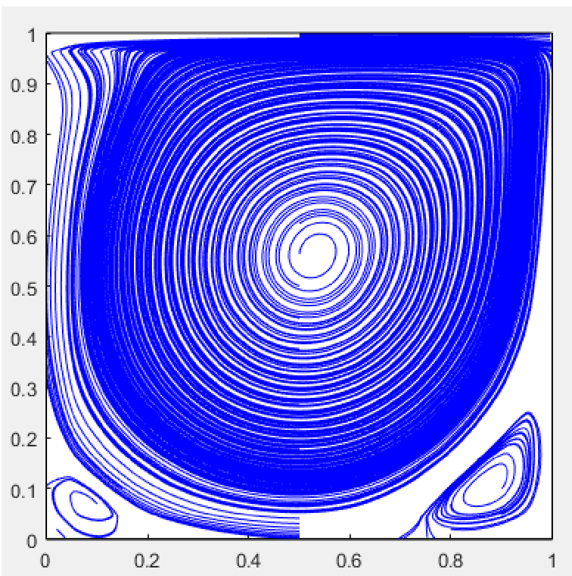


Figure 7 Re=1000

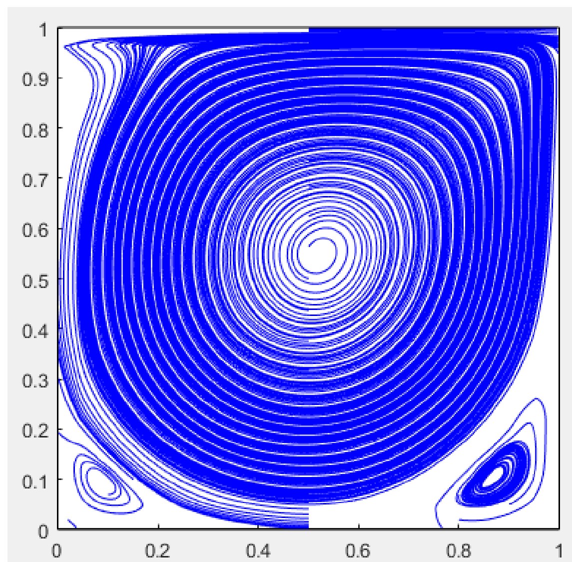


Figure 8 Re=2000

As seen, for high values of Reylonds, vortex appears on the left corner (Re=500) and for higher values the vortex appears both on the right and left corners (Re=1000 and 2000).

As the value of Reynolds is increased the number of iterations is also increased. For Re=100, 500, 1000 and 2000 the numbers of iterations to reach the solution are 13, 29, 35 and 69, respectively.