Incompressible Navier-Stokes equations

PRADEEP KUMAR BAL

May 24, 2018

Stokes problem with analytical solution:

1.1 Con-vergence for Q2Q1 and Q2Q0 elements:

A Matlab function has been written to compute the velocity and pressure errors. The con-vergence for Q2Q1 and Q2Q0 elements has been checked. For Q2Q1 elements the errors converge with slope 2 for both the velocity and pressure as expected. For Q2Q0 elements, For velocity errors converge with a slop of 0.9, and for pressure the slope of -0.2 is observed. These can be observed from the plots below.

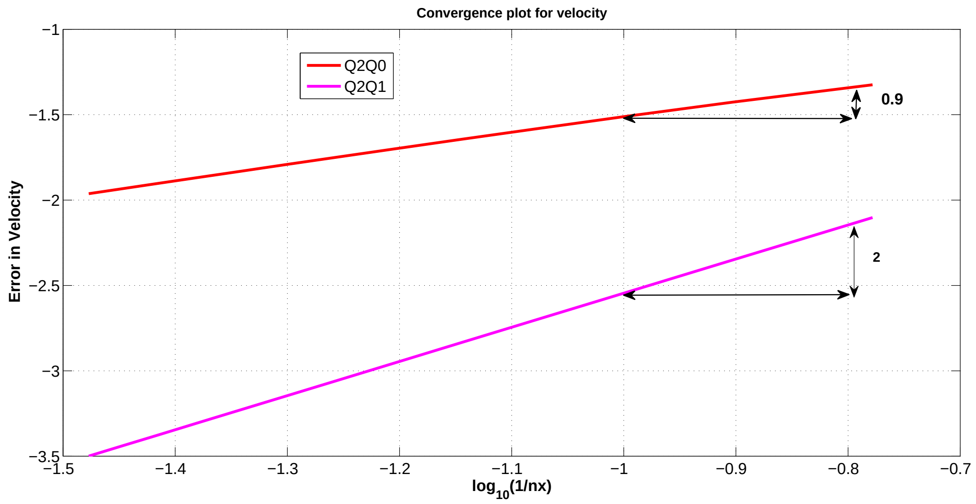

Figure 1: The error convergence of velocity

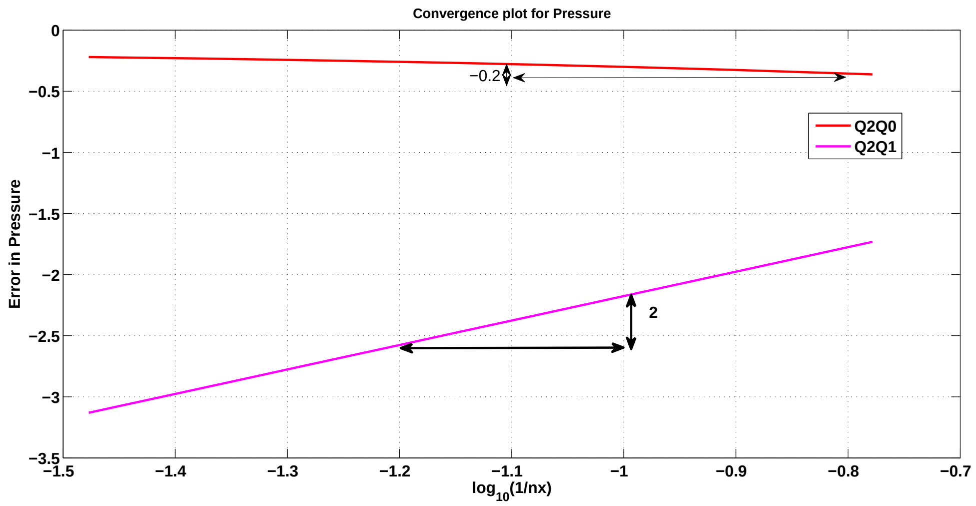

Figure 2: The error convergence of pressure

1.2:**Solution using P1P1 elements:** The given stoke's problem has been solved using P1P1 elements. As these elements are not LBB stable, A GLS formulation has been used for the stabilization. The formulation of the GLS has been presented below for the stoke's problem.

$$\mathbf{a}(\mathbf{w}^h, \mathbf{v}^h) + \mathbf{b}(\mathbf{w}^h, p^h) = (\mathbf{w}^h, b^h) + (\mathbf{w}^h, \mathbf{t}^h)_{\Gamma_N}$$

$$\mathbf{b}(\mathbf{v}^h, q^h) - \sum_{e=1}^{n_{el}} \tau_e (\nabla q^h, \nabla p^h)_{\Omega^e} = -\sum_{e=1}^{n_{el}} \tau_e (\nabla q^h, \mathbf{b}^h)_{\Omega^e}$$

Where $\tau_e = \frac{\alpha_0 {h_e}^2}{4\gamma}, \alpha_0 = 1/3$. interesting consequence of the GLS stabilization of the Stokes problem is that elements with equal order interpolations, which are unstable in the Galerkin formulation, now become stable. The velocity and pressure fields are obtained by implementing this formulation has been compared with that obtained from the previous Q1Q1 formulation without the inclusion of stabilisation terms. These can be observed from the attached figure below using 20*20 element meshes.

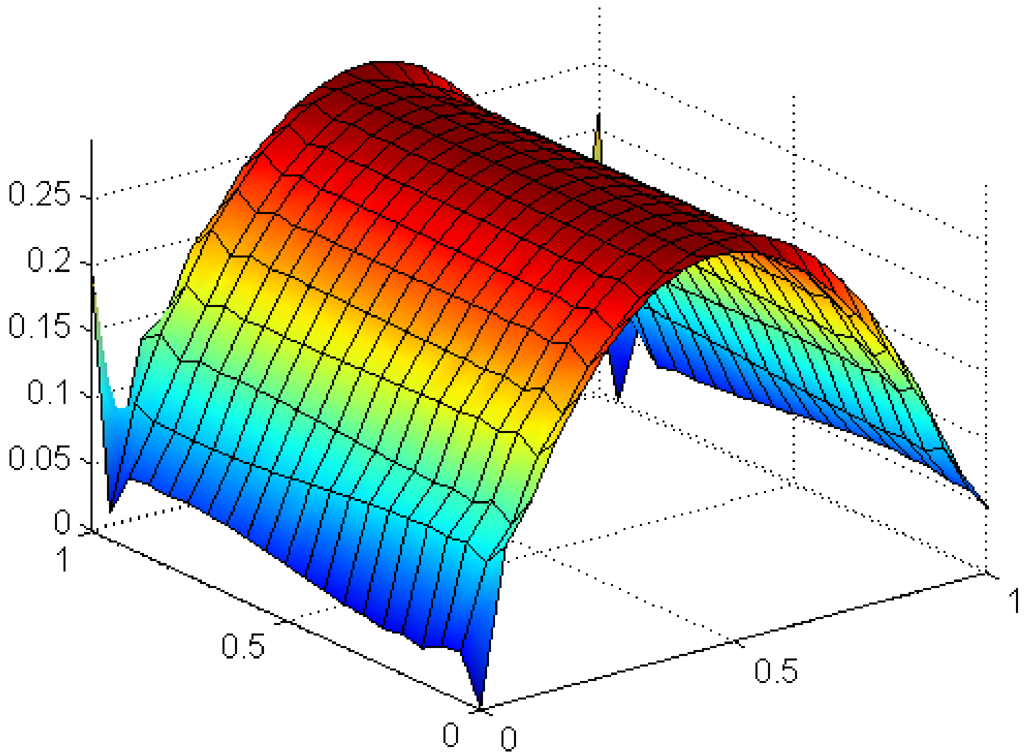

Figure 3: The Pressure profile obtained from the stabilised formulation using Q1Q1

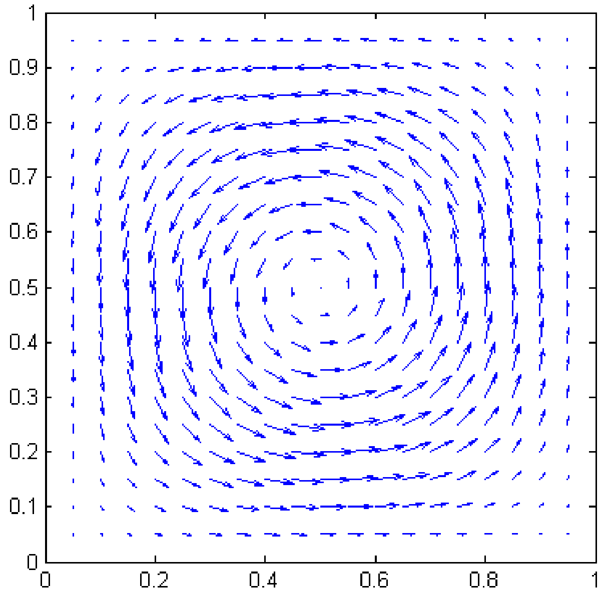

Figure 4: The velocity profile obtained from the stabilised formulation using Q1Q1

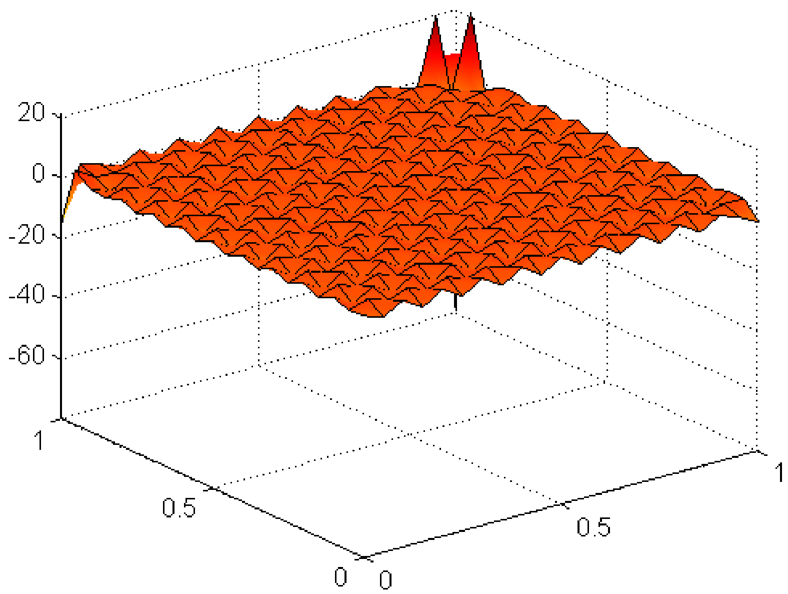

Figure 5: The Pressure profile obtained from using Q1Q1 (no stabilisation terms)

While on the other hand the obtained pressure field has been presented above using Q1Q1 elements with out including the stabilisation parameters. Using the stabilisation more improved pressure field results are obtained.

Convergence of Q1Q1 (Without Stabilization):

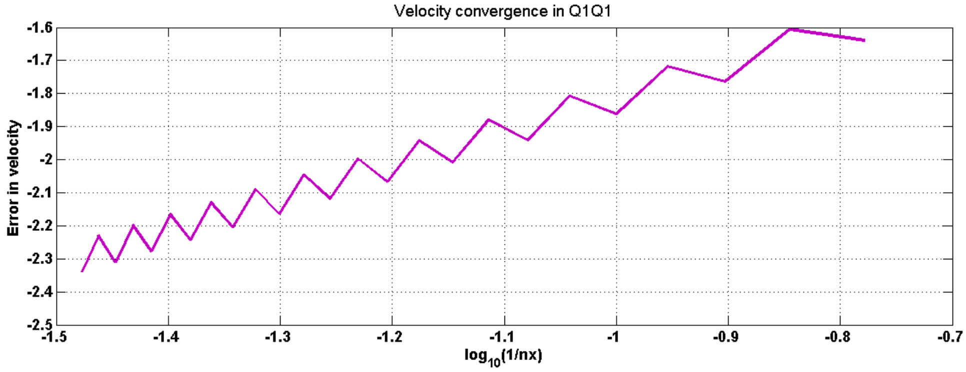

Figure 6: The error convergence of velocity in Q1Q1

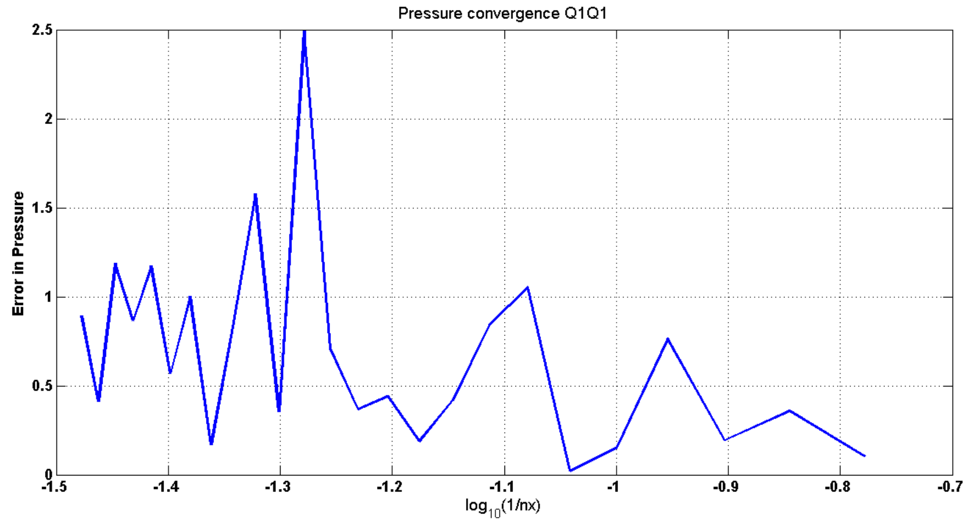

Figure 7: The error convergence of pressure in Q1Q1

2.1 The given problem models a plane flow of an isothermal fluid in a square lid-driven cavity. The upper side of the cavity moves in its own plane at unit speed, while the other sides are fixed. The boundary conditions are indicated in Figure below.

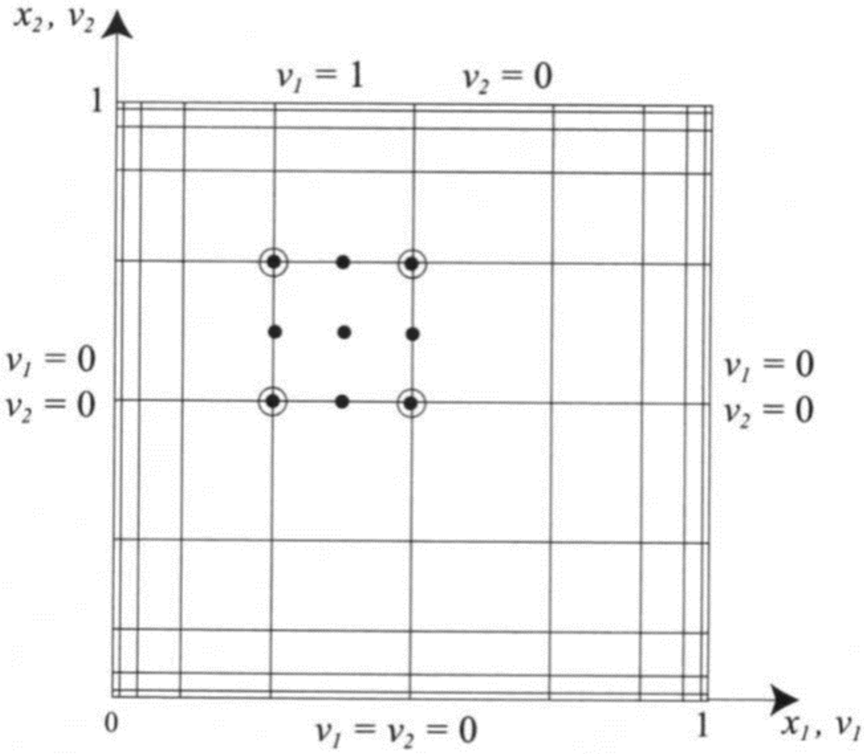

Figure 8: The given problem statement with boundary conditions

First, the given cavity problem has been solved using the Stokes problems.The solution of the Stokes problem has been obtained considering :

- A structured, uniform mesh of Q2Q1 elements with 20 elements per side
- A structured mesh of $20 * 20$ $Q2Q1$elements refined near the walls.

It can be observed that there is a discontinuity in the boundary conditions at the two upper corners of the cavity. Two cases can be envisioned: the two upper corners are either considered as belonging to the top mobile side (leaky cavity), or they are assumed to belong to the fixed vertical walls (non-leaky). The former(leaky cavity) case is adopted here. It introduces a singularity in the pressure field precisely at those two upper corners. As the Dirichlet boundary conditions are imposed on every boundary in this example, this implies that pressure is known up to a constant. Thus at an arbitrary point, the lower left corner of the cavity, the reference value $p = 0$ is prescribed.

The lid-driven cavity for the Stokes problem is solved using the standard Galerkin formulation. The main features in this case are the symmetry with respect to the vertical centerline and the pressure singularity at the two upper corners. In fact, no shear layers are present in the Stokes problem, but results (the pressure jump between both corners) improve if a non-uniform mesh is employed.The cavity is discretized with a a structured, uniform mesh of Q2Q1 elements with 20 elements per side (Case A) and with a structured non-uniform mesh of $20 * 20$ $Q2Q1$elements refined near the walls (Case B). As in both the cases, the Q2Q1 elements, which are LBB compliant, show, as expected, reasonable results for pressure. While in the first case,A, where the mesh is uniform, we can observe small oscillations which are more pronounced in the corners. It produces slightly inaccurate pressure results than the Case B. The element to- element oscillations are observed on uniform meshes (Case A). Figure 2 shows the symmetric streamlines for the Q2Q1 element. Figure 7 shows the pressure field for the considered cases.That is why the Case B(non uniform),elements refined near the walls, is the best.

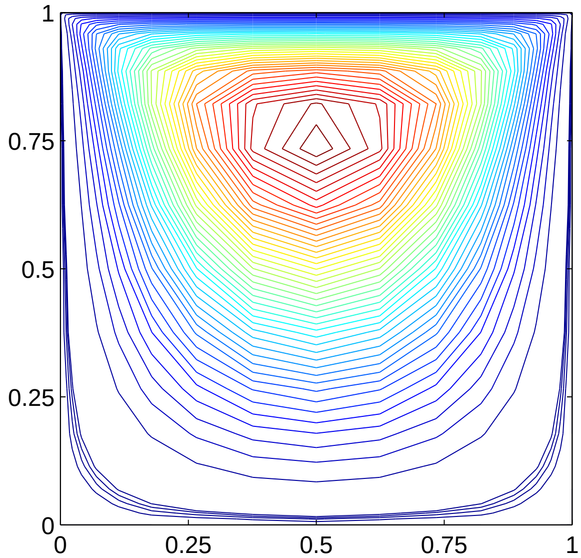

Figure 9: Streamlines for non uniform mesh (Case B)

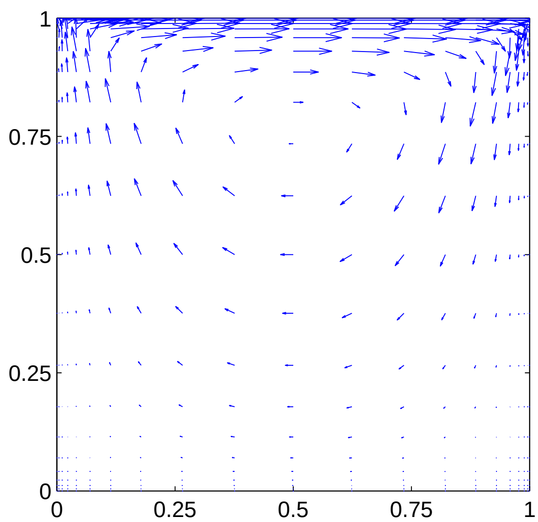

Figure 10: Velocity for non uniform mesh (Case B)

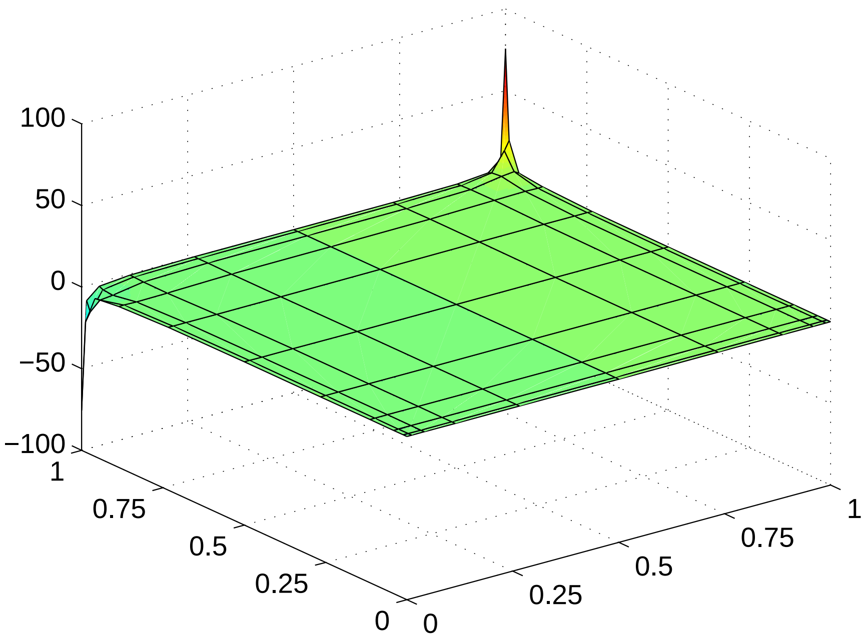

Figure 11: Pressure for non-uniform mesh (Case B)

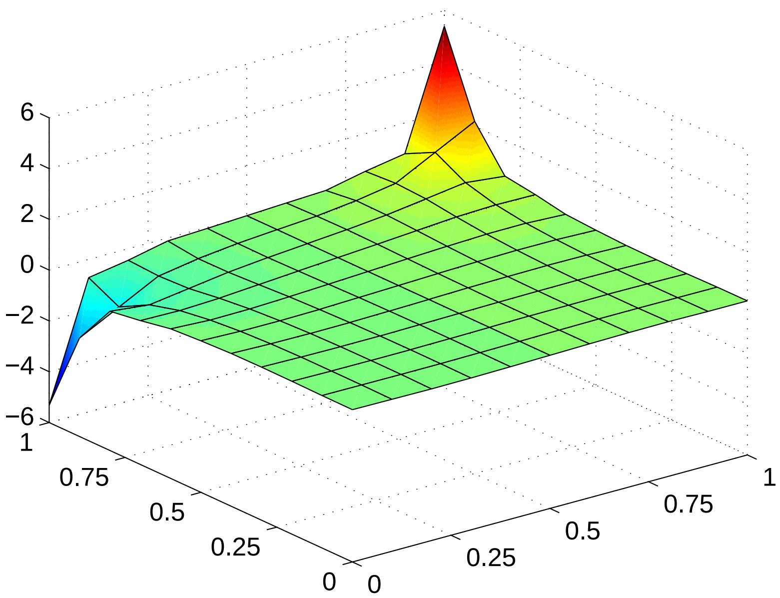

Figure 12: Pressure for uniform mesh (Case A)

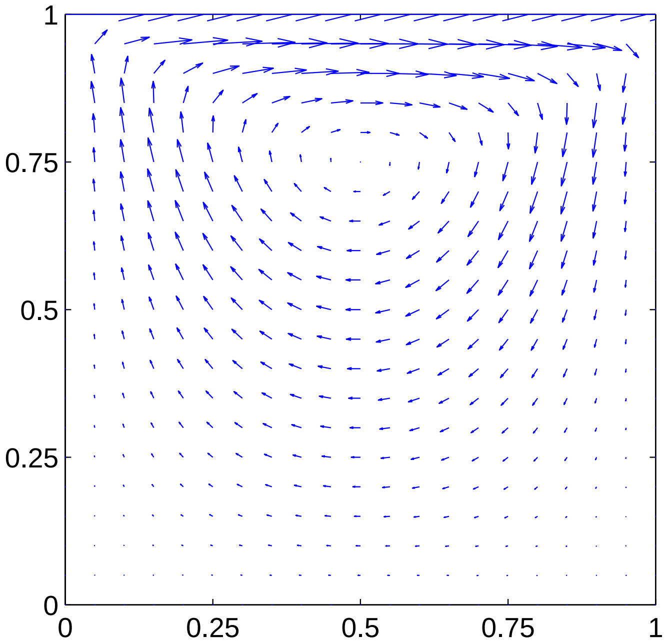

Figure 13: Velocity for uniform mesh (Case A)

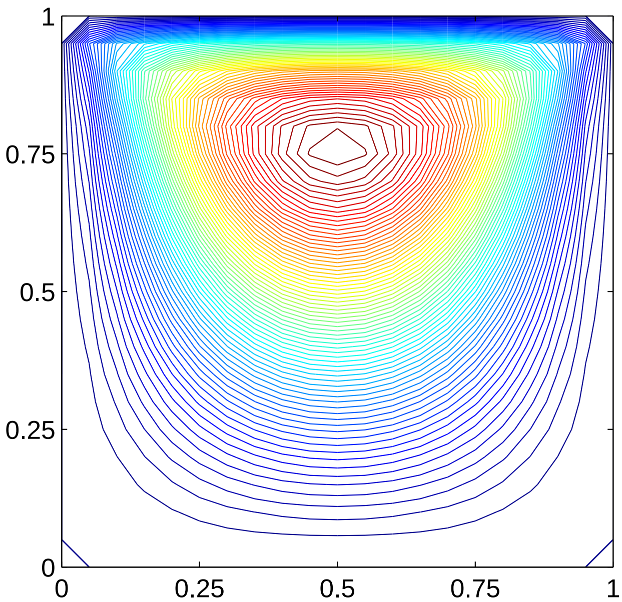

Figure 14: Streamlines for uniform mesh (Case A)

2.b

A Matlab function ConvectionMatrix.m to evaluate the matrix arising from the discretization of the convective term has been implemented.The Navier-Stokes equations using a structured mesh of Q2Q1 elements with 20 elements per side has been solved for different Reynolds numbers. The influence of the Reynolds number can be clearly observed from attached figures below showing the pressure and streamlines for the different considered cases. It is observed that as the Reynolds number increases boundary layers are more obvious and the variations in the velocity profile become sharper.The velocity and pressure results for Reynolds numbers of 100, 500,1000,2000 has been delineated in the figures below. The position of the main vortex can be seen from the images and the values has been mentioned in the table below. The number of iterations taken for the simulation has also been depicted in the table. It is observed that with increase in Re the number of iterations increases. A comparison with some available reference solutions from the literature is also indicated. A satisfactory agreement is observed for all values of the Reynolds number. As can be seen in these figures and in the table, the position of the main vortex moves towards the center of the cavity when the Reynolds number increases. The development of a secondary vortex in the right bottom corner of the cavity becomes progressively apparent and a third vortex appears at the lower left corner. Elevated velocity gradients develop near the cavity walls for large values of the flow Reynolds number. This generates non-physical oscillations in the Galerkin solution for the velocity. A stabilized formulation would then be required.

Re	Iterations	x_1	x_2	strength
100	13	0.62	0.74	0.103
500	26	0.563	0.6	0.110
1000	68	0.54	0.573	0.11
2000	99	0.51	0.54	0.12

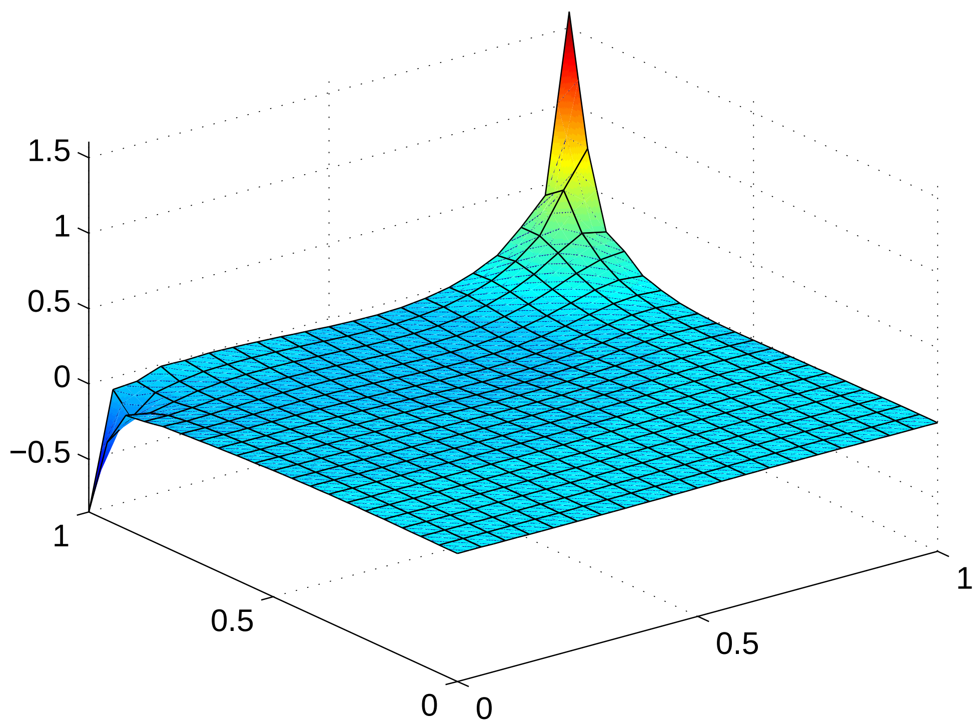

Figure 15: Pressure for Re 100

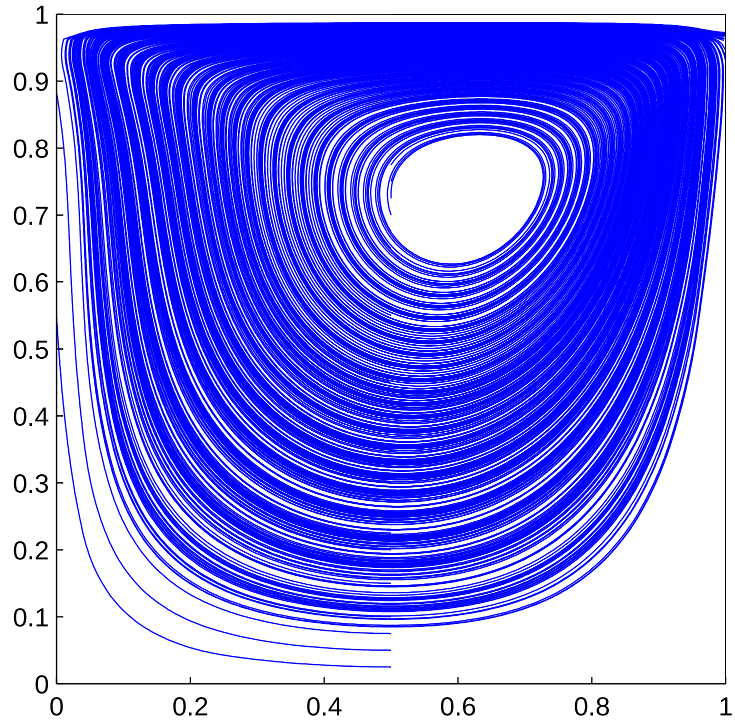

Figure 16: Streamlines for Re 100

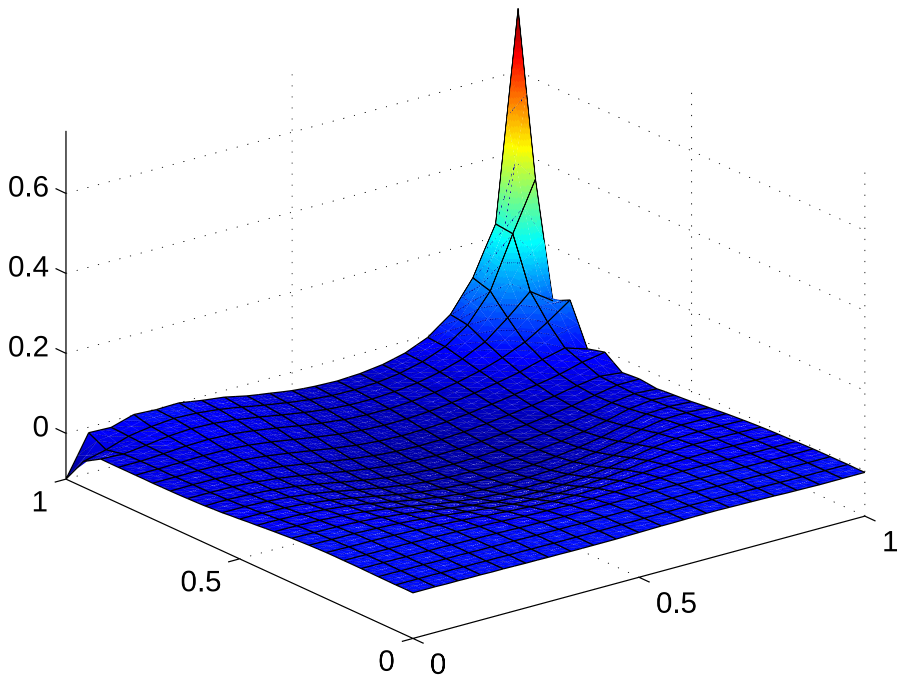

Figure 17: Pressure for Re 500

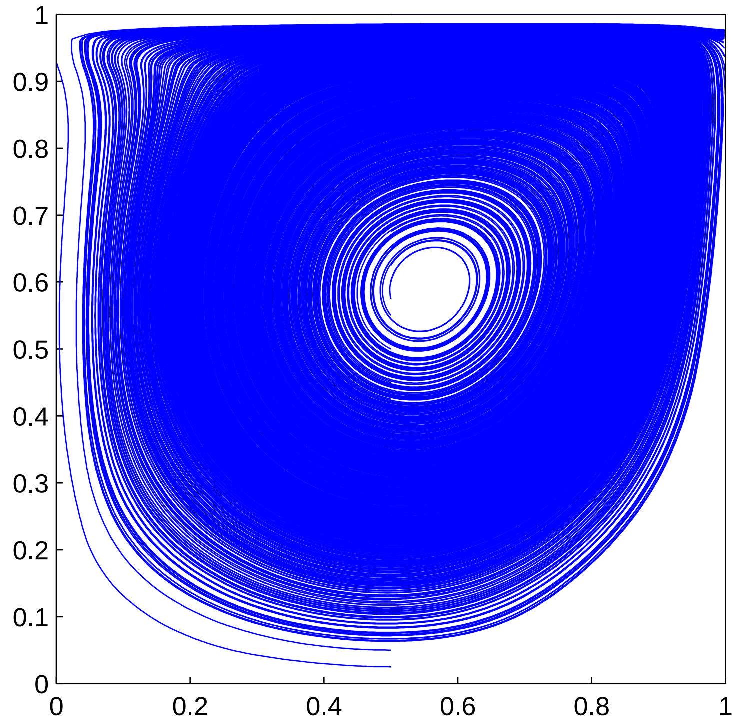

Figure 18: Streamlines for Re 500

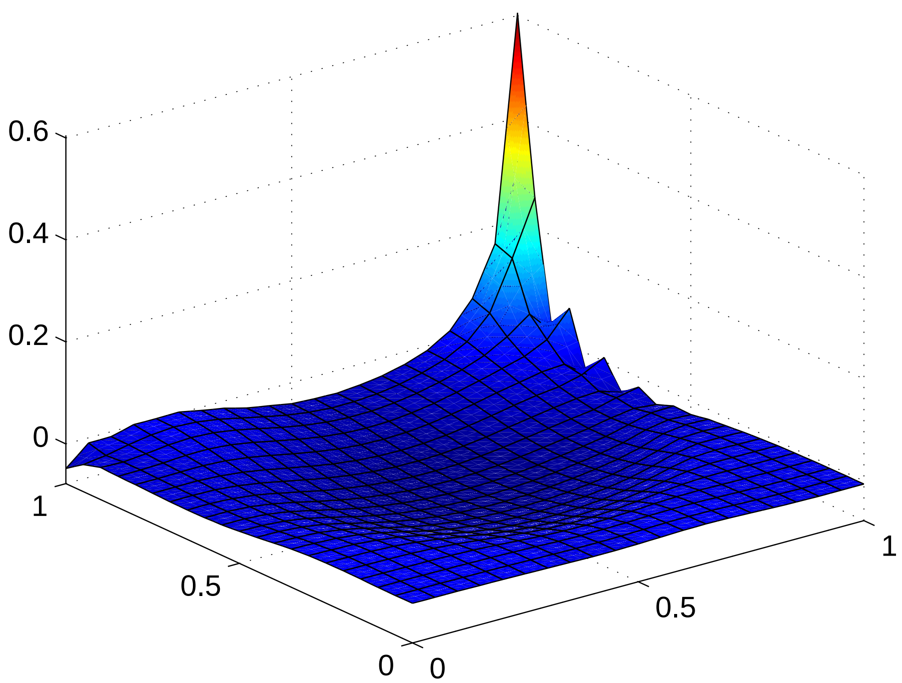

Figure 19: Pressure for Re 1000

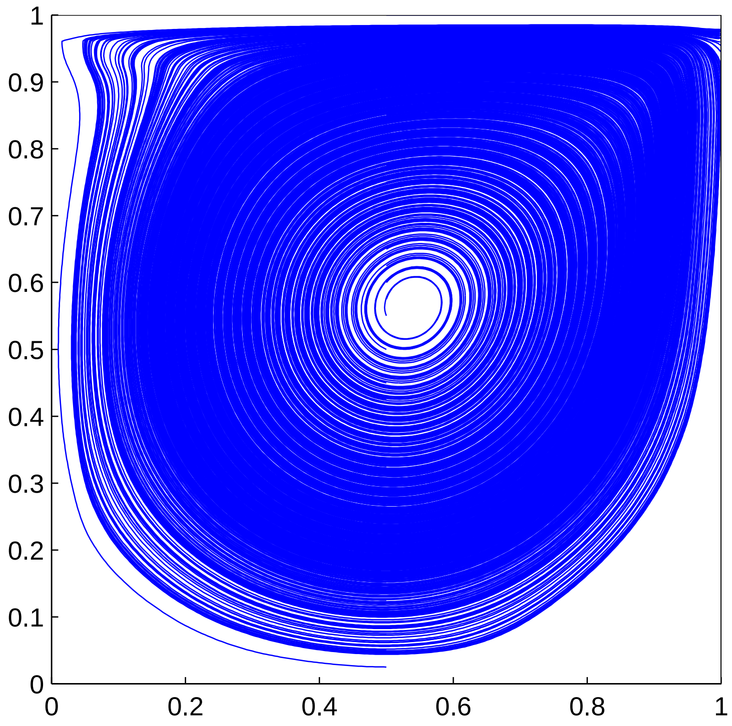

Figure 20: Streamlines for Re 1000

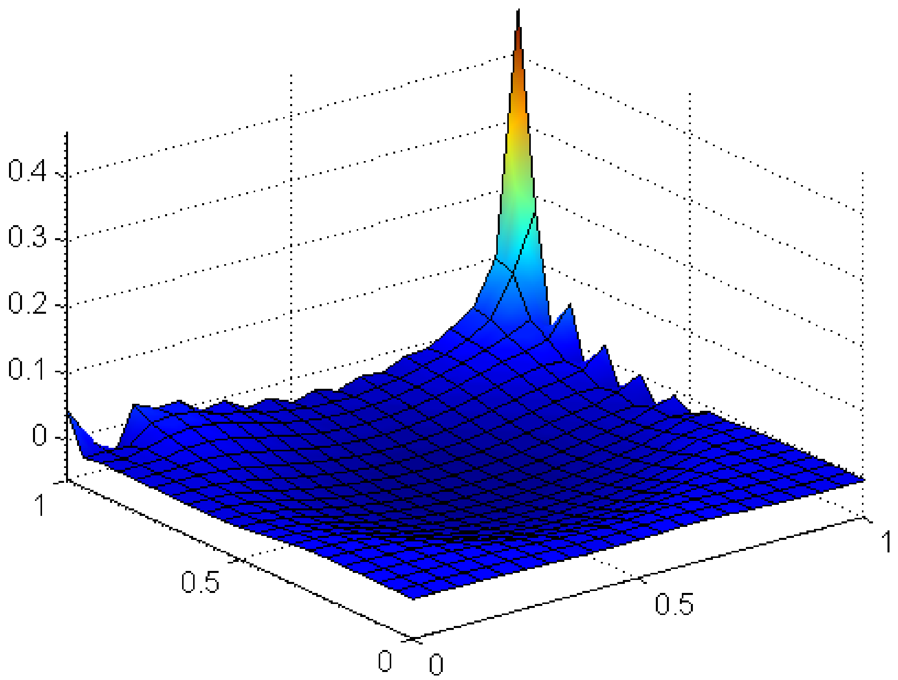

Figure 21: Pressure for Re 2000

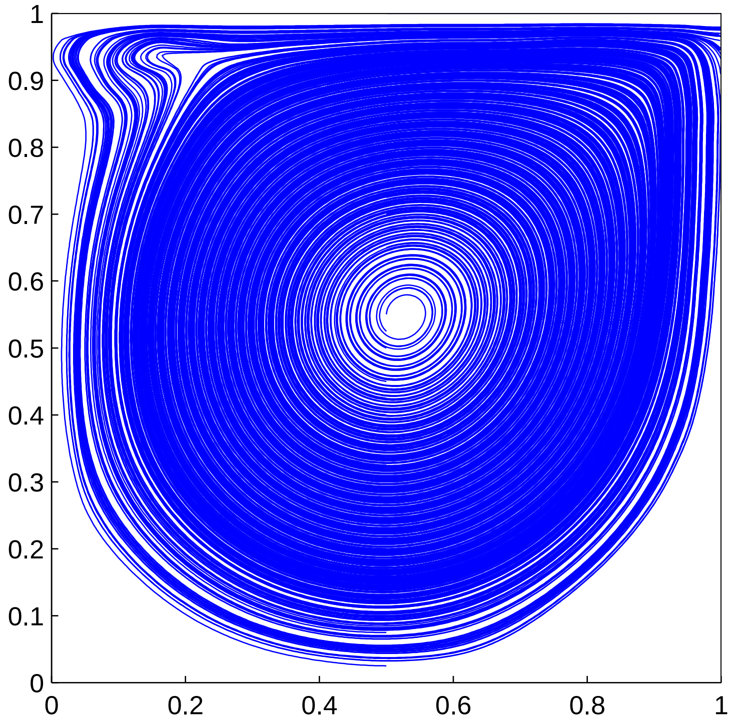

Figure 22: Streamlines for Re 2000

2.C

The given Cavity flow problem has been solved implementing the Navier-Stokes equations using the Newton-Raphson method.We obtain similar results as has been explained above using the Picard's method. Only by using the Newton Raphson's methods it takes less number of iterations for the results to converge. The velocity and pressure results for Reynolds numbers of 100, 500,1000,2000 has been delineated in the figures below. The number of iterations taken for the simulation has also been depicted in the table. It is observed that with increase in Re the number of iterations increases. It can be seen in those figures that, the position of the main vortex moves towards the center of the cavity when the Reynolds number increases as we observed previously.

Re	Iterations	x_1	x_2	strength
100	5	0.61	0.73	0.103
500	11	0.55	0.59	0.110
1000	24	0.53	0.58	0.11
2000	38	0.505	0.55	0.12

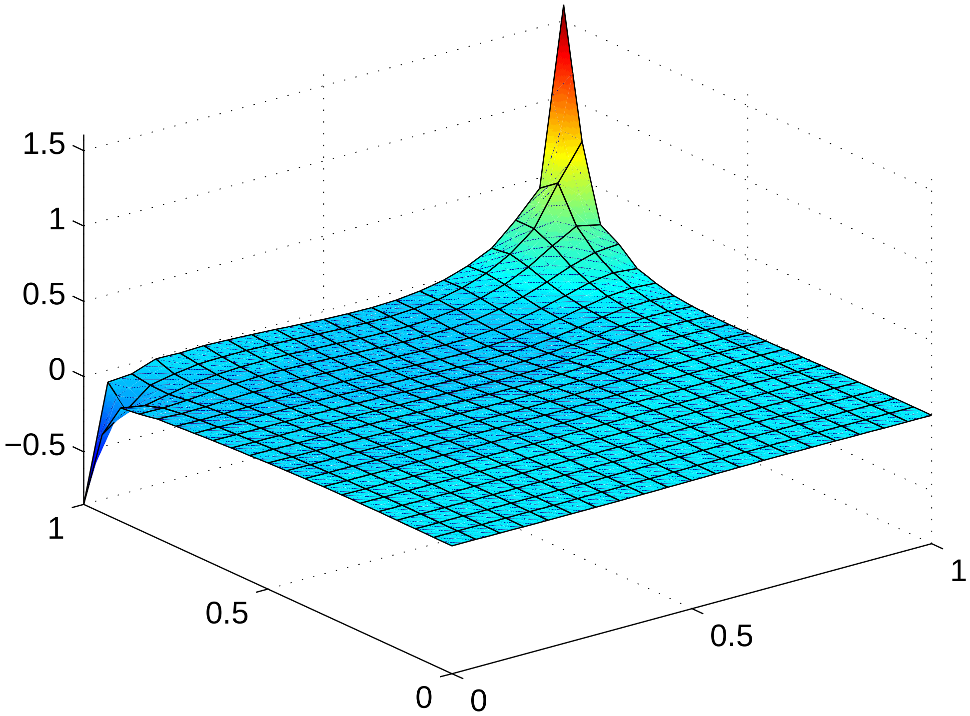

Figure 23: Pressure for Re 100

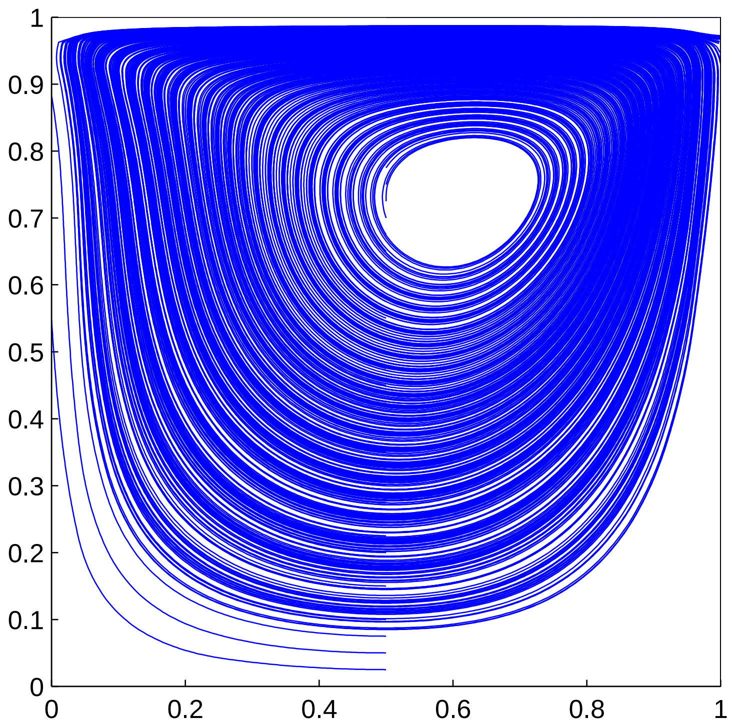

Figure 24: Streamlines for Re 100

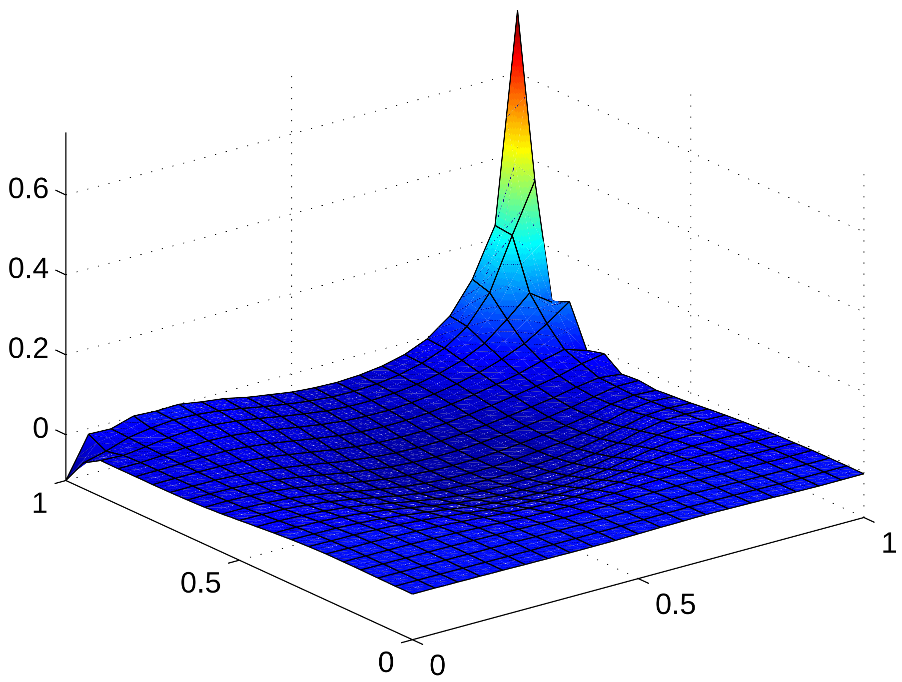

Figure 25: Pressure for Re 500

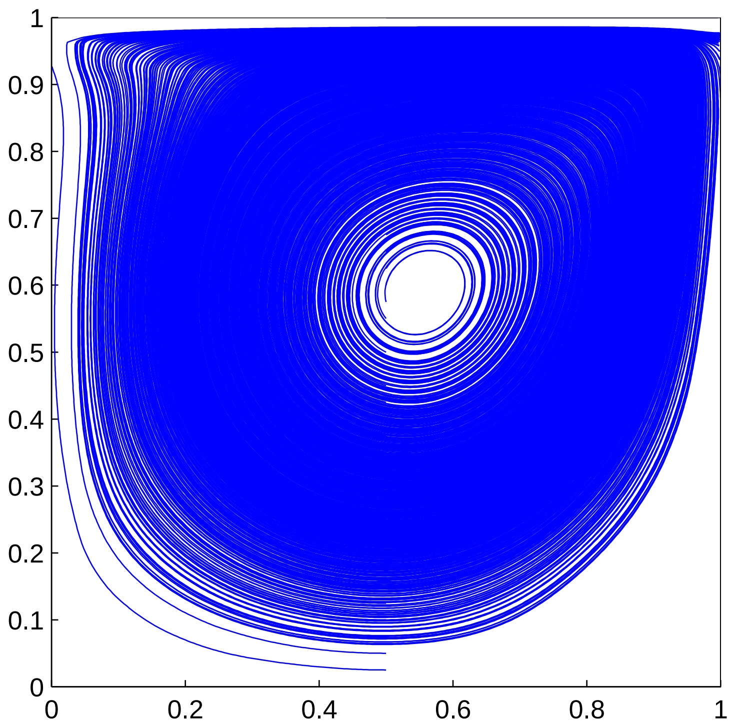

Figure 26: Streamlines for Re 500

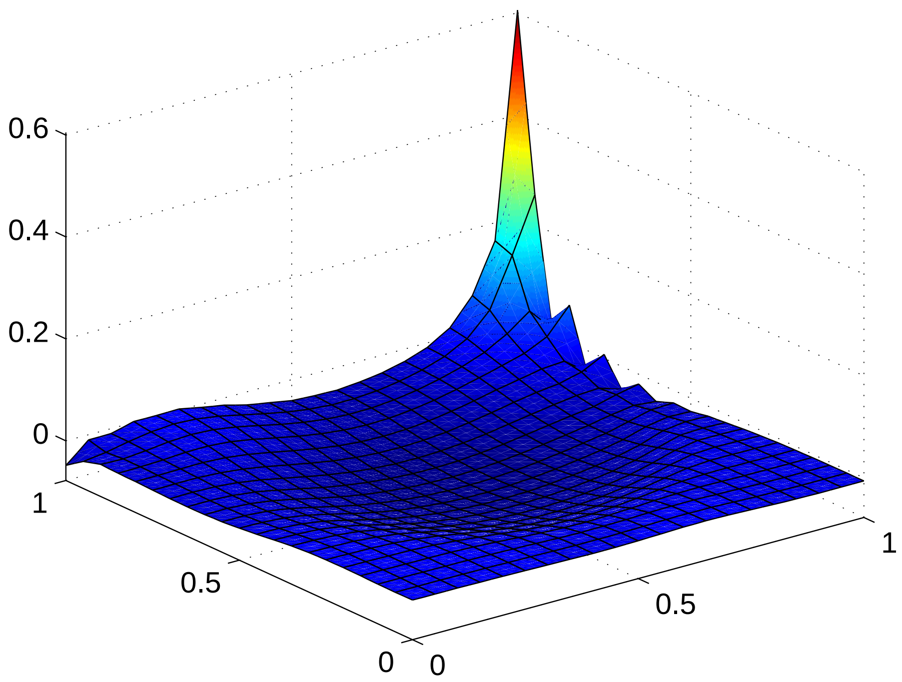

Figure 27: Pressure for Re 1000

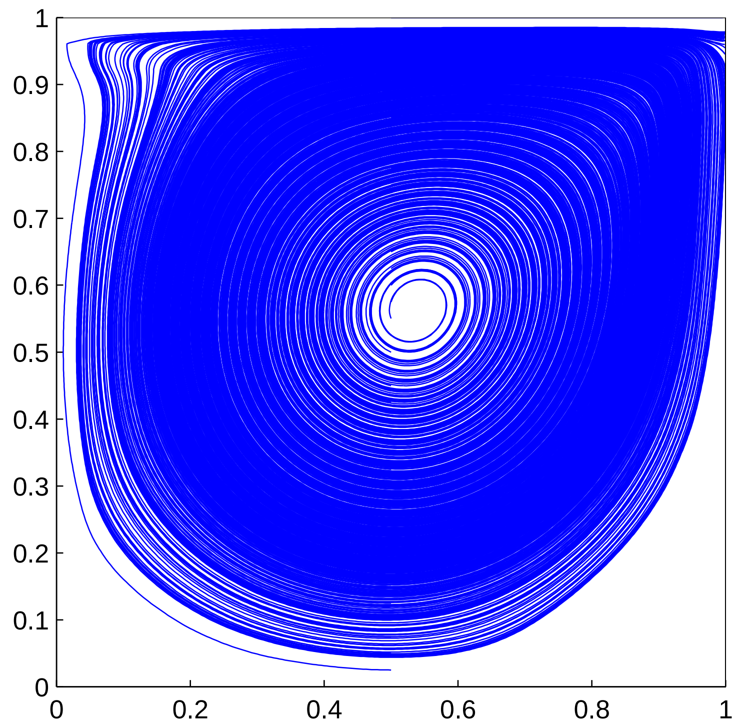

Figure 28: Streamlines for Re 1000

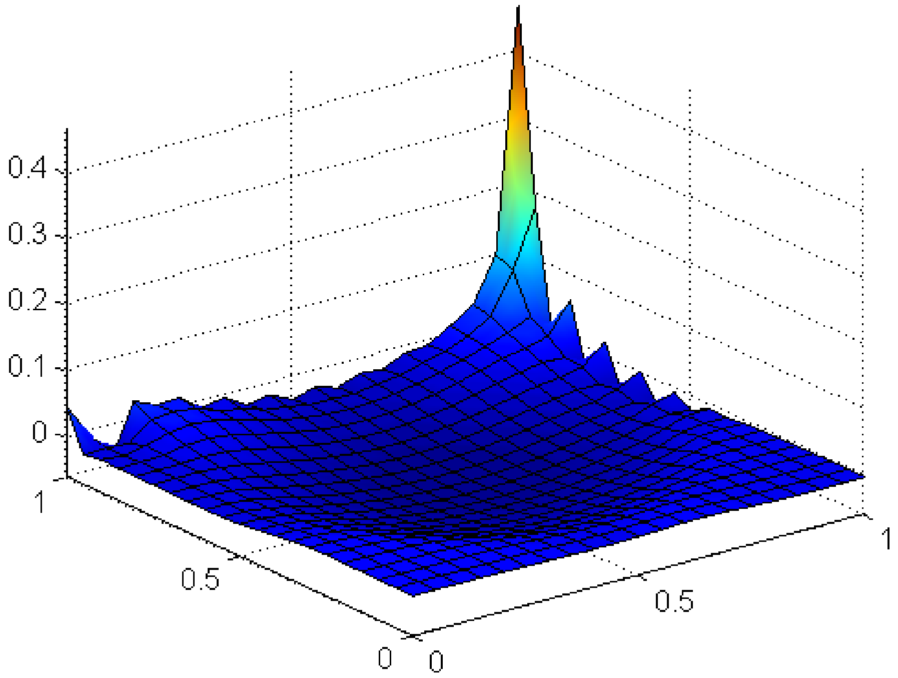

Figure 29: Pressure for Re 2000

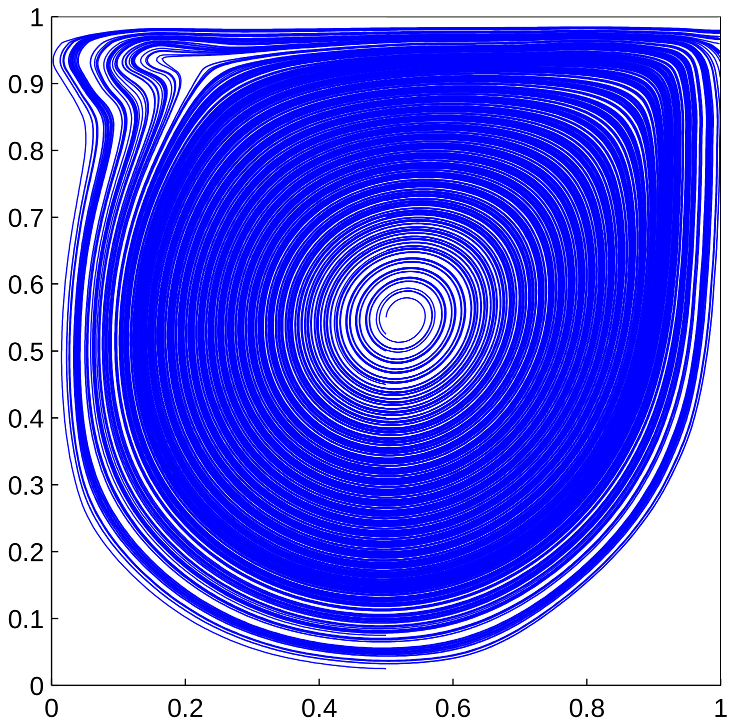

Figure 30: Streamlines for Re 2000