Technical University of Catalonia

FINITE ELEMENTS IN FLUIDS

ASSIGNMENT 2

INCOMPRESSIBLE VISCOUS FLOWS

Stokes / Navier-Stokes

M.Sc. Computational Mechanics – CIMNE

Mohammad Mohsen Zadehkamand

5 June 2017

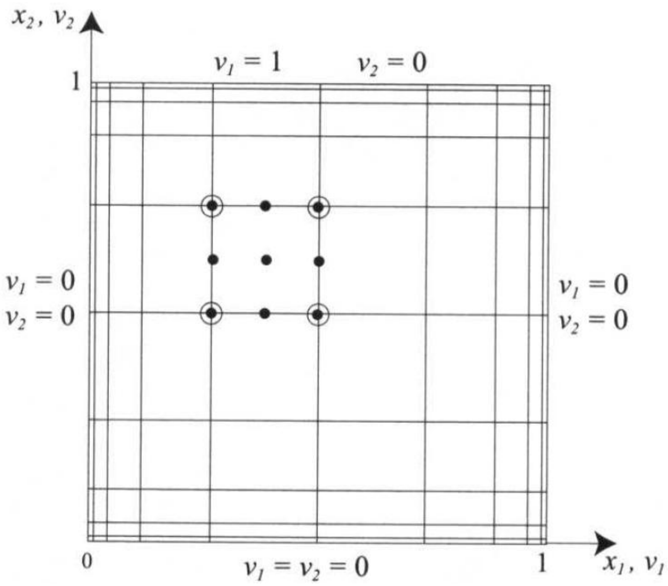
Table of Contents

1) *Assignment Statement*

This is the report for **Assignment_2** of the course **"Finite Element in Fluids"** which deals with the cavity flow problem as the standard benchmark test for incompressible flows. The goal of this assignment is to analyze and get familiar with some behaviors of the Stokes and the Navier-Stokes equation. Two Matlab codes namely ***mainStokes.m*** for Stokes problem and ***mainNavierStokes.m*** for Navier-Stokes problems are provided and there would be some modifications in each one in order to run properly as one of the tasks of the assignment.

- **Stokes** : Incompressible highly viscous flows
- **Navier-Stokes** : Incompressible convective viscous flows

The figure below shows a schematic representation of the problem setting. It represents a plane flow of an isothermal fluid in a square lid-driven cavity in which the upper side of the cavity moves in its own plane at unit speed (horizontally from left to right), while the other sides are fixed. The two upper corners are considered (from pressure point of view) to belong to the top moving side which introduces a singularity in the pressure field precisely at those two upper corners. So, there is a discontinuity in the boundary conditions at two upper corners of the cavity and we will witness this effect.



Finally, it should be mentioned that Dirichlet boundary conditions are imposed on the whole boundary and as commented in the class and also in the main reference book, "***Finite Element Methods for Flow Problems***" by *Jean Done*a and *Antonio Huerta*, this implies that pressure would be known in infinite manners up to a constant. So to make it specific, at an arbitrary point, namely the lower left corner of the cavity, the reference value p = 0 is prescribed in the code.

2) Stokes Problem

a) Element selection effect

In this part we are going to solve the cavity problem for the Stokes equation using the standard Galerkin formulation. ***Figure1*** and ***Figure2*** portray the pressure distribution and the streamline representation in the domain due to this highly viscous (nu=1) incompressible flow.

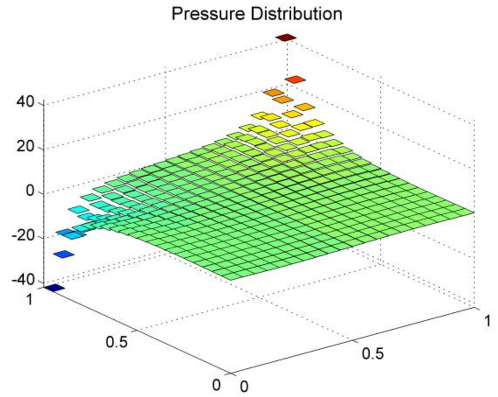
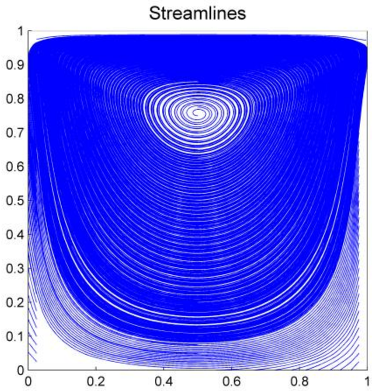
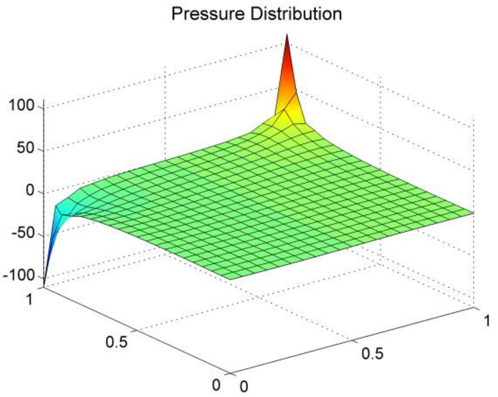
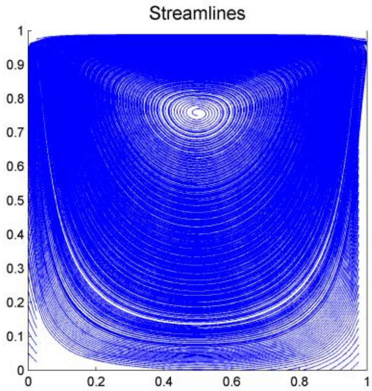
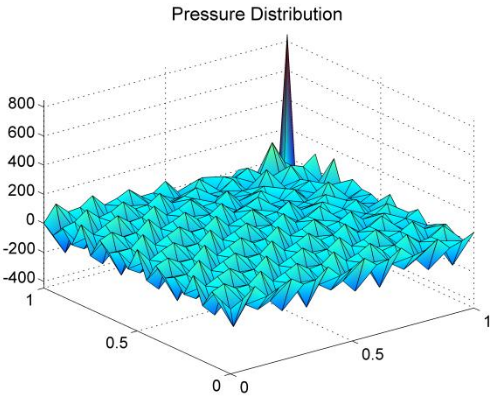
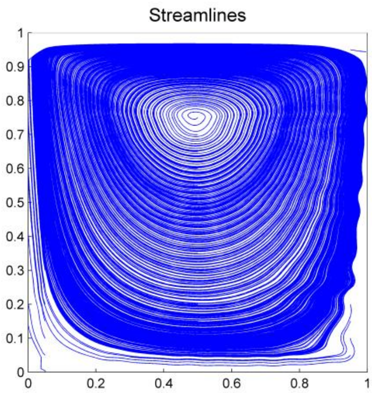
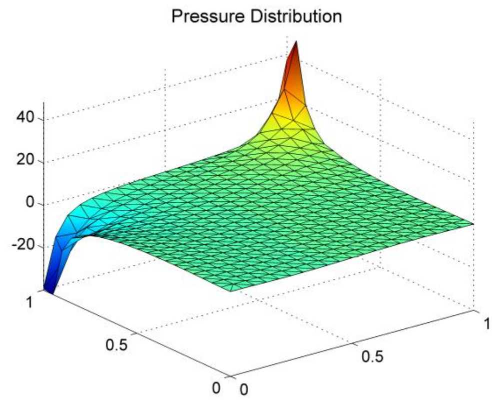
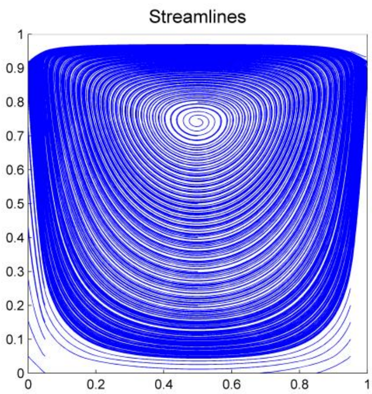
As for the element types, 4 main cases is chosen to study:

ELEMENT	TYPE	ORDERS
Q2/Q0	Quadrilaterals	2^{nd} order velocity – constant pressure
Q2/Q1	Quadrilaterals	2^{nd} order velocity – 1^{st} order pressure
P1/P1	Triangular	1^{st} order velocity – 1^{st} order pressure
MINI P1+/P1	Triangular	1^{st} order velocity with bubble function – 1^{st} order pressure

As it can be seen in ***Figure1*** and ***Figure2***, we have captured the pressure singularity at two upper corners (the values are opposite to each other because of the direction of flow) and the streamlines are symmetric with respect to the vertical centerline, as it was expected. This distribution of streamlines is almost the same for all cases except the P1/P1.

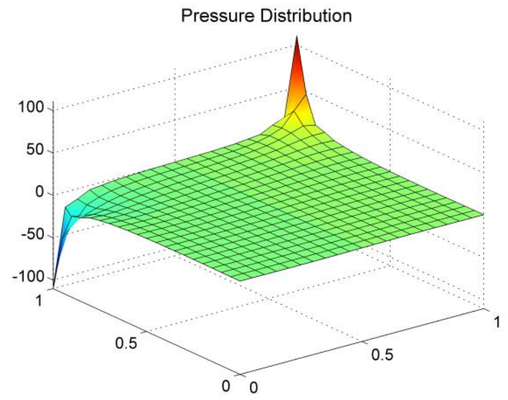
Figure1 and ***Figure2*** in the 3^{rd} row show that for P1/P1 which is not a LBB satisfactory element, we can observe has the anticipated oscillations in pressure and also some additional curves in the streamline.

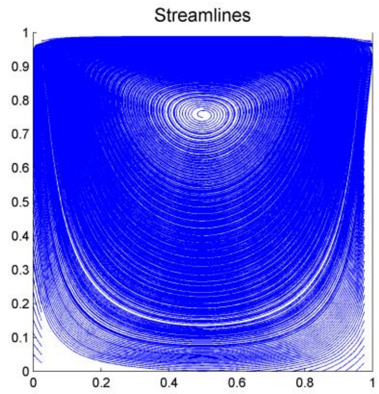
Both the Mini and the Q2Q1, which are LBB compliant, show, reasonable results for pressure, as expected. The Q2/Q0 due to its constant nature for the pressure, shows discontinues pressure distribution.

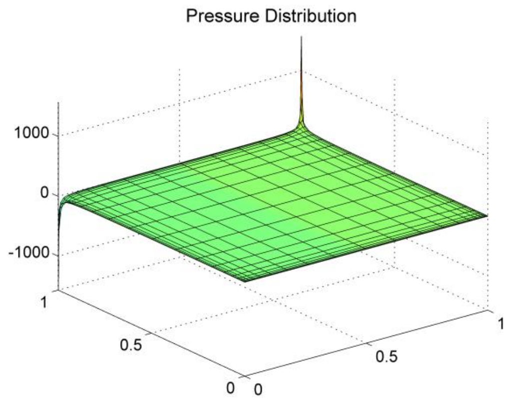
Figue1. Pressure distribution	***Figue2. Streamline***
 **Q2/Q0**	 **Q2/Q0**
 **Q2/Q1**	 **Q2/Q1**
 **P1/P1**	 **P1/P1**
 **MINI P1+/P1**	 **MINI P1+/P1**

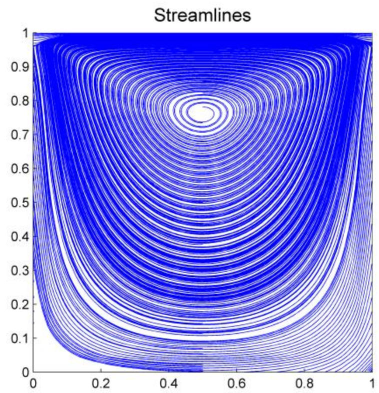
b) Mesh refinement effect

Using the provided code, another analysis is exerted regarding the mesh refinement effects. Namely, for the Q2/Q1, as one of the LBB compliant methods, we have conducted an analysis using 20*20 mesh once in a regular manner and then in a refined shape on sides and corners. We can clearly observe that the pressure jump between two upper corners would be improved if a non-uniform mesh (more refined in corners) is employed, since more emphasis and focus (fine mesh) is put on the places with pressure singularity.

<table>
<tr><th>Figue3. Pressure distribution</th><th>Figue4. Streamline</th></tr>
<tr><td>


Q2/Q1 regular 20*20 mesh</td><td>


Q2/Q1 regular 20*20 mesh</td></tr>
<tr><td>


Q2/Q1 refined 20*20 mesh on sides</td><td>


Q2/Q1 refined 20*20 mesh on sides</td></tr>
</table>

c) GLS stabilization effect

As it is discussed amply in the class and in the reference book, the final matrix form for the GLS stabilized Stokes equation is as following. In this equation two terms are added to the original non stabilized equation system, namely C and g. The expressions for required terms are provided in the table below [reference book equation 6.28]:

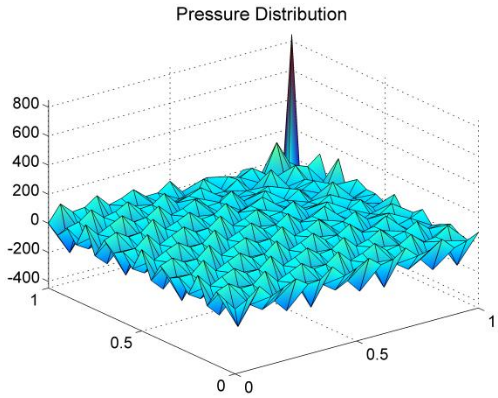
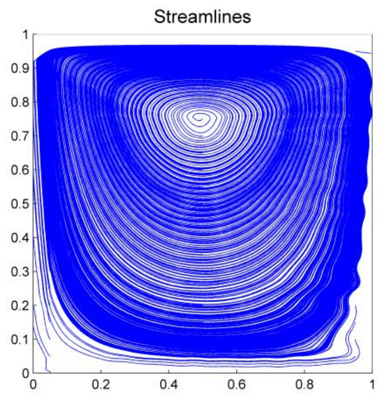
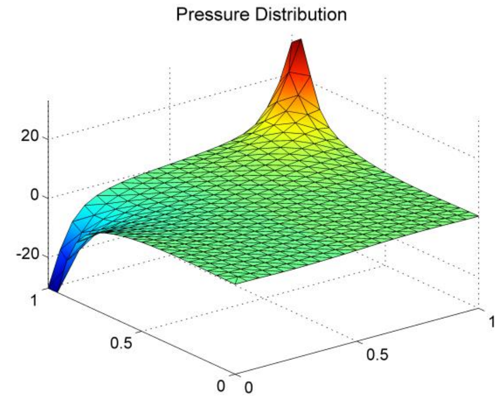
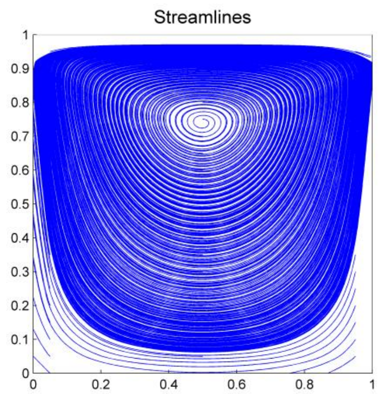
$$\begin{bmatrix} K & G \\ G & -C \end{bmatrix}\begin{bmatrix} V \\ P \end{bmatrix} = \begin{bmatrix} f \\ g \end{bmatrix}$$

K	$\boldsymbol{a(\omega, u) =}$	$\boldsymbol{\int_{\Omega} \nabla\omega : \nu\nabla u \, d\Omega}$	$\boldsymbol{\int_{\Omega e} \nabla N_a . (\nu\nabla N_b) \, d\Omega}$
G	$b(\omega, u) =$	$\int_{\Omega} p \, \nabla . \omega \, d\Omega$	$\int_{\Omega e} N_a . (a . \nabla N_b) \, d\Omega$
C		$\int_{\Omega} \tau \, \nabla q \, . \nabla p \, d\Omega$	$\int_{\Omega} \tau \, \nabla N_i \, . \nabla N_j \, d\Omega$
g		$\int_{\Omega} \tau \, \nabla q \, . \, b \, d\Omega$	$\int_{\Omega} \tau \, \nabla N_i \, . \, b \, d\Omega$

As for the stabilization parameter the formulation provided in the reference book is used:

$$\tau = \alpha_0 \frac{h^2}{4\vartheta} = \frac{1}{3} * \frac{\left(\frac{1}{20}\right)^2}{4 * 1} = \sim 0.0002083$$

As expected, the GLS stabilization method deletes all oscillations regarded the non LBB complaint P1/P1 element type. This fact is studied both in pressure distribution and streamline curves.

Figue5. Pressure distribution	***Figue6. Streamline***
 **P1/P1 non-stabilized**	 **P1/P1 non-stabilized**
 **P1/P1 GLS-stabilized**	 **P1/P1 GLS-stabilized**

3) Navier-Stokes Problem

In this part we are going to solve the Navier-Stokes equations. The script ***mainNavierStokes.m*** can be used to solve this equation adopting Picard method. In order to be able to use it, a Matlab function ***ConvectionMatrix.m*** is written to evaluate the matrix arising from the discretization of the convective term.

$$c(w, v, v^*) = \int_{\Omega} w.(v^*.\nabla)v \, d\Omega$$

The Navier-Stokes equation using a structured mesh of Q2Q1 elements with 15 elements per side is solved (due to some problems with RAM capacity, it was not possible to model 20*20 mesh). 6 Reynolds numbers Re = 100, 200, 500, 1000, 1500 and 2000 are considered for the analysis.

The steady Navier-Stokes solution is characterized by the Reynolds number: $Re = \frac{VL}{\vartheta}$. Where ϑ is the kinematic viscosity of the fluid. The reference velocity is the velocity of the moving side [V = 1] and the reference length is the side length of the cavity [L = 1].

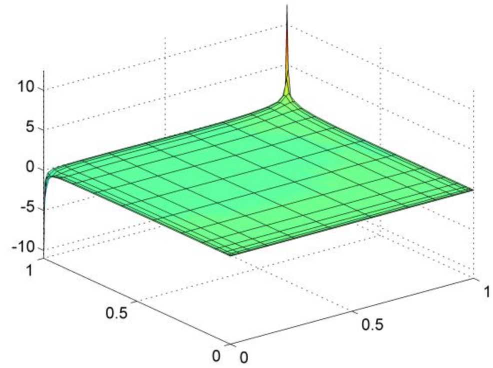
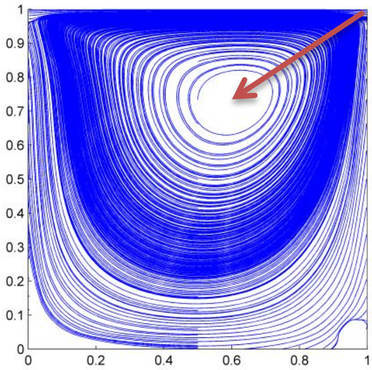
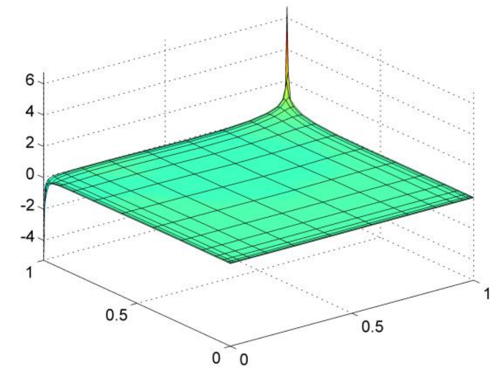
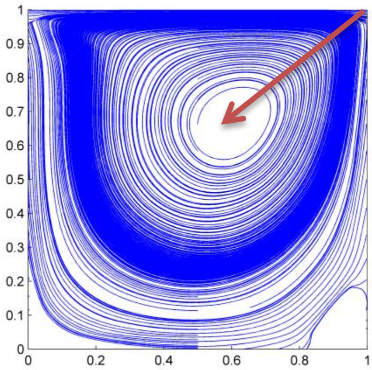
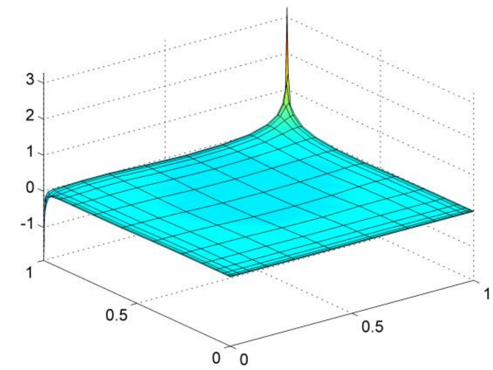
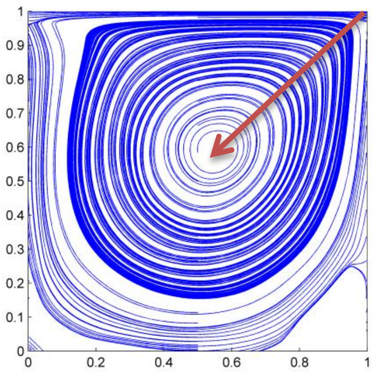
The influence of the Reynolds on the *streamline* can be clearly seen in ***Figure8*** and ***Figure10*** where the position of the main vortex moves towards the center of the cavity (which is highlighted by the red arrow in Figures) when the Reynolds number increases. Moreover the development of second vortex in the right bottom corner of the cavity becomes clear by increasing the Re number (the orange circle in Figure 10).

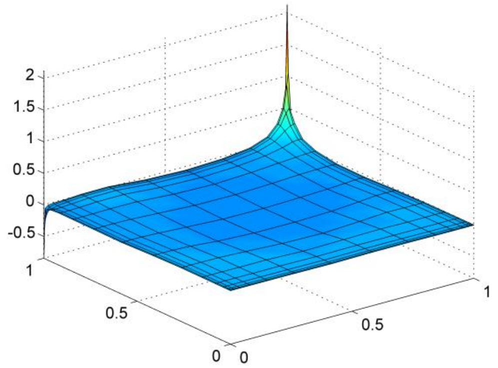
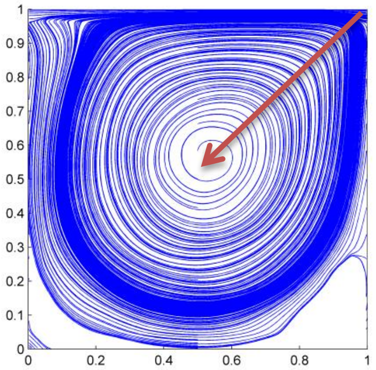
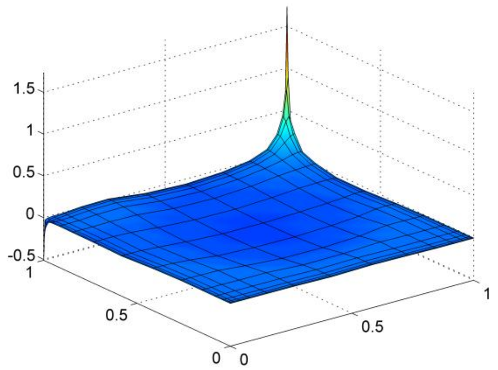
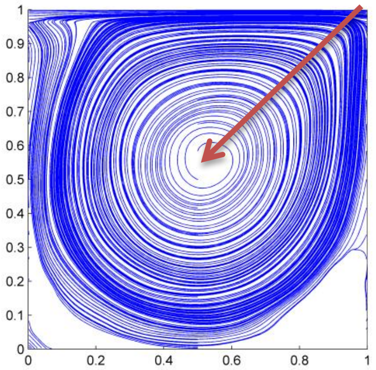
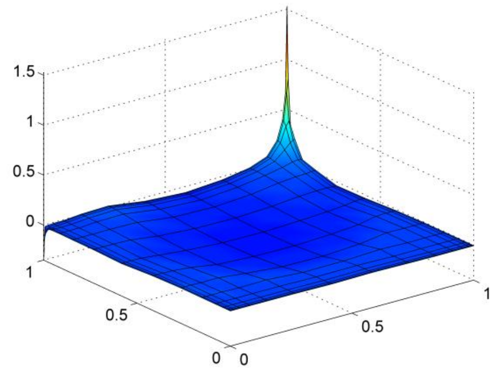
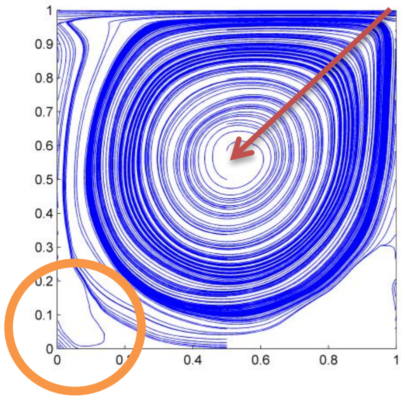
Figure7 and ***Figure9*** also visualize the *pressure* response for the cavity flow, in which the value of jump in pressure in corners converges to a lower quantity but we still have this phenomenon in all Re numbers, as expected due to the nature of problem. We may also capture a slight decrease in the value of pressure in the middle of domain compared to the sides in high Re numbers.

As a conclusion one can say that by increasing the Re number due to the highlighted effect of convection, both the *nonlinear term* and also the natural oscillation due to the *convection dominated flows,* become effective. So, the number of iterations goes higher and the need to use lower Re number's results for capturing higher one's with less effort becomes necessary. Table below provides the number of iteration for each case:

Table. Number of iterations for each case of Navier-Stokes analysis with different Re #

Initial velocity	Re 100	Re 200	Re 500	Re 1000	Re 1500	Re 2000
Non	10	14	21	46	66	-
Re 200	-	-	19	-	-	-
Re 500	-	-	-	39	-	-
Re 1000	-	-	-	-	50	-
Re 1500	-	-	-	-	-	139

Figue7. Pressure distribution	***Figue8. Streamline***
 **Re = 100**	 **Re = 100**
 **Re = 200**	 **Re = 200**
 **Re = 500**	 **Re = 500**

Figue9. Pressure distribution	***Figue10. Streamline***
 **Re = 1000**	 **Re = 1000**
 **Re = 1500**	 **Re = 1500**
 **Re = 2000**	 **Re = 2000**