

Due Tuesday, November 30nd, 12:00 midnight

This challenging but very rewarding homework is considering the finite element analysis of advection-diffusion and incompressible fluid flow problems.

- Problem 1 can be addressed by modifying the 1DBVP MatLab programs to include SUPG stabilization.
- Problems 2 and 3 need the FEM libraries provided here that are based on a stabilized second-order fractional-step method FEM implementation for the solution of incompressible unsteady Navier-Stokes equations (read the provided Readme file for the theory, implementation and examples).
- Finally, Problem 4 requires that you modify the 2DBVP libraries for scalar fields (to include time dependence and advection terms) and couple them with the provided fluid flow libraries in order to address natural convection problems. An overall algorithm for the needed computational approach is provided.

Problem 1 – Advection-Diffusion problems using SUPG (streamline upwind Petrov-Galerkin) methods (MatLab)

Consider a one-dimensional stationary diffusion equation:

$$-\varepsilon \frac{d^2 c}{dx^2} + u \frac{dc}{dx} = 0$$

with boundary conditions as

$$c(0) = 0, c(1) = 1$$

- a) Show that by applying classical Galerkin formulation the solution shows wiggles as long as $\Delta x > \frac{2}{Pe}$, where the Peclet number Pe is defined as $Pe = \frac{u}{\varepsilon}$. Plot the computed solution and the exact solution for $h = \Delta x = 0.1$, $u = 1$ and $\varepsilon = 0.01$. Repeat the calculation for number of nodes 6, 11 and 21 and $\varepsilon = 1, 0.1, 0.005$. For each of these cases tabulate the finite element error in max-norm.
- b) Modify your 1DBVP and repeat the above calculation by considering an SUPG formulation. In this case, the weak form is as follows:

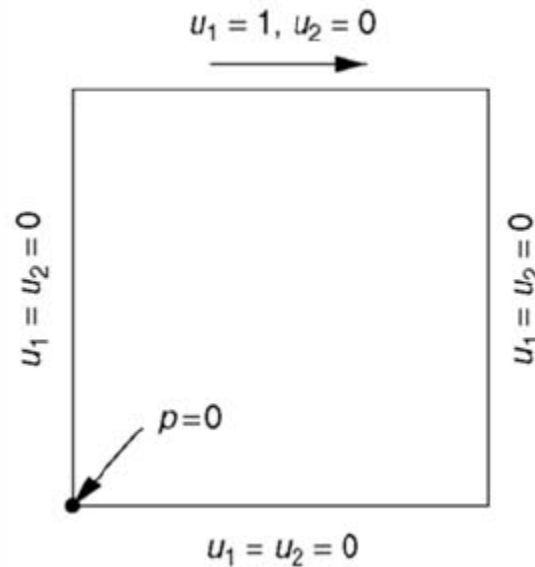
$$\int_{\Omega} \left\{ u \frac{dc_h}{dx} w_h + \varepsilon \frac{dc_h}{dx} \frac{dw_h}{dx} \right\} d\Omega + \sum_{k=1}^{N_e} \int_{\Omega^k} p_h u \frac{dc_h}{dx} d\Omega = 0$$

The stabilization parameter is taken as $p_h = \frac{h \bar{\xi}}{2} \frac{dw_h}{dx}$ where

$$\bar{\xi} = \coth(\alpha) - \frac{1}{\alpha}, \quad \alpha = \text{Element Peclet Number} = \frac{uh}{2\varepsilon}$$

Repeat the calculation from case (a) above ($h=\Delta x=0.1, u=1, \varepsilon=0.01$) and show that the wiggles disappear. Repeat the calculation for number of nodes 6, 11 and 21 and $\varepsilon=1, 0.1, 0.005$ and show the accuracy of the method is now practically $O(h^2)$ for small ε .

Problem 2 – A lid-driven cavity flow (MatLab)



This problem has become a standard benchmark test for incompressible flow. The fluid motion generated in a square cavity by the uniform translation of the upper surface of the cavity is a classical example of recirculating fluid flows in a confined area. From a purely computational view point, the cavity flow is an ideal prototype non-linear problem which is readily posed for numerical solution. Its geometric simplicity, comparatively minor singularity, well-defined boundary conditions and no preferred flow directions makes it very attractive as a benchmark problem for new numerical techniques. The problem definition is shown in the above figure. There is discontinuity in the boundary conditions at the two upper corners of the cavity. Here the two corner points are assumed to belong to the fixed vertical walls (non-leaky). Finally, it should be noticed that Dirichlet boundary conditions are imposed on every boundary in this example. Thus at an arbitrary point, the lower left corner of the cavity, the reference value $p = 0$ is prescribed.

The problem has been studied by many investigators but probably the most detailed investigation was that of Ghia¹ *et al.* in which they quote many solutions and data for different Reynolds numbers. We shall use those results for comparison.

The criterion for convergence to steady-state was generally taken to be

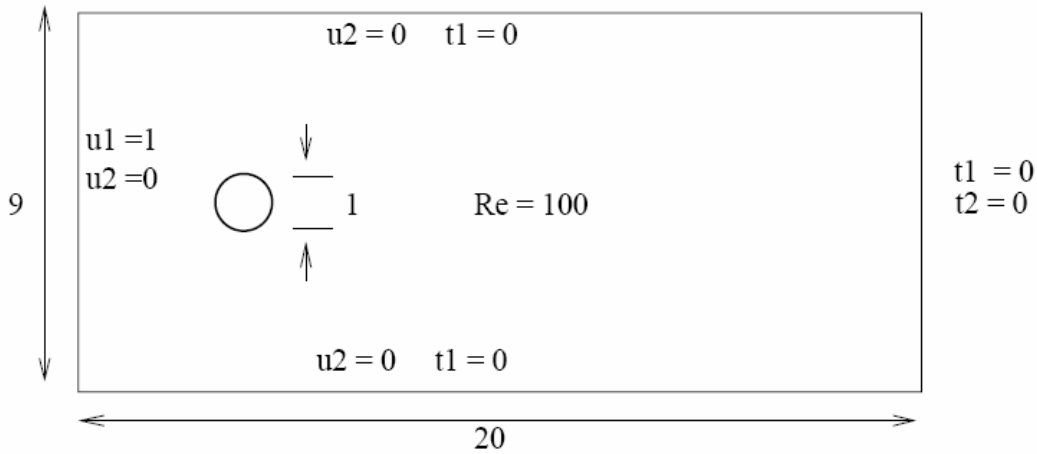
$$\frac{\sum_{i=1}^{\text{No.Nodes}} \sqrt{(|\mathbf{u}|^{n+1} - |\mathbf{u}|^n)^2}}{\sum_{i=1}^{\text{No.Nodes}} \sqrt{(|\mathbf{u}|^{n+1})^2}} \leq 1.0 \times 10^{-4}$$

In this problem, we will use a 64×64 uniform mesh and take $\Delta t = 0.1$. In the example, we have already seen the steady state solutions for $Re = 100$. Now repeat the calculations and compare the steady-state solutions with those of Ghia *et al.* for $Re = 400$ and 1000 .

- (1) The streamline function, pressure and vorticity contour
- (2) The velocity vector plot
- (3) Plot u velocity along the line $x = 0.5$, v velocity along the line $y = 0.5$ and vorticity ω along moving boundary. Compare your results with that listed in Table I, II and IV of the paper Ghia¹.

[1]. U. Ghia, K. N. Ghia and C. T. Shin, "High-resolution for incompressible flow using the Navier-Stokes equations and multigrid method", [Journal of Computational Physics. 48: 387-411, 1982.](#)

Problem 2 – Unsteady flow past a circular cylinder at Reynolds number 100 (MatLab)



Simulation of flow past a circular cylinder is one of the most challenging problems in CFD as unlike other typical numerical tests all the terms in the governing equations are significant in this case. The problem consists of a circular cylinder immersed in a flowing viscous fluid. At Reynolds number below about 40, a pair of symmetrical eddies forms on the downstream side of the cylinder. At higher Reynolds numbers, the symmetrical eddies become unstable and periodic vortex shedding occurs. The eddies are transported downstream, resulting in the well-known Karman vortex street.

A Reynolds number of 100 is considered to be the standard for testing FEM algorithms on the cylinder problem. It is high enough for vortex shedding to occur, but low enough that the boundary layers can be easily resolved.

The domain and the boundary conditions are shown in the figure. The inlet flow is uniform and the cylinder is placed at the centerline between two slip walls. The distance from the inlet and slip walls to the center of the cylinder are 4.5. A no-slip condition is applied on the cylinder surface. In order to simulate the outflow condition, we take $p = 0$ at the outflow (right) boundary. Take time step as 0.05. The initial condition is that of zero velocity.

Initially, a pair of symmetric attached eddies grew behind the cylinder and the flow reach a steady state. At this state, the flow has not developed fully yet, and the flow appears symmetric, with twin vortices forming behind the cylinder. As time progresses, the flow becomes asymmetric and periodic vortex shedding over the cylinder is attained. In the present problem, generate your own mesh from ANSYS. Make sure your mesh is fine enough near the cylinder. Since our code is very stable, you may choose a time step $\Delta t = 0.05$ and run the problem up to $t = 70$.

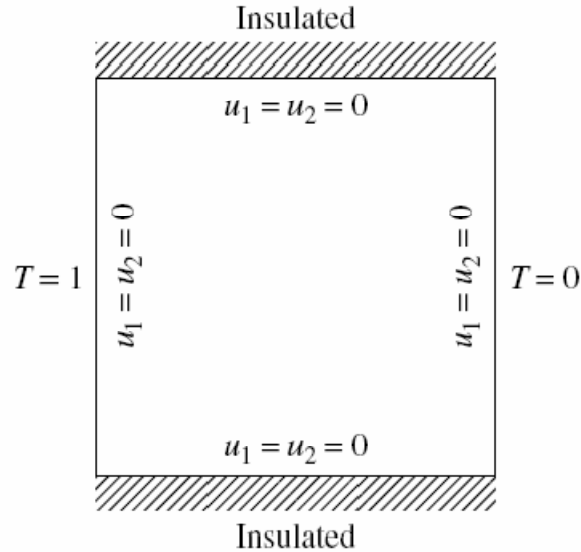
- (1) Plot the time history of vertical v velocity at the mid-point of exit. Identify the steady state at $t=9$ and shedding period τ . Compute the dimensionless shedding frequency, or Strouhal number $S = \frac{D}{u_0 \tau}$, where $u_0 = 1$ in present problem.
- (2) Give the velocity vector plot, pressure, vorticity and streamline contours at the steady state at $t=9$. Clearly identify the symmetric twin vortex behind the cylinder.
- (3) Give the same plots as in (2) during a half period of vortex shedding when the flow has fully developed yet, i.e. at the beginning, $\frac{1}{4}$, $\frac{1}{2}$ of the period. Clearly identify the asymmetric vortex shedding behind the cylinder.

Compare your results with following two literatures or any other papers you can find:

[2]. T. E. Tezduyar, S. Mittal, S. E. Ray and R. Shih, "Incompressible flow computations with stabilized bilinear and linear equal-order-interpolation velocity pressure elements", [Computer Methods in Applied Mechanics and Engineering 95 \(1992\) 221-242](#).

[3]. R. Sampath and N. Zabaras, "A diffpack implementation of the Brooks/Hughes Streamline-upwind/Petrov-Galerkin FEM formulation of the incompressible Navier-Stokes equations", [Research Report MM-97-01](#), Cornell University, March 21. 1997

Problem 3 – Natural convection flow in a cavity (MatLab)



To illustrate the use of the flow in another incompressible flow problem, we shall describe the splitting-up approximate solution of unsteady natural convection problems. Such problems involve a coupling between the Navier-Stokes equations describing the fluid motion and the thermal energy equation governing the space-time evolution of the temperature. The forces which induce natural convection are in fact spatially variable gravity forces generated by (buoyancy) density variations in the fluid due to the non-uniformity of the temperature. The dimensionless equations, in vector form, are

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + \text{Pr} \nabla^2 \mathbf{u} - \text{Pr} Ra T \mathbf{e}_g \quad \text{in } \Omega \times [0, T]$$

$$\nabla \cdot \mathbf{u} = 0 \quad \text{in } \Omega \times [0, T]$$

$$\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T = \nabla^2 T \quad \text{in } \Omega$$

where Pr is the Prandtl number, Ra is the Rayleigh number and \mathbf{e}_g is the unit vector in the direction of gravity.

The problem of interest here is the unit square cavity containing a viscous fluid. The walls are solid and subjected to no-slip velocity boundary conditions (zero-velocity components). One of the vertical walls is subjected to a higher temperature ($T = 1$) than the other vertical wall ($T = 0$). Both the top and bottom walls are assumed to be insulated (zero heat flux). The steady state solution to this problem is sought herein. In this calculation the Prandtl number is kept constant at 0.71 (air) and the problem is solved for four different Rayleigh numbers: $10^3, 10^4, 10^5, 10^6$. Please choose a smaller time step when the Rayleigh numbers increases.

- (1) Develop the finite element code based on the fluid flow solver and heat solver based on the following time discretization:

$$\frac{\mathbf{u}^{n+1} - \mathbf{u}^n}{dt} + (\mathbf{u}^n \cdot \nabla) \mathbf{u}^{n+1} = -\nabla p^{n+1} + \text{Pr} \nabla^2 \mathbf{u}^{n+1} + \text{Pr} Ra T^n \mathbf{e}_g$$

$$\nabla \cdot \mathbf{u}^{n+1} = 0$$

$$\frac{T^{n+1} - T^n}{dt} + \mathbf{u}^{n+1} \cdot \nabla T^{n+1} = \nabla^2 T^{n+1}$$

In other words, first solve the fluid velocity given temperature from last time step using the given fluid solver and then use the velocity at this time step to update the temperature.

- (2) Plot the streamlines, vorticity contours, pressure contours and isothermal lines for each case. Compare your results with those in [4].
- (3) Compare the quantitative results with that of de Vahl Davis^{5,6} in the following table:

Ra	Maximum stream function magnitude	Maximum horizontal velocity	Maximum vertical velocity
10^3	1.170	3.649	3.697
10^4	5.071	16.178	19.617
10^5	9.612	34.73	68.59
10^6	16.750	64.3	219.36

[4]. H. G. Choi, H. Choi and J. Y. Yoo, "A fractional four-step finite element formulation of the unsteady incompressible Navier-Stokes equations using SUPG and linear equal-order element methods", [Comput. Methods Appl. Mechrg](#), 143 (1997) 333-348.

[5] G. deVahl Davis, "Natural convection on air in a square cavity: A benchmark numerical solution", *International Journal of Numerical methods fluids*, 3 (1982) 249-264.

[6] P. L. Quere and T. A. DE Roquefort, "Computation of natural convection in two-dimensional cavities with chebyshev polynomials", [Journal of Computational Physics](#), 57 (1985) 210-228.

Note: The quantitative results of velocity for $Ra=10^4$ and 10^5 are not very accurate since the convection is dominated in this case and we do not have stabilization (i.e. SUPG) for convective terms. However, we can still get very good visualization of velocity and temperature distributions.