Preprint of the paper

"Finite element solution of the Navier-Stokes equation using a SUPG formulation"

P. Vellando, J. Puertas, J. Bonillo, J. Fe (2000)

Advances in computational engineering and sciences. Tech Science Press 2000 pp 856-861.

http://caminos.udc.es/gmni

Finite Element solution of the Navier-Stokes equations using a SUPG formulation.

*P. Vellando, **J. Puertas, **J. Bonillo, *J. Fe.
E.T.S. de Ingenieros de Caminos, Canales y Puertos de La Coruña.
*Dpto. de Métodos Matemáticos. **Dpto. de Tecnología de la Construcción.
Universidad de La Coruña. Campus de Elviña. 15192 La Coruña, Spain.
Tel: (34) 981 167000, FAX: (34) 981 167170.e-mail: vellando@iccp.udc.es

Summary

A Finite Element based program has been released to solve the steady 2D Navier-Stokes equations. The mixed-variable algorithm is used as a first approach to solve the differential problem. In order to reduce the number of equations, both a penalty and segregated techniques are implemented to give solution to the viscous incompressible flow and their results are compared with the former formulation. The program makes use of a SUPG type algorithm as a stabilisation procedure, in order to eliminate the numerical oscillations, which may appear when the boundary conditions force a sudden change in the solution, without necessarily refining the mesh. The three different algorithms are checked making use of the cavity flow benchmark problem and their results are commented.

Governing Equations

The equations used are the steady Navier-Stokes equations, which may be written in the usual indicial notation as follows:

$$u_j u_{i,j} - n\left(u_{i,j} + u_{j,i}\right)_{,j} + \frac{1}{r} p_{,i} = f_i \,; \qquad u_{i,i} = 0 \text{ in } \Omega \qquad (1)$$

where u is velocity, p is pressure, r is the density, n is the cinematic viscosity, f is the body force and the derivation is made with respect to indices after commas. The boundary conditions given to complete the definition of the differential problem are

$$u_i = g_i \text{ in } \Gamma_g \text{, and } s_{ij} n_j = h_i \text{ in } \Gamma_h \qquad (2)$$

FEM Formulation

The method of weighted residuals is applied, therefore the Navier-Stokes equation is multiplied by a weighting function and integrated over the domain Ω

$$\int_\Omega w_i \left(u_{i,t} + u_j u_{i,j} + \frac{1}{r} p_{,i} - n u_{i,jj} - f_i \right) d\Omega = 0 \qquad \int_\Omega q u_{i,i} \, d\Omega = 0 \qquad (3)$$

The consideration of flows with large enough Reynolds numbers, may be the cause of the appearance of the well known 'wiggles' in the resulting velocity field, when using the standard Galerkin formulation. Although this may be suppressed by a severe refinement of the mesh, the usage of a SUPG (Streamline Upwinding / Petrov-Galerkin) algorithm, manages to overcome this difficulty with less computational cost. The SUPG algorithm, makes an upwind weighting of the equations by using weighting functions that differ from trial functions in a term depending on the first derivative of the former. This procedure is the cause of the appearance of an over diffusion in the direction normal to the flow. To overcome this difficulty an artificial diffusion coefficient modifies the newly introduced term. The numerical instability is thus, drastically reduced even for large convective-term-including equations. To implement the SUPG formulation we are going to add to the weighting function a term $\tilde{p}_i$ (see [1]), for the convective terms, where:

$$p_i^h = \frac{\bar{\kappa} u_j^h w_{i,j}^h}{\left\| u^h \right\|^2} \quad \text{with} \quad \bar{\kappa} = \frac{\bar{\xi} u_x^h h_x + \bar{\eta} u_\eta^h h_\eta}{2} \quad \bar{\xi} = \left(\coth \alpha_x - \frac{1}{\alpha_x} \right) \quad \bar{\eta} = \left(\coth \alpha_\eta - \frac{1}{\alpha_\eta} \right)$$

$$\alpha_x = \frac{u_x^h h_x}{2\nu} \qquad \alpha_\eta = \frac{u_\eta^h h_\eta}{2\nu} \qquad u_x^h = e_{xi} u_{ei}^h \qquad u_\eta^h = e_{\eta i} u_{ei}^h \qquad (4)$$

Three different approaches to the problem have been used, the mixed, the penalty and the segregated

Mixed formulation

First, a formulation to solve velocity and pressure in a simultaneous procedure is introduced. Once we have integrated by parts equation (3) and applied the Gauss theorem we obtain the weak expression:

$$\int_\Omega w_i \left(u_j u_{i,j} - f_i \right) + w_{i,j} \nu \left(u_{i,j} + u_{j,i} \right) - w_{i,i} \frac{1}{\rho} p \, d\Omega = \int_{\Gamma_h} h_i w_i \, d\Gamma_h \; ; \int_\Omega q u_{i,i} d\Omega = \int_{\Gamma_h} q u_i n_i d\Gamma_h \; ; (5)$$

Then, the discretization of velocity and pressure in terms of the basic shape functions with respect to a Q_1P_0 (bilinear pressure-constant velocity) elements is carried out

$$u_i \approx u_i^h = \sum_{j=1}^{N} \alpha_i^j N^j \qquad \text{and} \qquad p \approx p^h = \sum_{j=1}^{N} \beta^j q^j \qquad (6)$$

Once the approximation is introduced and the elementary matrices are integrated by the Gauss 2x2point rule, the matrices are assembled to yield.

$$\mathbf{C}(\mathbf{u}) + \nu \mathbf{A}\mathbf{u} + \mathbf{B}\mathbf{p} = \mathbf{f} \qquad \mathbf{B}^T \mathbf{u} = \mathbf{0} \qquad (7)$$

A successive approximation algorithm now linearizes the convective non-linear term, in order to turn the non-linear system of differential equations into a linear system of algebraic equations.

$$C\left(\mathbf{u}^k\right) \approx \mathbf{C}\left(\mathbf{u}^{k-1}\right) u_i^k = \int_\Omega \left(u_j^{k-1} u_{i,j}^k \right) w_i \; d\Omega \qquad (8)$$

Penalty formulation

This alternative method, based upon the Lagrange multipliers theory, gives the possibility of imposing the incompressibility constraint without solving an auxiliary pressure equation by replacing the continuity equation by $u_{i,i} = -\varepsilon P$, where the so-called penalty parameter ε is a number that tends to zero. This equation is incorporated into the dynamic equation and therefore a system of equations that depends on both velocity and pressure is transformed into a velocity-dependant single equation that converges to the fully incompressible problem as ε approaches to zero. By applying this method we can achieve a considerable reduction in the memory requirements. The equation to be solved is now

$$\int_{\Omega_h} w_i^h \left(u_j^h u_{i,j}^h \right) + \nu u_{i,i}^h w_{i,i}^h + \frac{1}{\varepsilon} u_{i,i}^h w_{i,i}^h d\Omega = \int_{\Omega_h} f_i w_i^h d\Omega \qquad (9)$$

and afterward the value of the pressure field can be post-processed by using $p^h = -\frac{1}{e} u^h_{i,i}$. A 'reduced numerical integration' is used to integrate penalty elementary matrices, thus avoiding the 'locking of the solution'. Once the basic element has been chosen and the approximation for u_i is introduced, we can carry out the integration and assembling of the elementary matrices to obtain the 'single matrix equation

$$\mathbf{C}(\mathbf{u})\mathbf{u} + \mathbf{A}\mathbf{u} + \frac{1}{e}\overline{\mathbf{B}}\mathbf{p} = \mathbf{f} \qquad (10)$$

Segregated Formulation

The penalty method succeeds in solving the Navier-Stokes Equations with a large reduction in the execution time and great memory savings, thanks to the smaller number of equations to be solved. Anyhow, this is an approximate method that depends on the election of the parameter e, which for very small values produces an ill conditioning of the stiffness matrix and a certain loss of accuracy and for too large values, may prevent the system from converging. The segregated method calculates velocities and pressures in an alternative iterative sequence, requiring less storing requirements than the conventional mixed method. Moreover, achieves a greater reduction in the number of equations compared to the penalty parameter that is reduced to the number of nodes, and avoids the use of the sometimes-inconvenient penalty parameter. Another gain of these segregated algorithms is that a mixed-order interpolation can be used.

The momentum equations are treated by the weighted residuals finite element method as in the former cases, but this time, the pressure term $\int_\Omega w_i p_{,i} \, d\Omega$ is not considered as an unknown but included in the right hand of the system

$$\mathbf{C}(\mathbf{u},\mathbf{v}) + n\mathbf{A}\mathbf{u} = \mathbf{K}^u\mathbf{u} = \mathbf{f}^u - \int_\Omega w_i \frac{\partial N_j}{\partial x} p_j \, d\Omega \, ; \mathbf{C}(\mathbf{u},\mathbf{v}) + n\mathbf{A}\mathbf{v} = \mathbf{K}^v\mathbf{v} = \mathbf{f}^v - \int_\Omega w_i \frac{\partial N_j}{\partial y} p_j \, d\Omega$$

(11), once we have obtained the velocity field we solve the pressure equation. Rewriting the dynamic equation as:

$$u_i = \tilde{u}_i - \left(K^u \frac{\partial p}{\partial x} \right)_i ; \qquad v_i = \tilde{v}_i - \left(K^v \frac{\partial p}{\partial y} \right)_i \qquad (12)$$

where the pseudo-velocities and the coefficients K_i^u, K_i^v are equal to:

$$\tilde{u}_i = \frac{1}{k_{ii}^u} \left(-\underset{i \neq j}{k_{ij}^u u_j} + f_i^u \right) ; \tilde{v}_i = \frac{1}{k_{ii}^v} \left(-\underset{i \neq j}{k_{ij}^v v_j} + f_i^v \right) ; K_i^u = \frac{1}{k_{ii}^u} \int_\Omega w_i d\Omega \, ; K_i^u = \frac{1}{k_{ii}^u} \int_\Omega w_i d\Omega \, ; (13)$$

and K_i is taken as zero when the velocity is prescribed in the node.

Applying the Galerkin method of weighted residuals to the continuity equation we obtain:

$$\int_\Omega \frac{\partial N_i}{\partial x} N_j u_j + \frac{\partial N_i}{\partial y} N_j v_j \, d\Omega = \int N_i \left(N_j u_j n_x + N_j v_j n_y \right) \, d\Gamma \qquad (14)$$

Which once assembled can be written as:

$$\mathbf{K}^p \mathbf{p} = \mathbf{f}^p \qquad k_{ij}^p = \int_\Omega \frac{\partial N_i}{\partial x} N_k K_k^u \frac{\partial N_j}{\partial x} + \frac{\partial N_i}{\partial y} N_k K_k^v \frac{\partial N_j}{\partial y} \, d\Omega$$

$$f_i^p = \int_\Omega \frac{\partial N_i}{\partial x} N_j \tilde{u}_j + \frac{\partial N_i}{\partial y} N_j \tilde{v}_j \, d\Omega - \int_{\Gamma_h} N_i \left(N_j u_j n_x + N_j v_j n_y \right) \qquad (15)$$

Once we have solved the pressure system, velocities are updated using:

$$u_i = \tilde{u}_i - \frac{1}{k_{ii}^u}\int_\Omega w_i \frac{\partial N_j}{\partial x} p_j \; d\Omega\,; \qquad v_i = \tilde{v}_i - \frac{1}{k_{ii}^v}\int_\Omega w_i \frac{\partial N_j}{\partial y} p_j \; d\Omega\,, \qquad (16)$$

to ensure continuity. Then the dynamic system is again assembled and solved and the same procedure is repeated until convergence is achieved.

Numerical Results

The benchmark problem of the flow in a square cavity with a prescription of unitary velocity on the topside and the no-slip condition on the other sides, has been considered to check the algorithms. The pressure is fixed as zero in the centre of the lower side of the cavity. The domain has been interpolated in terms of a 31x31 node non-regular mesh with Q1/P0 basic elements. We will assume we have reached convergence once $\max_{i=1,\dots N}\left|f_i^n - f_i^{n-1}\right| < 10^{-4}$ for each of the unknowns.

For the mixed formulation a direct Crout algorithm has been used to solve the system of equations with a column profile storing procedure. The results for the pressure and velocity for Reynolds numbers of 1000 and 5000, compared to those of other authors [3] for more refined meshes, are shown bellow.

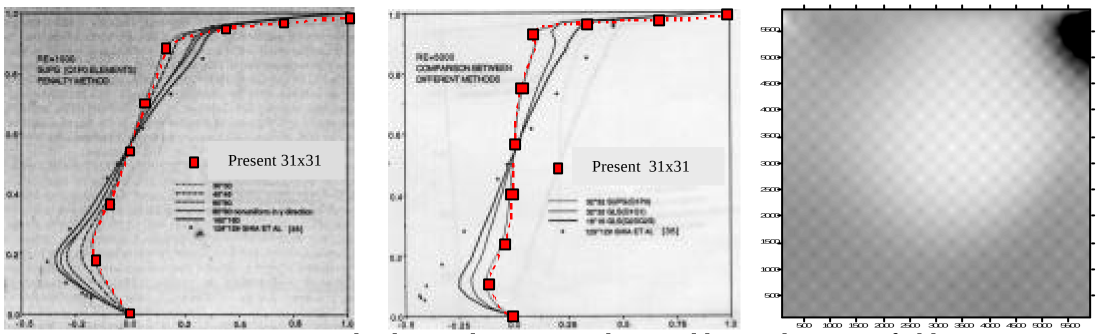

Fig1. - Horizontal velocity along a central vertical line and pressure field.

The results for the penalty algorithm for Reynolds numbers of 100, 1000, 5000 and 10000 are shown bellow. In all the cases considered the penalty parameter has been taken as 10^{-4}. The solution has been obtained using a PBCG iterative method.

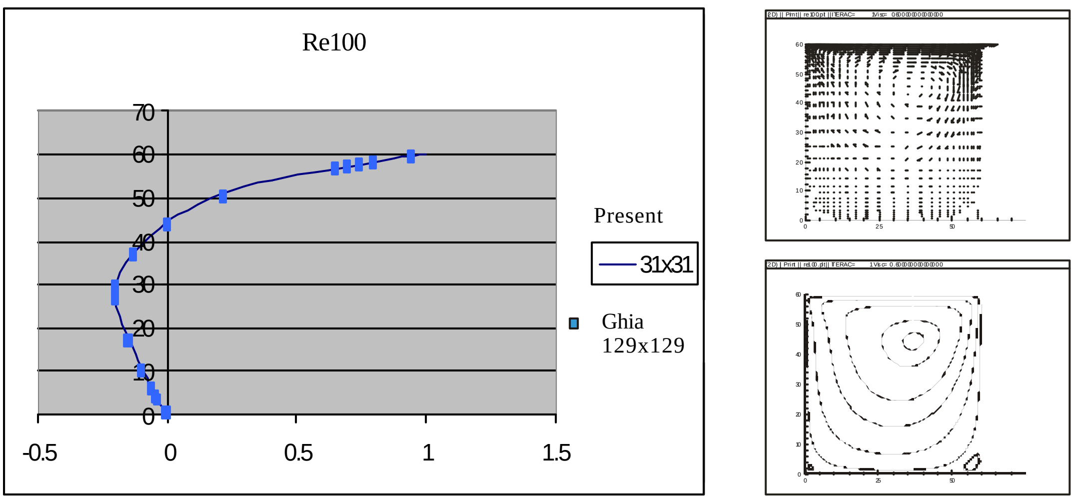

Fig 2. - Horizontal velocities along a central vertical line compared with those of Ghia [2] for a Reynolds number of 100. Velocity field and streamlines

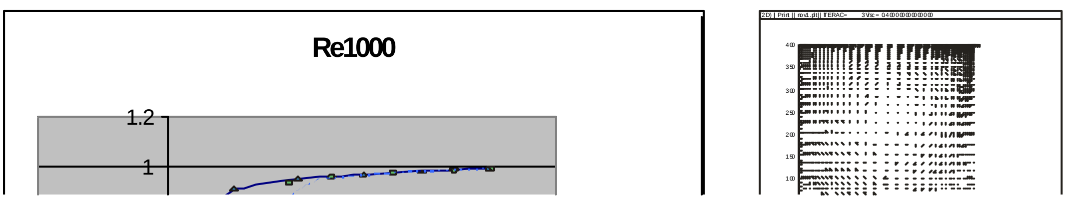

Fig 3. - Horizontal velocities along a central vertical line compared with those of Hannani [3] and Ghia [2] for a Reynolds number of 1000. Velocity field and streamlines.

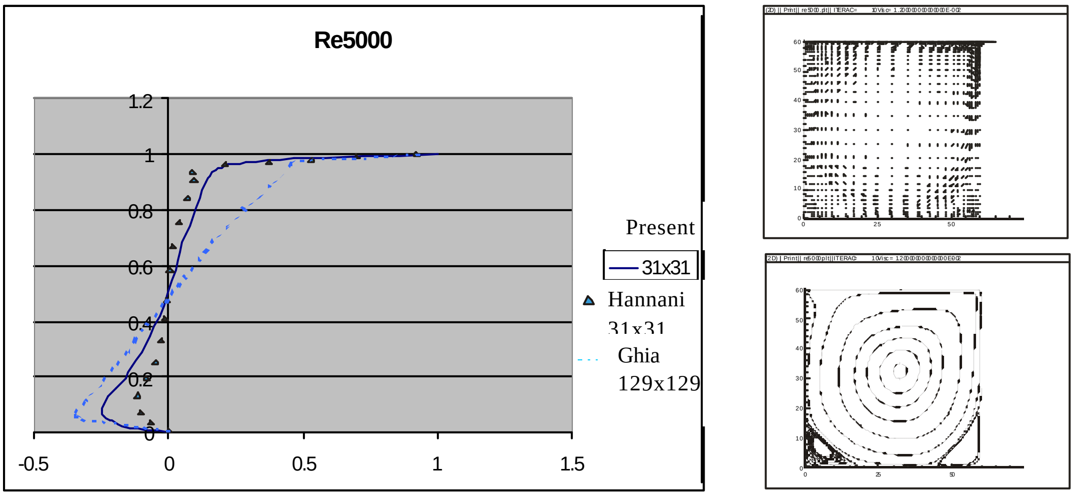

Fig 4. - Horizontal velocities along a central vertical line compared with those of Hannani [3] and Ghia [2] for a Reynolds number of 5000. Velocity field and streamlines.

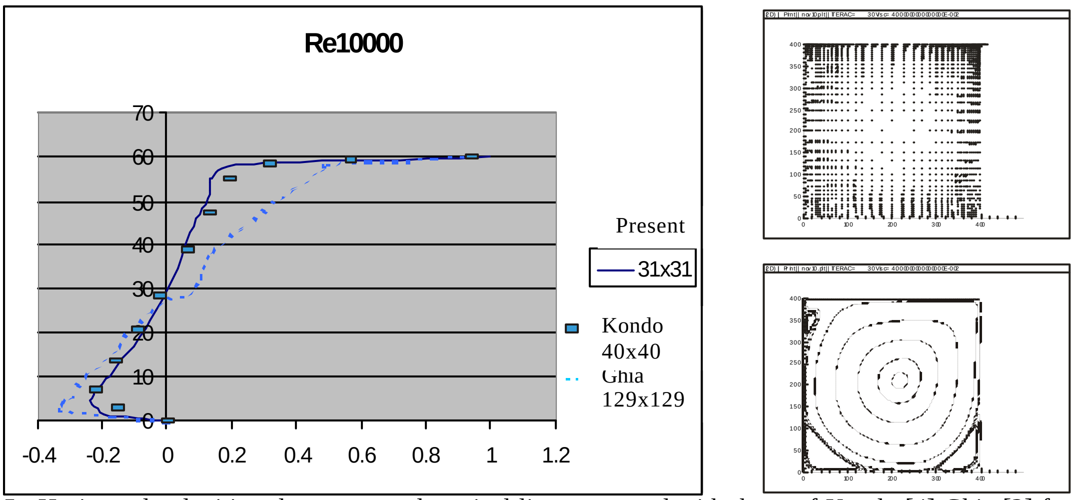

Fig 5.- Horizontal velocities along a central vertical line compared with those of Kondo [4] Ghia [2] for a Reynolds number of 10000. Velocity field and streamlines.

When the **segregated algorithm** is used, an under-relaxation of the unknowns has to be introduced in order of the algorithm to converge. The relaxation parameters used were taken as $a_u = 0.7$ and $a_p = 0.2$, with the relaxation formula $f^n = f^{n-1} + a\left(f^n - f^{n-1}\right)$.

The results obtained for the segregated formulation when a Reynolds number of 400 is used are shown bellow.

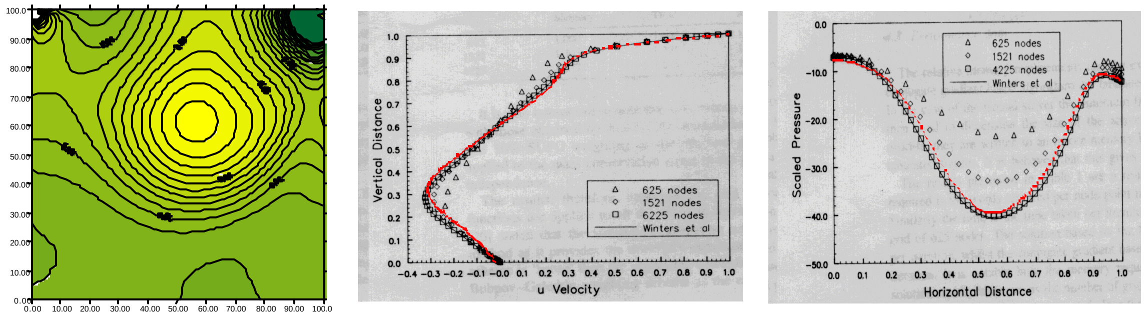

Fig 5. - Pressure field, Horizontal velocities along a central vertical line and Pressures along a horizontal central line compared with those of Winters and du Toit [5].

Conclusions

The program seems to achieve good results for the three formulations as can be seen in the plots, compared with results from Winters, Hannani, Ghia, Kondo and others. The results from the present study seem to adjust to those of the others, with even a less refined mesh. When a mixed formulation is used, the matrices involved in the resolution of the Navier-Stokes equations became large and this implies that very big meshes can not be used, therefore small vortices are not detected. However the iteration process is reduced to the achievement of the convection effect, so a few iterations are needed, and therefore the CPU time involved is less than one hour in a conventional PC. When a mixed or segregated algorithm is used, the iterative process becomes much longer. The program has been run in a Digital AlphaServer 1000A computer, taking CPU times of one or two hours for the 31x31 mesh, depending on the Reynolds number. With respect to the basic elements, when a Q1/P0 basic element is used, the pressure results for the mixed algorithm are polluted by a checker board pressure mode. This unwanted distortion does not appear when an equal order four-node basic element is used for the segregated procedure.

References

1. A. N. Brooks, J. ,R. Hughes. Streamline Upwind/Petrov-Galerkin formulations for convection dominated flows with particular emphasis on the incompressible Navier-Stokes equations. *Comput. Methods Appl. Mech. Engrg 32 (1982) 199-259.*

2. U. Ghia, K. N. Ghia and C.T. Shin. Hgh- Re solutions for incompressible flow using the Navier-Stokes equation and the multigrid method. *Journal of Computational Physics 48, 387-411 (1982).*

3. S. K. Hannani, M. Stanislas and P. Dupont. Incompressible Navier-Stokes computations with SUPG and GLS formulations- A comparison study. *Comput. Methods Appl. Mech. Engrg 124 (1995) 153-170.*

4. N. Kondo. Third-Order upwind finite element formulations for incompressible viscous flow problems. *Comput. Methods Appl. Mech. Engrg. 93 (1991) 169-187.*

5. C. G. du Toit. Finite element solution of the Navier-Stokes equations for incompressible flow using a segregated algorithm. *Comput. Methods Appl. Mech. Engrg. 151 (1998) 131-141.*