Mecánica Computacional Vol. 14, páginas 227-232
compilado por Sergio Idelsohn y Victorio Sonzogni
Editado por AMCA, Santa Fè, 1994

COMPRESSIBLE AND INCOMPRESSIBLE FLOWS WITH EXPLICIT INTEGRATION

S. Idelsohn, M. Storti and N. Nigro
Grupo de Tecnología Mecánica del INTEC, CONICET - U. N. del Litoral
Güemes 3450, 3000- Santa Fe, ARGENTINA
Fax: 54-42-55.09.44, e-mail: rngtm@arcride.edu.ar

ABSTRACT

We present a stabilized algorithm for the incompressible or nearly incompressible Navier-Stokes equations, allowing equal order interpolations. The stabilization terms are obtained as a straightforward extension of the SUPG (for *Streamline Upwind Petrov-Galerkin*) method to multidimensional advective-diffusive system of equations. We show also how the explicit scheme can be used in the incompressible regime through the use of a specially devised preconditioning. Using this techniques, rates of convergence independent of Mach number are achieved and compressible explicit codes can be used in the incompressible limit.

1. INTRODUCTION

Solving flow problems in compressible or incompressible regime has been, historically, a very different matter. There are mainly two difficulties associated to incompressible flows, namely a compatibility condition on the interpolation spaces and bad conditioning of the system matrices. Compressible codes generally use equal order interpolations. If such a compressible code is used in a nearly incompressible flow pattern, i.e. with a Mach number $M << 1$, then oscillations in pressure (*"checkerboard modes"*) are likely to appear. Moreover, this oscillations are likely to occur in locally incompressible regions, like stagnation points, embedded in a globally compressible flows[1]. The subject has been extensively studied by numerical analysts and the conclusion is that the interpolation spaces must satisfy a stability condition named the LBB condition [2,3]. On the other hand, compressible flows can be solved with explicit schemes. If the objective is to reach steady state, then with techniques like *local time stepping* and *absorbing boundary conditions*, an efficient, easy to code and highly paralellizable algorithm requiring a very low amount of core memory is obtained. However, in the limit of incompressible flows the rate of convergence drops like $1/M$ due to bad conditioning, and the strategy becomes unfeasible.

In this work, this two difficulties are addressed. In section §2 we present a stabilized algorithm for the incompressible or nearly incompressible Navier-Stokes equations, allowing equal order interpolations. The stabilization terms are obtained as a straightforward extension of the SUPG (for *Streamline Upwind Petrov-Galerkin*) method to multidimensional advective-diffusive system of equations. In section §3 we show how the explicit scheme can be used in the incompressible regime through the use of a specially devised preconditioning. Using this techniques, rates of convergence independent of Mach number are achieved and the compressible explicit code can be used in the incompressible limit.

2. STABILIZED EQUAL INTERPOLATIONS

Consider the Navier-Stokes equations in conservation form: $\mathbf{F}^i_{a,i} = \mathbf{F}^i_{d,i} + \mathbf{b}$. $\mathbf{U} = [\rho, \rho\mathbf{u}^T, \rho e]^T$, is the fluid local state vector, with $\rho, \mathbf{u}, e$ are the density, velocity and total energy. $\mathbf{F}_a, \mathbf{F}_d \in \mathbb{R}^{5\times 3}$ are the advective and diffusive fluxes, respectively, that depend on the state vector and its gradient as:

$$\mathbf{F}_a(\mathbf{U}) = \begin{bmatrix} \rho\mathbf{u}^T \\ \rho\mathbf{u}\mathbf{u}^T + p\mathbf{I}_{3\times 3} \\ (\rho e + p)\mathbf{u}^T \end{bmatrix} \qquad \mathbf{F}_d(\mathbf{U}, \nabla\mathbf{U}) = \begin{bmatrix} 0 \\ \boldsymbol{\tau} \\ (\boldsymbol{\tau}\cdot\mathbf{u} + \mathbf{q})^T \end{bmatrix} \tag{1}$$

where $p, \boldsymbol{\tau}, \mathbf{q}$ are the thermodynamic pressure, the stress deviatoric tensor and the heat flux vector, respectively. Thermodynamic variables are related through the state equation of the fluid $\theta = \theta(p, \rho)$ and an expression for the internal energy $e = e(p, \rho)$.

2.1. Numerical Spatial Discretization - An SUPG Formulation Overview

The numerical formulation is based on Petrov-Galerkin weighted residual method which allows test functions that can be different from the interpolation ones and not necessarily continuous. This method introduces the numerical dissipation needed to stabilize the system in advection-dominated problems, keeping the consistency with the continuum problem [4]. We suppose that $\mathbf{F}^i_a(\mathbf{U}) = \mathbf{A}_i\mathbf{U}$, and $\mathbf{F}^i_d(\mathbf{U}, \nabla\mathbf{U}) = \mathbf{K}_{ij}\mathbf{U}_{,j}$, where $\mathbf{A}_i$ and $\mathbf{K}_{ij}$ are constant matrices. For each node a there is an interpolation function $\mathbf{N}_a$ (hat type in $1D$, bilinear in $2D$, and multilinear in general) and a test function $\mathbf{W}_a = \mathbf{N}_a + \mathbf{P}_a$, where $\mathbf{P}_a$ is called the perturbation function. The $\mathbf{W}_a$ (and, of course, $\mathbf{P}_a$) are not necessarily continuous through the inter-element boundaries. The variational formulation employed is:

$$\int_\Omega \left(\mathbf{N}_a^T\mathbf{A}_i\mathbf{U}_{,i} + \mathbf{N}_{a,i}^T\mathbf{K}_{ij}\mathbf{U}_{,j}\right)\, d\Omega + \\ + \sum_{e=1}^{N_{\mathrm{el}}} \int_{\Omega_e} \mathbf{P}_a^{eT}\left(\mathbf{A}_i\mathbf{U}_{,i} + \mathbf{K}_{ij}\mathbf{U}_{ij} - \mathbf{b}\right)\, d\Omega = \int_\Omega \mathbf{N}_a^T\mathbf{b} + \int_\Gamma \mathbf{N}_a^T\,\mathbf{h}\, d\Gamma \tag{2}$$

$\mathbf{h}$ is the diffusive flux. It can be shown, by classical integration by parts, that this is a weighted residual formulation and, then, consistency is warranted.

2.2. Extension of Superconvergence to Systems

In the SUPG formulation, the perturbation function is of the form: $\mathbf{P}_a = \mathbf{N}_{a,j}^T\mathbf{A}_j\boldsymbol{\tau}$. The *intrinsic time matrix* $\boldsymbol{\tau}$ controls the amount of numerical diffusion added. For one-dimensional scalar systems it is chosen as: $\tau = (\Delta x/2|a|)\psi(\mathrm{Pe})$, with: $\mathrm{Pe} = u\Delta x/2k$ and $\psi(x) = \coth(x) - 1/x$. ψ is called the *"magic function"*, since it is related to the phenomenon of *"superconvergence"*, i.e. the discrete system gives exact nodal values in certain, very restricted, situations (no source term, constant mesh size).

We extended this result to advective diffusive systems of equations as:

$$\boldsymbol{\tau} = \frac{\Delta x^2}{2}\phi\left(\mathbf{K}^{-1}\mathbf{A}\Delta x\right)\mathbf{K}^{-1}, \quad \phi(x) = \frac{\psi(x)}{x} \tag{3}$$

where ϕ is a closely related, well-behaved "magic" function. As is well known, matrix valued functional expressions are evaluated through an eigenvalue decomposition. This extension to systems of equations preserves the nice feature of "superconvergence". Of course, this expression has sense only if $\mathbf{K}$ is non-singular, which, actually, is not the case for the Navier-Stokes equations.

The problem of the singularity of $\mathbf{K}$ is solved by replacing $\mathbf{K}$ by a regularized non-singular matrix $\mathbf{K}_\epsilon = \mathbf{K} + \epsilon\mathbf{I}$, *only for the sake of computing* $\boldsymbol{\tau}$. The choice of ϵ is $\epsilon = h^2/\nu$ and is discussed in detail in [5]. With this election of the intrinsic time, numerical instabilities due to low Mach number (incompressibility) and high element Reynolds number (advection) are stabilized.

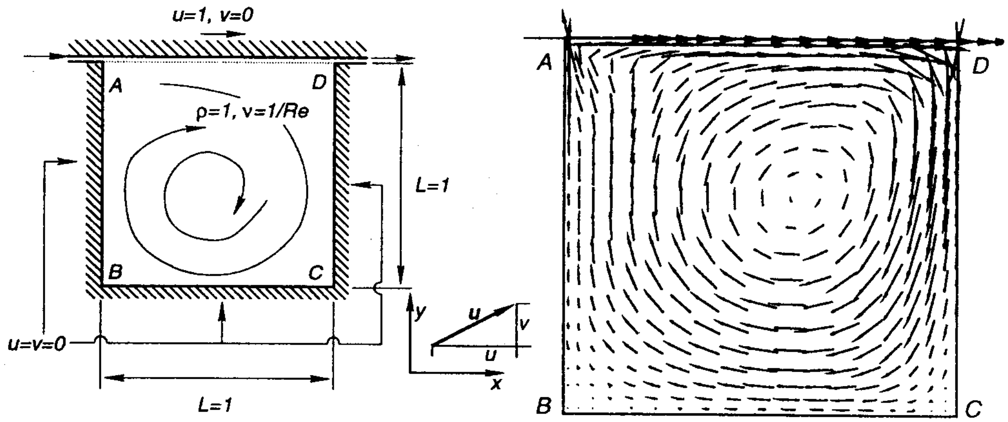

Figure 1: Lid driven cavity flow at Re=1000

2.3. Numerical Results

In figure 1 we show the numerical results for Re = 1000 in the lid-driven cavity flow benchmark. They have been obtained with the simplest $Q1/Q1$ interpolation, stabilized with the scheme described above.

3. INCOMPRESSIBLE FLOWS WITH FULLY EXPLICIT SCHEMES

3.1. Rate of Convergence in 1D Advective Systems

Now we will present a simple example that allows us to understand the main factors affecting convergence rate in advective systems. The analysis will be restricted to the continuum system since the lowest rates are those of the smoother modes. We consider a onedimensional, linear and homogeneous system like:

$$\mathbf{M}\frac{\partial \mathbf{U}}{\partial t} + \mathbf{A}\frac{\partial \mathbf{U}}{\partial x} = 0, \qquad 0 < x < L,\ 0 < t < \infty \tag{4}$$

where $\mathbf{U} \in \mathbb{R}^2$ is the state vector and $\mathbf{A} \in \mathbb{R}^{2\times 2}$ is the advective flux jacobian. We suppose that $\mathbf{A}$ has two eigenvalues $\{a^+, -a^-\}$, with $a^\pm > 0$, and the boundary and initial conditions are: $\mathbf{B}_0\mathbf{U}(0,t) = 0$, $\mathbf{B}_L\mathbf{U}(L,t) = 0$ and $\mathbf{U}(x,0) = \mathbf{U}_0(x)$. $\mathbf{M}$ is a preconditioning matrix, so that $\mathbf{M} = \mathbf{I}$ for the non-preconditioned system. The rate of convergence r.o.c. can be determined by Laplace transform and: r.o.c. $= \mathrm{C}\log_{10}(1/|R_0 R_L|)/[N(1+\kappa)]$ with: $\kappa = a_{\max}/a_{\min}$ and $a_{\max} = \max\{a^+, a^-\}$ and a similar expression for $a_{\min}$. $\mathrm{C} = \Delta t a_{\max}/h$ is the Courant number. $R_{0,L}$ are reflection coefficients at $x = 0, L$. Note that the rate of convergence can be seriously deteriorated if the problem is bad conditioned, i.e. if the maximum and minimum characteristic speeds are very different. This is so for the subsonic ($\kappa \approx 1/\mathrm{M}$) and transonic ($\kappa \approx 2/|\mathrm{M}-1|$)

speeds. It is required also to have good (absorbing) boundary conditions and a stable discrete algorithm (C $\sim$ 1).

For multi-dimensional advective systems, the condition number is defined as $\kappa = |\mathbf{v}_{g\mu}|_{\max}/|\mathbf{v}_{g\mu}|_{\min}$, the sub-index g indicates that *group velocities* are considered[6]. κ is called *group velocity condition number*. The idea is to modify the original system, by including a *preconditioning mass matrix* (PMM) $\mathbf{M} \neq \mathbf{I}$ in such a way that the preconditioned system has a κ much lower than the non-preconditioned one [7].

3.2. The Proposed Preconditioning

The preconditioning we propose here is written [8], in primitive variables $(\rho, \mathbf{u}, p)$, as: $\mathbf{M}_{\mathrm{inc}} = \mathrm{diag}\{2\mathrm{M}, 2\mathrm{M}, 2\mathrm{M}, 1/\mathrm{M}\}$. The corresponding condition number behaves, in the incompressible regime (M $\to$ 0), like $\kappa \to 2$ and the Courant number is bounded from below by 0.78. This means that a rate of convergence independent of Mach number is achieved. Comparison with the artificial compressibility method of Chorin [9,10] is performed in [8]. Surprisingly enough, we have experimentally found that the numerical solutions are very much improved when this preconditioning is used, as will be reported in the numerical results.

3.3. Numerical Results

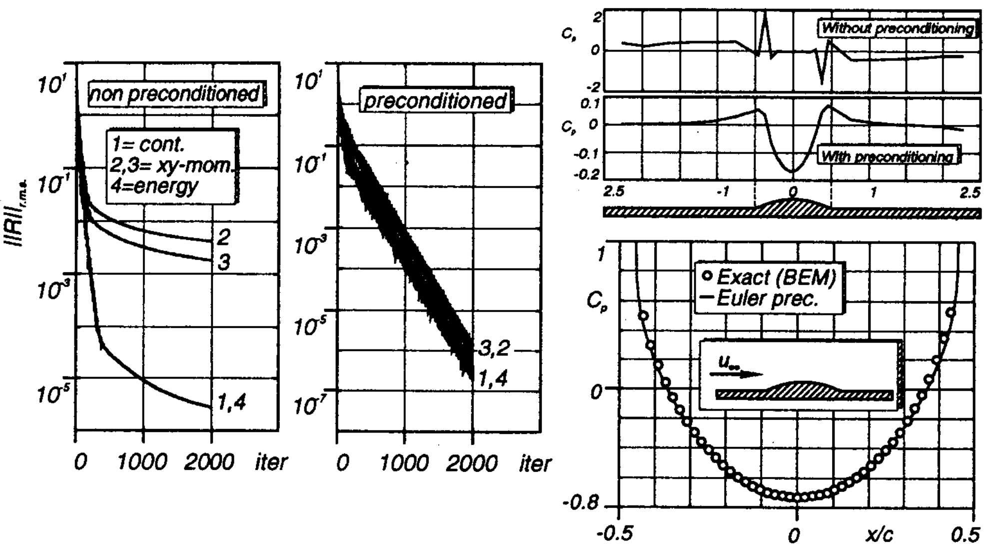

Figure 2: Bump at 12% thickness.

In figure 2 we can see the convergence history for the circular bump (thickness=12%) at $M = 0.001$ with the non-preconditioned scheme (left) and with the preconditioning mass matrix presented in this paper (right). We can appreciate that a significant improvement in the rate of convergence is achieved regardless the very low Mach number, due to the improvement in the condition number. We show also the C_p distribution for the non-preconditioned and preconditioned schemes on a coarse mesh. The non-preconditioned solution is polluted with spurious oscillations, ("checkerboard modes"), whereas the preconditioned one is smooth. We compare also the numerical results obtained with a finer mesh with others obtained with a very accurate BEM (incompressible flow) result that can be taken as the reference. Good agreement is observed.

The second example is a flow around a Joukowski profile (12% thickness, 4.6% camber) with an angle of attack $\alpha = -0.8872$ degrees and a Mach number $M = 10^{-3}$, see figure 3. Again we conclude that the rate of convergence experiments an important improvement with similar rates for each equation. The $-C_p$ distribution is compared against the analytical solution.

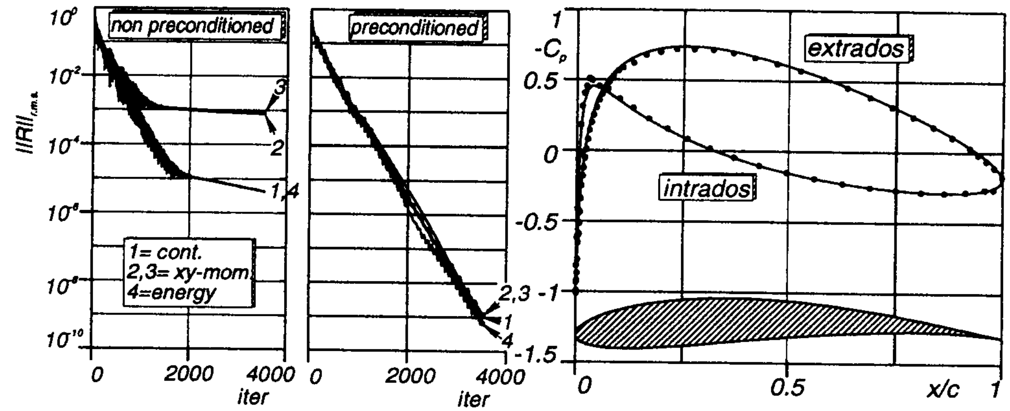

Figure 3: Joukowski profile 12% thickness, 4.6% maximum camber at $M = 0.001$.

ACKNOWLEDGEMENT

The authors wish to express their gratitude to *Consejo Nacional de Investigaciones Científicas y Técnicas* (CONICET, Argentina) for its financial support.

REFERENCES

1. M. Bristeau, R. Glowinski, L. Dutto, J. Periaux and G. Roge, "Compressible viscous flow calculations using compatible finite elements approximations", *Int. J. Num. Meth. Fluids* , vol. 11, pp. 719-749, (1990)

2. T. J. R. Hughes, L. Franca and M. Balestra, "A new finite element formulation for CFD: V. Circumventing the Babuska-Brezzi condition", *Comp. Meth. App. Mech. Engng.* , vol 59, pp 85-99, (1986)

3. S. L. Frey, L. P. Franca and R. Sampaio, "Stabilized methods for the incompressible Navier-Stokes flow", in Num. Meth. in Engng. and App. Sci., by H. Alder *et.al.*, (edts.), CIMNE, (1992).

4. T.J.R Hughes and M.Mallet, "A new finite element method for CFD: IV. A discontinuity-capturing operator for multidimensional advective-diffusive systems", *Comp. Meth. App. Mech. Engng.* 58, (1986), 329-336

5. M. Storti, N. Nigro and S. Idelsohn, "Stabilizing equal-order interpolations for mixed formulations of Navier-Stokes equations via SUPG method", submitted to *Comp. Meth. App. Mech. Engng.* , (1992).

6. R. Vichnevetsky and J. B. Bowles, *"Fourier analysis of numerical approximations of hyperbolic equations"*, SIAM Studies in Applied Mathematics (1982)

7. M. Storti, C. Baumann and S. Idelsohn, "A preconditioning mass matrix to accelerate the convergence to the steady Euler solutions using explicit schemes", *Comp. Meth. App. Mech. Engng.* , vol. 34, pp. 519-541, (1992).

8. M. Storti, N. Nigro and S. Idelsohn, "Steady state incompressible flows using explicit schemes with an optimal local preconditioning", in preparation, (1993).

9. A. Rizzi and L. Eriksson, *J. Fluid Mech.* , vol. 153, pp. 275-312, (1985)

10. J. Farmer, L. Martinelli and A. Jameson, "Fast multigrid method for solving incompressible hydrodynamic problems with free surfaces", *AIAA J.* , **32**, pp. 1175-118, (1994)