INPE – National Institute for Space Research
São José dos Campos – SP – Brazil – May 16-20, 2016

THE CHARACTERISTIC BASED SPLIT SCHEME APPLIED TO SOLVE THE NAVIER-STOKES EQUATIONS.

Gustavo Aparecido Pita Baggio[1], *João Batista Campos Silva*[2], *João Batista Aparecido*[3]

1São Paulo State University, Ilha Solteira, Brasil, gapbaggio@ dem.feis.unesp.br
2São Paulo State University, Ilha Solteira, Brasil, jbcampos@dem.feis.unesp.br
3São Paulo State University, Ilha Solteira, Brasil, jbaparecido@dem.feis.unesp.br

Abstract: In this work is applied the classical method of finite elements, known as Bubnov-Galerkin method (GFEM), applied to simulate two-dimensional incompressible steady and transient flows. The main purpose of this paper is to show how this method associated with the Characteristic Based Split scheme (CBS) deals with the nonlinearity of Navier-Stokes equations. To achieve that, some problems with common geometries was studied and the results show excellent agreement with literature.

keywords: Fluidodynamics, Numerical Simulation, Nonlinear Dynamics in Fluid Sciences, Finite Elements, CBS.

1. INTRODUCTION

The standard Galerkin finite element method (GFEM) gives the minimum error in the L2 norm for self-adjoint problems and results in a symmetric algebraic system of equations. However, the main equations that rule the fluid dynamics are non-self-adjoint equations and if the GFEM is used to solve them, without any stabilization, it may result in several oscillations. The instability due to the non-linear convective acceleration terms, which make the equations non-self-adjoint, leads to a system of non-symmetric equations. [1]

Also, the incompressible limit, Ladyshenskaya-Babuska-Brezzi (LBB) conditions, introduces instability if equal order interpolation functions are used for velocity and pressure fields. In this way, use of simple linear triangular elements results in highly oscillatory solutions when the viscous flows of incompressible fluids is solved using equal order interpolation. The violation of this condition often results in numerically unphysical oscillations in the pressure field. [1]

One way to remedy or reduce the instabilities of the solution is to apply upwind discretization techniques of the convective inertia terms, which are the main source of oscillations when solving this kind of problem. For stabilization via transient formulations, there is the Characteristic-Galerkin, which is the category that the CBS scheme belongs.

In order to apply the Characteristic Galerkin approach to the momentum equations, we have to introduce two steps. In the first step, the pressure term from the momentum equation will be dropped and an intermediate velocity field will be calculated. In the second step, the intermediate velocities will be corrected. This two-step procedure for the treatment of the momentum equations has two advantages. The first advantage is that without the pressure terms, each component of the momentum equation is similar to that of a convection–diffusion equation and the CG procedure can be readily applied. The second advantage is that removing the pressure term from the momentum equations enhances the pressure stability and allows the use of arbitrary interpolation functions for both velocity and pressure. In other words, the well-known Babuska–Brezzi condition is satisfied. Owing to the split introduced in the equations, the method is referred to as the Characteristic Based Split (CBS) scheme. [2]

The characteristic based split (CBS) scheme for compressible and incompressible flow problems was first introduced into the finite element literature in 1995 by Zienkiewicz and Codina and co-workers. The CBS scheme is essentially a fractional time-stepping algorithm, based on an original Finite Differences velocity-projection scheme for solving incompressible flows. Since its introduction to the computational and numerical methods community, the CBS scheme has received great interest and has undergone extensive research for both incompressible and compressible flows. [1]

The CBS scheme has also been extended to investigate other applications such as shallow water flows, thermal and porous medium flows. More recently the extension has included transient flow problems; steady and unsteady turbulent-incompressible flows; problems of viscous-elastic flow; and a special matrix free fractional step method for the Stokes problem has been applied to solve static and dynamic incompressible solid mechanics. All of which, prove the scheme's robustness and applicability for different simulation scenarios. In this thesis some technical investigations are carried out, for improving the results for compressible flow, using the global Galerkin CBS scheme. [1]

2. GOVERNING EQUATIONS

The governing equations of the problem are the continuity equation, the two-dimensional Navier-Stokes equations and the transport equation. For forced convection problems, these equations are commonly non-dimensionalized and they assume following set of equations.

$$\frac{\partial u_i^*}{\partial x_i^*} = 0 \tag{1}$$

$$\frac{\partial \left(u_i^*\right)}{\partial t^*} + \frac{\partial \left(u_j^* u_i^*\right)}{\partial x_j^*} = -\frac{\partial p^*}{\partial x_i^*} + \frac{\partial}{\partial x_j^*}\left(\frac{1}{\mathrm{Re}}\frac{\partial u_i^*}{\partial x_j^*}\right) \tag{2}$$

$$\frac{\partial \left(T^*\right)}{\partial t^*} + \frac{\partial \left(u_j^* T^*\right)}{\partial x_j^*} = \frac{\partial}{\partial x_j^*}\left(\frac{1}{\mathrm{Re}\,\mathrm{Pr}}\frac{\partial T^*}{\partial x_j^*}\right) \tag{3}$$

2.1. The Characteristic Based Split

The application of the CBS scheme for flow equations involves the discretization of the time derivative in steps. Just for remember, as shown in the previous section, the CG procedure is the first step of CBS and we are able to calculating an intermediary velocity field, since the pressure terms be removed from N-S equations, fact that leads to the need of an 'adjust', which happens in the following steps of CBS scheme. In the second step, the pressure field is calculated from a Poisson equation in scheme known as semi-implicit. The pressure equation is derivate from the fact that the intermediate velocities at the first step need to be corrected. When the equations of momentum are subtracted from the original equations with the pressure term and the continuity equation is used, ones obtain the Poisson equation for the pressure field.

With the pressure field in hands, the third step of the CBS scheme is to correct the intermediary velocity field calculated in the first step. Once we get the pressure and the corrected velocities fields, in the last step we can calculate any additional scalar variables with the appropriate governing equation.

The following summarizes the four steps for the time discretization of the equations that make up the CBS scheme from [3].

Step 1: Intermediate velocity field:

$$\frac{\tilde{u}_i - u_i^n}{\Delta t} = -u_j \frac{\partial u_i^n}{\partial x_j} + \nu\left(\frac{\partial^2 u_i}{\partial x_j^2}\right)^n + u_k \frac{\Delta t}{2}\frac{\partial}{\partial x_k}\left[u_j \frac{\partial u_i}{\partial x_j}\right]^n \tag{4}$$

Step 2: Pressure field:

$$\frac{1}{\rho}\left(\frac{\partial^2 p}{\partial x_i^2}\right)^n = \frac{1}{\Delta t}\left(\frac{\partial \tilde{u}_i}{\partial x_i}\right) \tag{5}$$

Step 3: Velocity field correction:

$$\frac{u_i^{n+1} - \tilde{u}_i}{\Delta t} = -\frac{1}{\rho}\frac{\partial p^n}{\partial x_i} + u_j \frac{\Delta t}{2}\frac{\partial}{\partial x_j}\left(\frac{1}{\rho}\frac{\partial p}{\partial x_i}\right)^n \tag{6}$$

Step 4: Temperature field:

$$\frac{T^{n+1} - T^n}{\Delta t} = -u_i \frac{\partial T^n}{\partial x_i} + \alpha\left(\frac{\partial^2 T}{\partial x_j^2}\right)^n + u_k \frac{\Delta t}{2}\frac{\partial}{\partial x_k}\left[u_j \frac{\partial T}{\partial x_j}\right]^n \tag{7}$$

3. RESULTS

3.1. Square Cavity

The flow in the cavity is a widely discussed issue in order to validate the results of the computer codes. The geometry and the boundary conditions of this case are well-known but it can be found in [3] and [2]. Despite the simple geometry, the cavity can present the most complex phenomena of a flow such as recirculation. The irregular mesh contains 87301 nodes and 173576 triangular elements. The results are for Reynolds number equal to 100, 400 and 1000, respectively. Figure 1 shows the velocity component in *x* direction, in the center of the cavity (*y=0,5*).

3.2. Flow around a cylinder

As in previous cases, this is also a case widely studied for the computational code validation. One can find in literature many computational studies and also many experimental studies because it is a relatively easy problem to reproduce in the laboratory. Here we present some studies regarding this case. This geometry is in according with the studies presented by [4] and [5], and the objective is to compare the results of the pressure coefficient over the cylinder surface for Reynolds number equal to 40. The mesh contains 80304 nodes and 159744 elements and the results are compared in Figure 2.

Now, in order to study the transient response, a probe was fixed behind the cylinder at a distance equal to 11 times its diameter. In this case, the time evolution of the flow is shown in Figure 3 for a Reynolds number equal to 100 and time-step equal to 0,001. For comparison with other studies in the literature, the Strouhal number calculated for this case is equal to 0,1676.

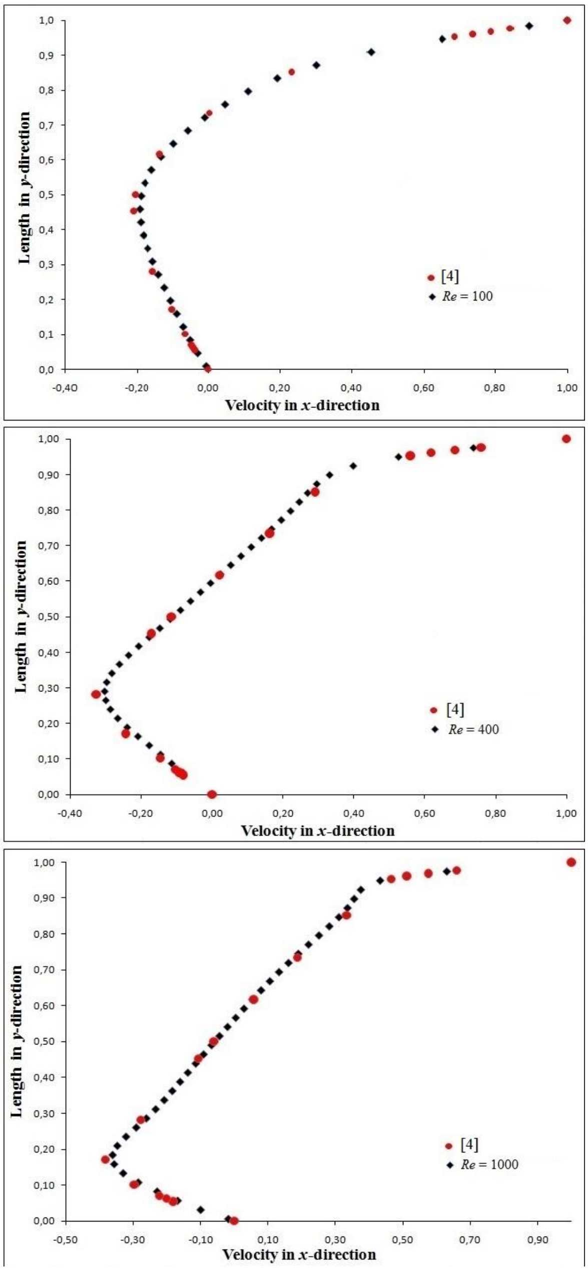

Figure 1 – Results for Reynolds number equal to 100, 400 and 1000, respectively.

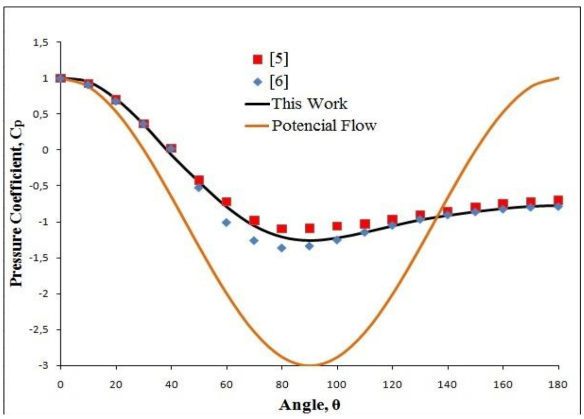

Figure 2 – Results for pressure coefficient over the cylinder surface.

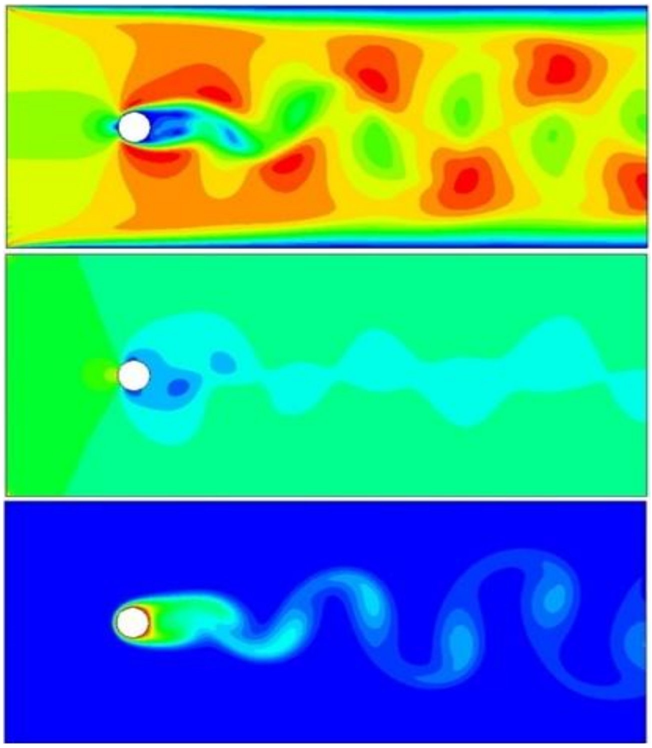

Figure 3 – Velocity magnitude field, pressure field and temperature field, respectively. (*Re*=100)

4. CONCLUSION

In this work, a semi-implicit GFEM-CBS was applied to simulate 2D incompressible fluid flows and the obtained results are in agreement with results from the literature for the benchmark problems with the expected behavior. This work also served as a step for learning more about this numerical method for flow simulations and then to enable a construction of a 3D code. All the results shown in this work were simulated with low and moderate Reynolds numbers, since none turbulence model was applied.

ACKNOWLEDGMENTS

The authors are grateful to the financial support provided by CAPES.

References

[1] Liu, C.B. “The Characteristic Based Split (CBS) scheme for laminar and turbulent incompressible flow Simulations”, Thesis, Civil and Computational Engineering Centre, School of Engineering, University of Wales, Swansea, UK, 2005.

[2] Lewis, R.W., Nithiarasu, P. and Seetharamu, K.N. “Fundamentals of the Finite Element Method for Heat and Fluid Flow”, John Wiley and Sons, Ltd, Chichester, England, 341 p, 2004.

[3] Nithiarasu, P., Codina, R., Zienkiewicz, O. C. “The Characteristic Based Split (CBS) Scheme - a unified approach to fluid dynamics”. International Journal for Numerical Methods in Engineering; Special issue, 2005.

[4] Ghia, U., Ghia, K. N., Shin, C. T. “High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method”. Journal of Computational Physics, vol. 48, pp. 387-411, 1982.

[5] Cheng, L., Armfield, S. “A Simplified Markers and Cell Method for Unsteady Flows on Non-Staggered Grids”. International Journal for Numerical Methods in Fluid, V. 21, p. 15-34, 1995.

[6] Silva, J. B. C., “Simulação Numérica de Escoamentos de Fluidos pelo Método de Elementos Finitos Baseado em Volumes de Controle”. Thesis, Universidade Estadual de Campinas – UNICAMP, Faculdade de Engenharia Mecânica, Campinas, SP, 1998. (*in portuguese*)