

Finite Elements Method With Characteristic Based Split (CBS) Scheme Applied For Incompressible Flow Simulations

Gustavo Aparecido Pita Baggio¹, Jeane Batista de Carvalho², João Batista Campos Silva³

¹Docente de Engenharia Mecânico Instituto Federal de Educação, Ciência e Tecnologia do Mato Grosso do Sul, campus Campo Grande, Rua Taquari, nº 831, Campo Grande-MS, Brasil.

²Docente de Engenharia Mecânica na Universidade Estadual de Santa Cruz - UESC, Rodovia Jorge Amado, Km 16, Ilhéus-BA, Brasil.

³Docente de Engenharia Mecânica Universidade Estadual Paulista - UNESP, Campus Ilha Solteira, Ilha Solteira-SP, Brasil.

¹Corresponding Author

Abstract: This paper presents computational results of incompressible flows simulations by solving the Navier-Stokes equations using a finite element method combined with the Characteristic Based Split scheme (CBS). The classical finite element method called Bubnov-Galerkin or Galerkin Method (GFEM) is considered the best method for solving purely diffusion problems, however, in case of problems with convection dominant as in many problems of fluid flows, the GFEM produces oscillating solutions for high Reynolds numbers. The main goal of this work is to highlight the simulation method and the results of some common 2D problems used for code validation.

Keywords: Finite elements method, Galerkin method, Incompressible flows, CBS.

Date Of Submission: 07-04-2026

Date of Acceptance: 20-04-2026

I. 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 rules 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 [4].

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 [4].

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, [5], [7] and [8] which is the category that the CBS scheme belongs to.

In order to apply the Characteristic-Galerkin approach to the momentum equations, firstly, the pressure term is dropped out and a moving coordinate system is assumed, like a Lagrangian fluid dynamics approach. Although this approach eliminates the convection term responsible for spatial oscillation when discretized in space, the complication of a moving coordinate system is introduced. However, a simple spatial Taylor series expansion in space avoids such a moving coordinate approach. So, we are able to calculate an intermediate velocity field and this is the first step of CBS scheme. In a second step, the pressure field is calculated from a Poisson equation. Then, in a third step, the intermediate velocities will be corrected. With the pressure and the corrected velocities in hand, in the last step we can calculate any additional scalar variables with the appropriate governing equation. Owing to the split introduced in the equations, the method is referred as the Characteristic Based Split (CBS) scheme [3].

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. Since its

introduction to the computational and numerical methods community, the CBS scheme has received great interest and has been subject of study for several researchers for both incompressible and compressible flows [4].

In this work, we use a semi-implicit CBS scheme to solve some 2D incompressible steady flow in a square cavity and a transient flow in a flow around a cylinder. Another objective of the work is to learning some aspects of implementation of the GFEM-CBS in order to simulate in sequence of the work tridimensional flows problems. The Reynolds numbers considered are from low to moderate range in order to keep within the laminar flow range, because none turbulence model is used, and thereby ensure that the results obtained will have a physically expected behavior.

II. NUMERICAL PROCEDURE

2.1 Governing Equations.

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

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

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

$$\frac{\partial (T^*)}{\partial t^*} + \frac{\partial (u_j^* T^*)}{\partial x_j^*} = \frac{\partial}{\partial x_j^*} \left(\frac{1}{\text{RePr}} \frac{\partial T^*}{\partial x_j^*} \right)$$

2.2 The Characteristic Galerkin

Substances Now, the Characteristic Galerkin (CG) procedure will be described for a simple 1D convection-diffusion equation for simplicity. Consider the following transport equation for a scalar variable:

$$\frac{\partial \phi}{\partial t} + u_1 \frac{\partial \phi}{\partial x_1} - \frac{\partial}{\partial x_1} \left(k \frac{\partial \phi}{\partial x_1} \right) + Q = 0$$

The Characteristic Galerkin is based on evaluation of the time derivative along the characteristic that eliminate the convective term in the transport equation (as a Lagrangian fluid dynamics approach). So, the previous equation is now:

$$\frac{\partial \phi}{\partial t} (x_1, t) - \frac{\partial}{\partial x_1} \left(k \frac{\partial \phi}{\partial x_1} \right) = 0$$

Here, although this approach eliminates the convection term responsible for spatial oscillation when discretized in space, the complication of a moving coordinate system is introduced. It is worth to say that it is a self-adjoint equation.

In order to solve the problem of a moving coordinate system, a simple Taylor expansion is used and the semi-discrete form, turns to the following:

$$\frac{\phi^{n+1} - \phi^n}{\Delta t} = -u_1 \frac{\partial \phi^n}{\partial x_1} + u_1^2 \frac{\Delta t}{2} \frac{\partial^2 \phi^n}{\partial x_1^2} + \frac{\partial}{\partial x_1} \left(k \frac{\partial \phi}{\partial x_1} \right)^n$$

At this point, we get to the end of the Characteristic Galerkin procedure and the equation is ready for spatial discretization. In this work, the GFEM is used to spatial discretization.

Once GFEM is used, before proceed with the integration, we must apply Green's lemma to some of the integrals in order to get the weak formulation.

For a 2D convection-diffusion equation, after the Characteristic Galerkin and the GFEM procedures have been applied, we get:

$$\begin{aligned}
 \int_{\Omega} [N]^T [N] \frac{\{\phi\}^{n+1} - \{\phi\}^n}{\Delta t} d\Omega &= -u_1 \int_{\Omega} [N]^T \frac{\partial [N]}{\partial x_1} \{\phi\}^n d\Omega - u_2 \int_{\Omega} [N]^T \frac{\partial [N]}{\partial x_2} \{\phi\}^n d\Omega \\
 &- \int_{\Omega} \frac{\partial [N]^T}{\partial x_1} k_x \frac{\partial [N]}{\partial x_1} \{\phi\}^n d\Omega + \int_{\Gamma} [N]^T k_1 \frac{\partial [N]}{\partial x_1} \{\phi\}^n n_1 d\Gamma \\
 &- \int_{\Omega} \frac{\partial [N]^T}{\partial x_2} k_2 \frac{\partial [N]}{\partial x_2} \{\phi\}^n d\Omega + \int_{\Gamma} [N]^T k_2 \frac{\partial [N]}{\partial x_2} \{\phi\}^n n_2 d\Gamma \\
 &- u_1^2 \frac{\Delta t}{2} \int_{\Omega} \frac{\partial [N]^T}{\partial x_1} \frac{\partial [N]}{\partial x_1} \{\phi\}^n d\Omega + u_1^2 \frac{\Delta t}{2} \int_{\Gamma} [N]^T \frac{\partial [N]}{\partial x_1} \{\phi\}^n n_1 d\Gamma \\
 &- u_1 u_2 \frac{\Delta t}{2} \int_{\Omega} \frac{\partial [N]^T}{\partial x_1} \frac{\partial [N]}{\partial x_2} \{\phi\}^n d\Omega + u_1 u_2 \frac{\Delta t}{2} \int_{\Gamma} [N]^T \frac{\partial [N]}{\partial x_2} \{\phi\}^n n_1 d\Gamma \\
 &- u_2^2 \frac{\Delta t}{2} \int_{\Omega} \frac{\partial [N]^T}{\partial x_2} \frac{\partial [N]}{\partial x_2} \{\phi\}^n d\Omega + u_2^2 \frac{\Delta t}{2} \int_{\Gamma} [N]^T \frac{\partial [N]}{\partial x_2} \{\phi\}^n n_2 d\Gamma \\
 &- u_2 u_1 \frac{\Delta t}{2} \int_{\Omega} \frac{\partial [N]^T}{\partial x_2} \frac{\partial [N]}{\partial x_1} \{\phi\}^n d\Omega + u_2 u_1 \frac{\Delta t}{2} \int_{\Gamma} [N]^T \frac{\partial [N]}{\partial x_1} \{\phi\}^n n_2 d\Gamma
 \end{aligned}$$

Wh

ere n_1 and n_2 are the direction cosines of the outward normal \mathbf{n} , Ω is the domain and Γ is the contour of the domain. The Characteristic Galerkin procedure and the GFEM discretization are described in more details in [3].

2.3 Characteristic Based Split

The 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 [5]:

Step 1: Intermediate velocity field

$$\begin{aligned}
 \frac{\tilde{u}_1 - u_1^n}{\Delta t} &= -u_1 \frac{\partial u_1^n}{\partial x_1} - u_2 \frac{\partial u_1^n}{\partial x_2} + \nu \left(\frac{\partial^2 u_1}{\partial x_1^2} + \frac{\partial^2 u_1}{\partial x_2^2} \right)^n + u_1 \frac{\Delta t}{2} \frac{\partial}{\partial x_1} \left[u_1 \frac{\partial u_1}{\partial x_1} + u_2 \frac{\partial u_1}{\partial x_2} \right]^n + u_2 \frac{\Delta t}{2} \frac{\partial}{\partial x_2} \left[u_1 \frac{\partial u_1}{\partial x_1} + u_2 \frac{\partial u_1}{\partial x_2} \right]^n \\
 \frac{\tilde{u}_2 - u_2^n}{\Delta t} &= -u_1 \frac{\partial u_2^n}{\partial x_1} - u_2 \frac{\partial u_2^n}{\partial x_2} + \nu \left(\frac{\partial^2 u_2}{\partial x_1^2} + \frac{\partial^2 u_2}{\partial x_2^2} \right)^n + u_1 \frac{\Delta t}{2} \frac{\partial}{\partial x_1} \left[u_1 \frac{\partial u_2}{\partial x_1} + u_2 \frac{\partial u_2}{\partial x_2} \right]^n + u_2 \frac{\Delta t}{2} \frac{\partial}{\partial x_2} \left[u_1 \frac{\partial u_2}{\partial x_1} + u_2 \frac{\partial u_2}{\partial x_2} \right]^n
 \end{aligned}$$

Step 2: Pressure Calculation

$$\frac{1}{\rho} \left(\frac{\partial^2 p}{\partial x_1^2} + \frac{\partial^2 p}{\partial x_2^2} \right)^n = \frac{1}{\Delta t} \left(\frac{\partial \tilde{u}_1}{\partial x_1} + \frac{\partial \tilde{u}_2}{\partial x_2} \right)$$

Step 3: Velocity Correction

$$\frac{u_1^{n+1} - \tilde{u}_1}{\Delta t} = -\frac{1}{\rho} \frac{\partial p^n}{\partial x_1} + u_1 \frac{\Delta t}{2} \frac{\partial}{\partial x_1} \left(\frac{1}{\rho} \frac{\partial p}{\partial x_1} \right)^n + u_2 \frac{\Delta t}{2} \frac{\partial}{\partial x_2} \left(\frac{1}{\rho} \frac{\partial p}{\partial x_1} \right)^n$$

$$\frac{u_2^{n+1} - \tilde{u}_2}{\Delta t} = -\frac{1}{\rho} \frac{\partial p^n}{\partial x_2} + u_1 \frac{\Delta t}{2} \frac{\partial}{\partial x_1} \left(\frac{1}{\rho} \frac{\partial p}{\partial x_1} \right)^n + u_2 \frac{\Delta t}{2} \frac{\partial}{\partial x_2} \left(\frac{1}{\rho} \frac{\partial p}{\partial x_2} \right)^n$$

Step 4: Temperature Calculation

$$\frac{T^{n+1} - T^n}{\Delta t} = -u_1 \frac{\partial T^n}{\partial x_1} - u_2 \frac{\partial T^n}{\partial x_2} + \alpha \left(\frac{\partial^2 T}{\partial x_1^2} + \frac{\partial^2 T}{\partial x_2^2} \right)^n + u_1 \frac{\Delta t}{2} \frac{\partial}{\partial x_1} \left[u_1 \frac{\partial T}{\partial x_1} + u_2 \frac{\partial T}{\partial x_2} \right]^n + u_2 \frac{\Delta t}{2} \frac{\partial}{\partial x_2} \left[u_1 \frac{\partial T}{\partial x_1} + u_2 \frac{\partial T}{\partial x_2} \right]^n$$

Now, applying the Galerkin method for spatial discretization, the 4 steps of CBS scheme in matrix form become:

Step 1: Intermediate velocity field

$$[M] \frac{\Delta \{\tilde{u}_1\}}{\Delta t} = -[C] \{u_1\}^n - [K_m] \{u_1\}^n - [K_s] \{u_1\}^n + \{f_1\}$$

$$[M] \frac{\Delta \{\tilde{u}_2\}}{\Delta t} = -[C] \{u_2\}^n - [K_m] \{u_2\}^n - [K_s] \{u_2\}^n + \{f_2\}$$

Step 2: Pressure Calculation

$$[K] \{p\}^n = -\frac{1}{\Delta t} \left[[G_1] \{\tilde{u}_1\} + [G_2] \{\tilde{u}_2\} \right] + \{f_3\}$$

Step 3: Velocity Correction

$$[M] \{u_1\}^{n+1} = [M] \{\tilde{u}_1\} - \Delta t [G_1] \{p\}^n$$

$$[M] \{u_2\}^{n+1} = [M] \{\tilde{u}_2\} - \Delta t [G_2] \{p\}^n$$

Step 4: Temperature Calculation

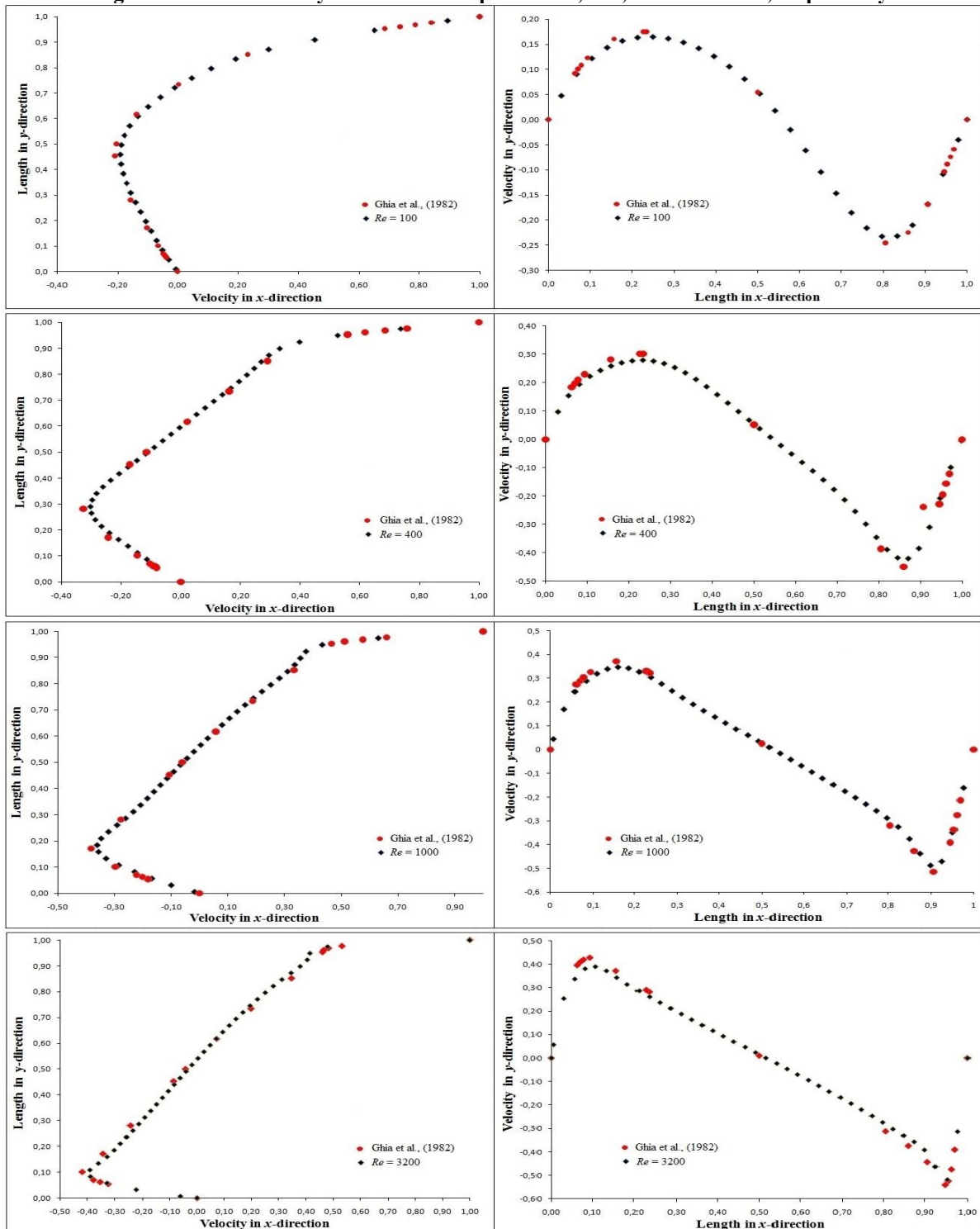
$$[M] \frac{\Delta \{T\}}{\Delta t} = -[C] \{T\}^n - [K_t] \{T\}^n - [K_s] \{T\}^n + \{f_4\}$$

III. RESULTS AND DISCUSSIONS

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 [2] and [3]. 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 Figure 1 shows the results for Reynolds number equal to 100, 400, 1000 and 3200, respectively. For each one, two results are presented. The first on left side show the velocity component in x , in the center of the cavity ($y=0,5$) and the second on the right side, show the velocity component in y direction, also in the center of the cavity ($x=0,5$).

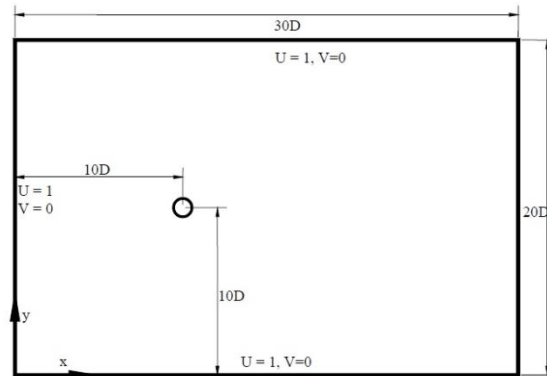
Figure 1. Results for Reynolds number equal to 100, 400, 1000 and 3200, respectively.



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 are presenting some studies regarding this case. This geometry is in according with the studies presented by [1] and [6], 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.

Figure 2. Geometry and boundary conditions for cylinder example.



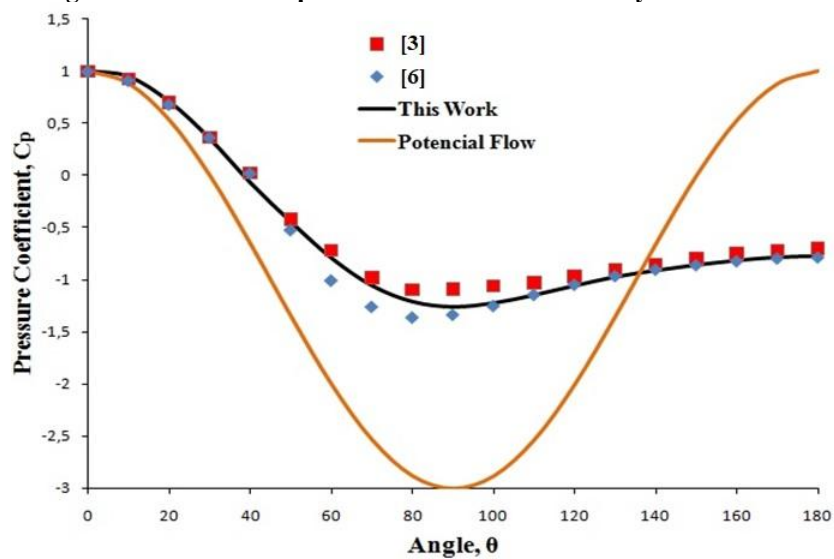
The pressure coefficient is calculated and then the values are normalized using the two following equations. The results are plotted in Fig. 3, where is presented the pressure coefficient value varying the angle θ , which is measured from the stagnation point to the rear of the cylinder.

$$C_p = \frac{p - p_0}{\frac{1}{2} \rho u_\infty^2}$$

$$C_p = 1 + p_i - p_0$$

By inspection of the results, it is possible to observe the excellent agreement of them. In Figure 3 is also show the result predicted using the theory of potential flow by the continuous line.

Figure 3: . Results for pressure coefficient over the cylinder surface.



Now, in order to study the transient response, a different arrangement for the cylinder case is presented in Fig. 4. The mesh generated for this problem has 41916 nodes and 83073 triangular elements. In this case, the time evolution of the flow is shown in Fig. 5 for a Reynolds number equal to 100 and time-step equal to 0,001. The boundary conditions in the cylinder are velocity equal to zero and temperature equals to 1. For comparison with other studies in the literature, the Strouhal number calculated for this case is equal to 0,1676. The Figure 6 presents the values of the y component of velocity measured on a probe located at 11D behind the cylinder center after $t = 60$ i order to have a homogeneous vortex emissions.

Figure 4. Geometry and boundary conditions for the transient analysis.

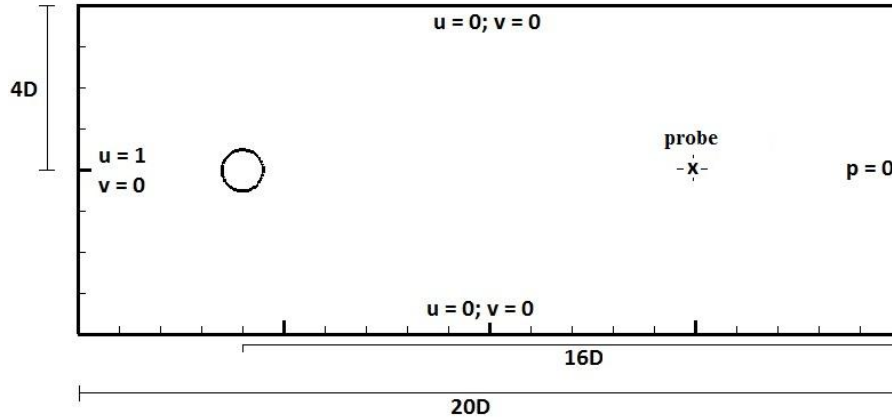


Figure 5. Velocity magnitude field, pressure field and temperature field, respectively. (Re = 100)

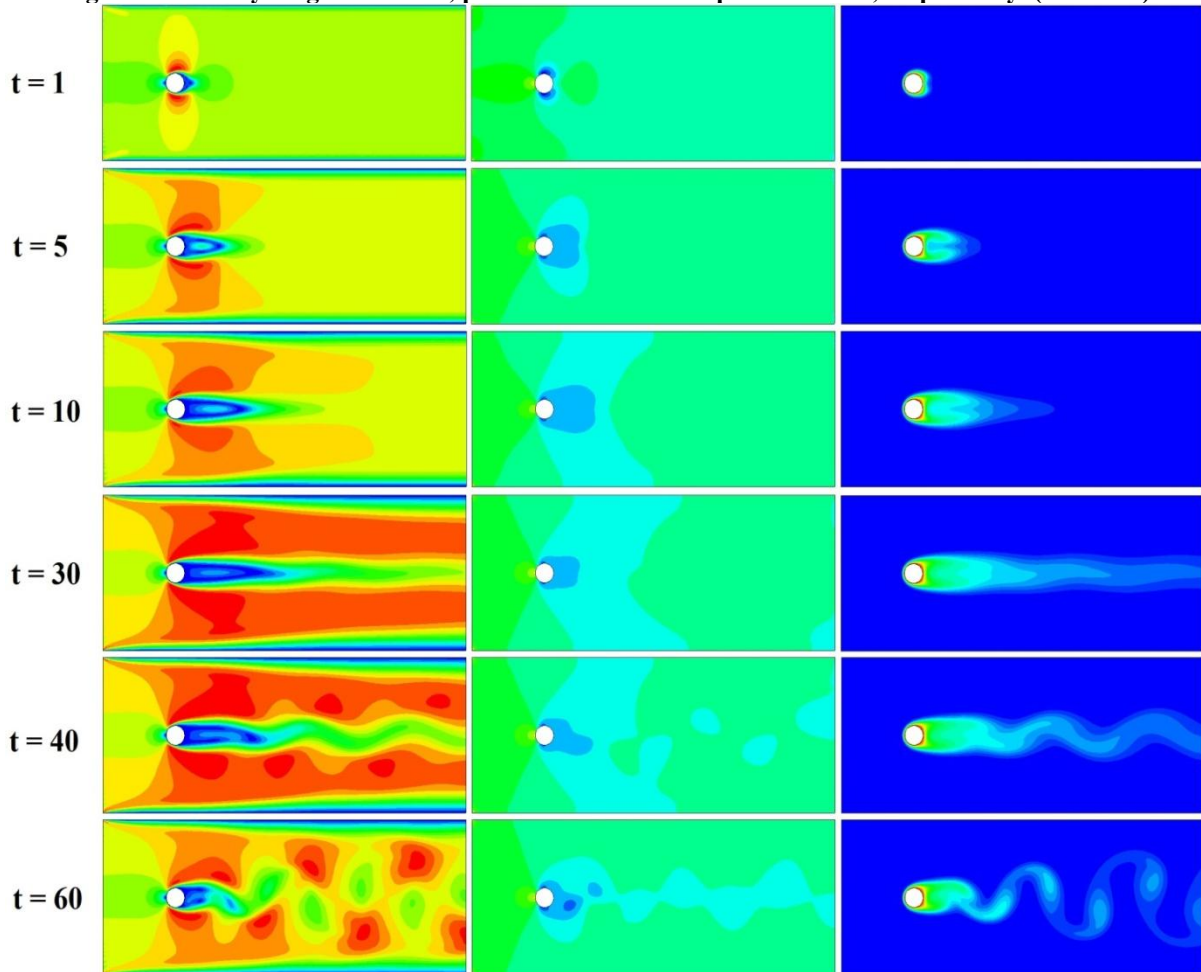
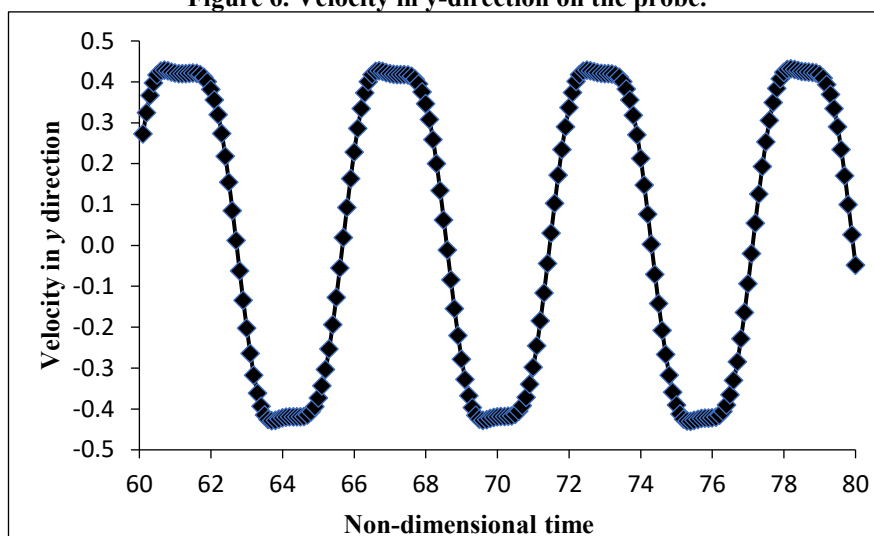


Figure 6. Velocity in y-direction on the probe.



IV. 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 low and moderate Reynolds numbers, since none turbulence model was applied.

Conflict of interest

There is no conflict to disclose.

ACKNOWLEDGEMENT

The authors are grateful to the National Council for Scientific and Technological Development - CNPq

REFERENCES

- [1]. 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.
- [2]. 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.
- [3]. 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.
- [4]. Liu, C.B. *The Characteristic Based Split (CBS) scheme for laminar and turbulent incompressible flows simulations*, Thesis, Civil and Computational Engineering Centre, School of Engineering, University of Wales Swansea, Swansea, UK, 2005.
- [5]. 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.
- [6]. Silva, J. B. C., *Simulação Numérica de Escoamentos de Fluidos pelo Método de Elementos Finitos Baseado em Volumes de Controle*. Tese de Doutorado. Universidade Estadual de Campinas – UNICAMP, Faculdade de Engenharia Mecânica, Campinas, SP, 1998. (in portuguese)
- [7]. Zienkiewicz O. C., Nithiarasu, P., Codina, R., Vazquez, M., Ortiz, P. The characteristic-based-split procedure: an efficient and accurate algorithm for fluid problems. *International Journal for Numerical Methods in Fluids*; 31(1):359-396, 1999.
- [8]. Zienkiewicz, O. C, Taylor, R. L. *The Finite Element Method vol. 3 Fluid Dynamics*, 5th edition. Butterworth and Heinemann, Oxford, 334 p., 2000.