AN ANALYSIS OF THE CONTROL VOLUME-FINITE ELEMENT METHOD WITH THE TIME SPLITTING TECHNIQUE FOR THE NUMERIC SIMULATION OF FLUID FLOWS

Marco Donisete de Campos
Universidade Federal de Mato Grosso - *Campus* de Sinop
Av. Brasília, 1200 - 78550-000 - Sinop, MT, Brasil
mcampos@ufmt.br

João Batista Campos Silva
Universidade Estadual Paulista - Faculdade de Engenharia
Caixa Postal 31 - 15385-000 - Ilha Solteira, SP, Brasil
jbcampos@dem.feis.unesp.br

Abstract. *The main purpose of this work is the numerical computation of incompressible fluid flows by Control Volume-Finite Element Method (CVFEM)***.** *The domain is discretized by using finite elements and the equations are discretized into control volumes around the nodes of the finite elements. The time discretization of the equations is done by using a fractional step. The flow equations are filtered by computating the large eddies and the small eddies are modeled by Smagorinsky's eddy viscosity model for the sub-grid-scale stresses. The two-dimensional benchmark problem of the lid-driven cavity flow is solved to validate the numerical code and the preliminary results are presented and compared with available results from the literature.*

Keywords*: Fractional step, Finite element method, Turbulent fluid flow.*

1. INTRODUCTION

In recent years, numerical solution of the incompressible Navier-Stokes equations there has been a considerable research effort invested in the design of high resolution and requires discretization in both space in time. With the growing confidence in these methods and increased computer power, these schemes are increasingly being applied to the solution of viscous flow problems.

Fractional step methods developed for the Navier-Stokes equations have been investigated since the pionners works of Chorin (1968a-b, 1969) and Teman (1969a-b). In this method the Navier-Stokes equations in time at each time-step by first solving the momentum equations using an approximate pressure field to yield an intermediate velocity field that will not, in general, satisfy continuity. A Poisson equation is then solved with the divergence of the intermediate velocity as a source term to provide a pressure or pressure correction, which is then used to correct the intermediate velocity field, providing a divergence free velocity. The pressure is updated and integration then proceeds to the next time step.

In this paper two methods for the solution of incompressible Navier-Stokes equations is outlined. In the first, called Method I (Ramaswmany *et al*., 1992) in the first step, the intermediate velocity was advanced in an explicit way. The boundary condition for the solution of the Poisson equation of the pressure was of first type, that is, null pressure in a die point. In the other, called Method D* (Kim and Lee, 2002), in the first step, the diffusive term was discretized in an implicit way and the boundary condition for solution of the Poisson equation of the pressure was of second type, that is, the free gradient of pressure at boundaries.

2. PROBLEM FORMULATION

2.1. Governing equations

In this section is to briefly the fluid flow equations considered for solution, with an emphasis on the physical aspects that are important when establishing finite element models. Mathematically, the laminar or turbulent flows may be expressed by the mass conservation and momentum equations that can be written in indicial notation, as follows:

$$\frac{\partial(r u_i)}{\partial x_i} = 0, \tag{1}$$

$$\frac{\partial(r u_i)}{\partial t} + \frac{\partial(r u_j u_i)}{\partial x_j} - \frac{\partial}{\partial x_j}\left(m \frac{\partial u_i}{\partial x_j}\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(m \frac{\partial u_j}{\partial x_i}\right) + S_{u_i} \tag{2}$$

where *i, j=1,2* and u_i are the velocity components along the Cartesian co-ordinates x_i; p is the pressure; r is the fluid density; m is the fluid dynamic viscosity and S_{u_i} is a source term accounting for the other terms not appearing explicitly in Eq. (2). Although needless, in the Eq. (1), the density was left within of partial derivative in order to facilitate the numerical program implementation.

2.2. Dimensionless equations and boundary condition

Defining the dimensional variables with the asterisk as superscript, the dimensionless variables can be written as $X_i = \frac{x_i}{L*}$; $U_i = \frac{u_i}{u_0}$; $P = \frac{p - p_0}{r_0 u_0^2}$; $t = \frac{t^* u_0}{L}$; $r = \frac{r^*}{r_0}$; $m = \frac{m^*}{m_0}$; $\mathrm{Re} = \frac{r_0 u_0 L}{m_0}$, where L is a characteristic length; u_0 a characteristic velocity; p_0 a reference pressure; r_0 a reference density, Re the Reynolds number of the flow under consideration and m_0 the reference absolute viscosity. Therefore, after filtering the equations (1) and (2) they can be rewritten in dimensionless form as

$$\frac{\partial (r U_i)}{\partial X_i} = 0, \tag{3}$$

$$\frac{\partial (\rho U_i)}{\partial t} + \frac{\partial (\rho U_j U_i)}{\partial X_j} - \frac{\partial}{\partial X_j}\left(\mu_e \frac{\partial U_i}{\partial X_j}\right) = -\frac{\partial P}{\partial X_i} + \frac{\partial}{\partial X_j}\left(\mu_e \frac{\partial U_j}{\partial X_i}\right) + S_{u_i} \tag{4}$$

The effective viscosity m_e is defined by

$$m_e = \begin{cases} \frac{m}{\mathrm{Re}} & \textit{for laminar flows} \\ \frac{m}{\mathrm{Re}} + m_t & \textit{for turbulent flows} \end{cases} \tag{5}$$

where the eddy viscosity, m_t, can be calculated using the LES - Large Eddy Simulation to numerical simulation of turbulent fluid flows.

The boundary conditions for solving equations (3) and (4) are null velocities (non slip condition) at walls and for pressure it is necessary only to fix a value at a single point in the domain since the flow is incompressible.

3. DEVELOPMENT OF THE ALGORITHM

3.1 Time and spatial discretization

Using the time-splitting method (Ramaswamy et al., 1992), the Eq. (3) and Eq. (4), can be rewritten as:

$$\frac{\partial (r U_i^*)}{\partial t} + \frac{\partial (r U_j^* U_i^*)}{\partial X_j} = \frac{\partial}{\partial X_j}\left(m_e \frac{\partial U_i^*}{\partial X_j}\right) + F_i, \tag{6}$$

$$\frac{\partial (r U_i)}{\partial t} = \frac{\partial P}{\partial X_i}, \tag{7}$$

$$\frac{\partial (r U_i)}{\partial X_i} = 0 \tag{8}$$

where F_i is a source term.

The intermediate velocity U_i^* can be achieved by the equations:

$$\frac{rU_i^*}{\Delta t} - \frac{rU_i^n}{\Delta t} + q^* \frac{\partial (rU_j U_i)^n}{\partial X_j} + \left(1 - q^*\right) \frac{\partial (rU_j U_i)^{n-1}}{\partial X_j}$$
$$= q \frac{\partial}{\partial X_j}\left(m_e \frac{\partial U_i^*}{\partial X_j} \right) + (1-q)\left(m_e \frac{\partial U_i}{\partial X_j} \right)^n + q^{**} F_i^{n+1} + (1-q^{**}) F_i^n \quad (9)$$

$$\frac{rU_i^{n+1}}{\Delta t} - \frac{rU_i^*}{\Delta t} = -\frac{\partial P^{n+1}}{\partial X_i} \quad (10)$$

$$\frac{\partial (rU_i)^{n+1}}{\partial X_i} = 0 \quad (11)$$

where q is a parameter of time discretization. Note that U_i^* do not satisfies the continuity equation.

By applying the divergence in Eq. (10) and using the Eq. (11), we obtain

$$-\frac{\partial}{\partial X_i}\left(\frac{\partial P^{n+1}}{\partial X_i} \right) = -\frac{1}{\Delta t} \frac{\partial \left(rU_i^* \right)}{\partial X_i} . \quad (12)$$

In this way, we can compute the flow in following three steps:

Step 1

$$\frac{rU_i^*}{\Delta t} - q \frac{\partial}{\partial X_j}\left(m_e \frac{\partial U_i^*}{\partial X_j} \right) = \frac{rU_i^n}{\Delta t} - q^* \frac{\partial (rU_j U_i)^n}{\partial X_j} - \left(1 - q^*\right) \frac{\partial (rU_j U_i)^{n-1}}{\partial X_j}$$
$$+ (1-q)\left(m_e \frac{\partial U_i}{\partial X_j} \right)^n + qF_i^* + (1-q \;) F_i^n \quad (13)$$

Step 2

$$\frac{\partial}{\partial X_i}\left(\frac{\partial P^{n+1}}{\partial X_i} \right) = \frac{1}{\Delta t} \frac{\partial \left(rU_i^* \right)}{\partial X_i} \quad (14)$$

Step 3

$$\frac{rU_i^{n+1}}{\Delta t} = \frac{rU_i^*}{\Delta t} - \frac{\partial P^{n+1}}{\partial X_i} . \quad (15)$$

In CVFEM, the spatial discretization is carried from variational formulation or weak form, obtained taking the scalar product of equations terms by weighting functions that are unity constants inside control volume around the nodes of the finite element mesh.

Therefore, integrating by parts the Eq. (13), (14) and (15), we obtain, respectively:

$$\int_V \left(\frac{rU_i^*}{\Delta t} \right) dV - \oint_A q \left(m_e \frac{\partial U_i^*}{\partial X_j} \right) n_j dA = \int_V \left(\frac{rU_i^n}{\Delta t} \right) dV - \oint_A q^* \left(rU_j U_i \right)^n n_j dA \quad (16)$$
$$- \oint_A \left(1 - q^*\right)\left(rU_j U_i \right)^{n-1} n_j dA + \oint_A (1-q)\left(m_e \frac{\partial U_i}{\partial X_j} \right)^n n_j dA + \int_V qF_i^* dV + \int_V (1-q \;) F_i^n dV$$

$$\int_A \left(\frac{\partial P^{n+1}}{\partial X_i} \right) n_i \, dA - \int_V \frac{1}{\Delta t} \left(\frac{\partial \left(r U_i^* \right)}{\partial X_i} \right) dV = 0 \; . \tag{17}$$

$$\int_V \frac{r U_i^{n+1}}{\Delta t} dV = \int_V \frac{r U_i^*}{\Delta t} dV - \int_V \frac{\partial P^{n+1}}{\partial X_i} dV \; . \tag{18}$$

where n_j is the outward normal vector to the area of a control volume where there are convective and diffusive fluxes. This normal vector has been defined as $\vec{n}dS = dy\vec{i} - dx\vec{j}$ for integration in the counterclockwise direction.

3.2 Finite -element formulation

In FEM, the unknown variables can be interpolated, inside a Lagrange element, in the following form

$$U_i^e(\Omega,t) = \sum_{a=1}^{NNEL} N_a^e(\Omega) \overline{U}_{ia}^e(t) \; . \tag{19}$$

$$P^e(\Omega,t) = \sum_{a'=1}^{NNEL} N_{a'}^e(\Omega) \overline{P}_{a'}^e(t) \, . \tag{20}$$

where N_a e $N_{a'}$ are the two-dimensional interpolating functions inside the element; U_{ia}^e e $P_{a'}^e$ are the nodal velocities and pressure values in nodes a and a', respectively. The notation a' is used to consider that the pressure can be interpolated by interpolation functions of different order of that used to interpolate the velocity or other scalar variable. Generally, the pressure is interpolated by functions of one order lower than the velocity, in order to avoid numerical instabilities.

Considering, then, the equations to a control sub-volume associate to a certain element node we obtain

$$\begin{aligned}
&\int_{A_a} \frac{rU_i^*}{\Delta t} dA + \sum_{K=1}^{NF} \left[\int_{\Gamma_{Ka}} \left(q^* \left(rUU_i \right)^n + \left(1-q^*\right)\left(rUU_i \right)^{n-1} - (1-q) \left(m_e \frac{\partial U_i}{\partial X} \right)^n - q \left(m_e \frac{\partial U_i^*}{\partial X} \right) \right) dY \right] \\
&- \sum_{K=1}^{NF} \left[\int_{\Gamma_{Ka}} \left(q^* \left(rVU_i \right)^n + \left(1-q^*\right)\left(rVU_i \right)^{n-1} + (1-q) \left(m_e \frac{\partial U_i}{\partial Y} \right)^n + q \left(m_e \frac{\partial U_i^*}{\partial Y} \right) \right) dX \right] - \int_{A_a} \frac{rU_i^n}{\Delta t} dA \\
&- q \int_{A_a} F_i^* dA - (1-q) \int_{A_a} F_i^n dA + \textit{similar contributions of other elements for the node } a \; + \\
&\textit{boundary contribuitions in the case} = 0
\end{aligned} \tag{21}$$

$$\sum_{k=1}^{NF} \left[\int_{\Gamma_a} \left(\frac{\partial P^{n+1}}{\partial X} \right) dY - \int_{\Gamma_a} \left(\frac{\partial P^{n+1}}{\partial Y} \right) dX \right] - \int_{A_a} \frac{1}{\Delta t} \left(\frac{\partial \left(rU_i^* \right)}{\partial X_i} \right) dA = 0 \; . \tag{22}$$

$$\int_{A_a} \frac{rU_i^{n+1}}{\Delta t} dA - \int_{A_a} \frac{rU_i^*}{\Delta t} dA + \int_{A_a} \frac{\partial P^{n+1}}{\partial X_i} dA + \textit{contribuitions of other elements for the node } a = 0 \; . \tag{23}$$

By substitution of the interpolated variables defined by Eq. (19) and (20) in Eq. (21) and (23), we obtain the following system, in the scalar form, to one element:

$$\begin{aligned}
&\frac{M_{ab}}{\Delta t} U_{ib}^* - qS_{ab} U_{ib}^* = \frac{M_{ab}}{\Delta t} U_{ib}^n - q^* C_{ab}^n U_{ib}^n - \left(1-q^*\right) C_{ab}^{n-1} U_{ib}^{n-1} + \\
&+ (1-q) S_{ab} U_{ib}^n + q^{**} F_{ia}^* + \left(1-q^{**}\right) F_{ia}^n
\end{aligned} \tag{24}$$

$$H_{a'b'}P_{b'}^{n+1} = \frac{1}{\Delta t} D_{ia'b} U_{ib}^{*} \tag{25}$$

$$\frac{M_{ab}}{\Delta t} U_{ib}^{n+1} = \frac{M_{ab}}{\Delta t} U_{ib}^{*} - G_{iab'} P_{b'}^{n+1} . \tag{26}$$

In the Equations (24) to (26), the matrices elements are defined as

$$M_{ab} = \int_{A_{SVCa}} r N_b \, dA \tag{27}$$

$$C_{ab} = \oint_{\Gamma_{SVCa}} r N_b U_j n_j \, d\Gamma \tag{28}$$

$$F_{ia} = \int_{\Gamma_{SVCa}} m_e \frac{\partial U_j}{\partial X_i} n_j \, d\Gamma + \int_{A_{SVCa}} r g_i \, dA \tag{29}$$

$$H_{a'b'} = \int_{\Gamma_{SVCa'}} \frac{\partial N_{b'}}{\partial X_i} n_i \, d\Gamma \tag{30}$$

$$D_{ia'b} = \int_{A_{SVCa'}} r \frac{\partial N_b}{\partial X_i} dA \tag{31}$$

$$G_{iab'} = \int_{A_{SVCa}} \frac{\partial N_{b'}}{\partial X_i} dA \tag{32}$$

An alternative formulation of Poisson pressure equation can be obtained from Eq. (10) by direct integration of that equation without compute the divergence and we obtain

$$\frac{M_{ab}}{\Delta t} U_{i}^{n+1} = \frac{M_{ab}}{\Delta t} U_{ib}^{*} - G_{iab'} P_{b'}^{n+1} . \tag{33}$$

From Eq. (11), we obtain

$$D_{ia'b} U_{ib}^{n+1} = 0 . \tag{34}$$

In this case, taking $M_{ab} = \overline{M}_{ab}$ in Eq. (33) and using the Eq. (34) results

$$D_{ia'b} \overline{M}_{ab} G_{iab'} P_{b'}^{n+1} = \frac{1}{\Delta t} D_{ia'b} U_{ib}^{*} + F_{pa'} \tag{35}$$

$$U_{ib}^{n+1} = U_{ib}^{*} - \Delta t \overline{M}_{ab} G_{iab'} P_{b'}^{n+1} . \tag{36}$$

Others formulations can be obtained, as, for example, the fractional step method D* (Bell *et al.*, 1989; Kim and Lee, 2002), in which, taking $F_{ia} = -G_{iab'} P_{b'}$, solve the Eq. (24) with $q = \frac{1}{2}$, $q^{*} = q^{**} = 0$ and Eq. (35) and (36). In this case, the matrices are defined as

$$H_{a'b'} = \int_{\Omega} \frac{\partial N_{a'}}{\partial X_i} \frac{\partial N_{b'}}{\partial X_i} d\Omega \,. \tag{37}$$

$$F_{pa'} = \int_{\Gamma_{SVCa}} N_{a'} \frac{\partial P}{\partial X_i} n_i d\Gamma \,. \tag{38}$$

$$D_{ia'b} = \int_{\Omega} N_{a'} \frac{\partial N_b}{\partial X_i} d\Omega \,. \tag{39}$$

In another formulation, known by method I (Ramaswamy, 1988; Ramaswamy *et al*., 1992), the Eq. (24) is solved taking $q = 0$ e $q^* = 1$ and the Eq. (35) and (36). In this case, the matrices in the Poisson equation are defined as

$$M_{ab} = \int_{\Omega} r N_a N_b d\Omega \tag{40}$$

$$H_{iab'} = \int_{\Omega} \frac{\partial N_a}{\partial X_i} N_b \, d\Omega \tag{41}$$

$$D_{ia'b} = \int_{\Omega} N_{a'} \frac{\partial N_b}{\partial X_i} d\Omega \tag{42}$$

$$F_{pa} = \int_{\Gamma_{SVCa}} N_a P n_i d\Gamma \tag{43}$$

4. NUMERICAL EXAMPLE AND RESULTS

The lid-driven cavity flow of a Newtonian fluid is taken as the application of the proposed scheme. The problem definition is given in Fig.1. The square cavity of unity side has its lower left corner at the origin of a Cartesian coordinate system. The velocities are set to zero at all walls, except the velocity along the x - axis for the upper plate. The pressure was set equal to zero at the mid bottom point. The problem definition is given in Fig. 1, as well the mesh for discretization of the domain. The typical element, Plane 77 or 82 of the preprocessor ANSYS 6.0®, modified for the inclusion of the central node, was used to create a regular non uniform mesh of 40 by 40 elements, with 81 by 81 nodes along the coordinated axes. The results for the Reynolds numbers of 100, 400, 1,000 and 3,200 of the present work were compared with results from Ghia *et al*. (1982) at steady state.

The lid-driven cavity flow of a Newtonian fluid has occupied the attention of the scientific computational community since the pioneering paper of Burggraf back in 1966. Over the years, the problem has spawned a large number of papers; mainly concerned with the development of computational algorithms where, in a continuous drive to demonstrate the superior accuracy and stability properties of their latest numerical method, authors have applied it to this problem in two-dimensional or three-dimensional forms. This is a classical example of re-circulating fluid flows in a confined area. From a purely computational viewpoint, is an ideal prototype non-linear problem which is a readily posed for numerical solution. The classical lid-driven cavity problem has been investigated by many authors. In the recent work of Bruneau and Saad (2006), simulations of the 2D lid-driven cavity flow have been performed for various Reynolds numbers. Accurate benchmark results are provided for steady solutions as well as for periodic solutions around the critical Reynolds number.

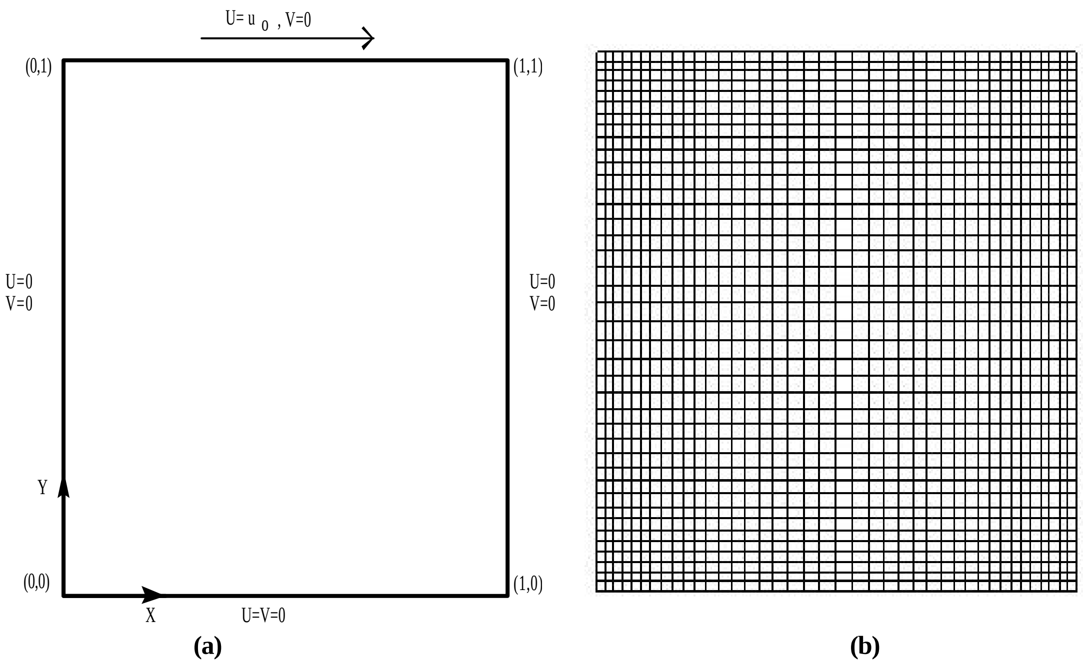

Figure 1 - Cavity flow: (a) configuration, coordinates and boundary conditions
(b) finite element mesh (40x40)

Two fractional step method's implementations were accomplished for obtaining the components of velocity U and V in the time, validated through the comparison with Ghia *et al*. (1982). In the first implementation, called Method 1, in the first step, the intermediate velocity was advanced in an explicit way. The boundary condition for the solution of the Poisson equation of the pressure was of first type, that is, null p in a die point. In the second implementation, called method D*, in the first step, the diffusive term was discretized in an implicit way and the boundary condition for solution of the Poisson equation of the pressure was of second type, that is, the free gradient of pressure at boundaries.

The Fig. 2 and 3 show the results for profiles of U(Y) velocity at X=0,5 (mid vertical line) and V(X) velocity at Y=0,5 (mid horizontal line) for the Reynolds number of 400 of the present work, respectively for Methods 1 and D*, compared with results from Ghia *et al*. (1982), at steady state. While, the Fig. 4 and 5 show the results for profiles of U(Y) velocity at X=0,5 (mid vertical line) and V(X) velocity at Y=0,5 (mid horizontal line) for the Reynolds number of 1000 of the present work, respectively for Methods 1 and D*, compared with results from Ghia et al. (1982), at steady state. Notice that, in the Method 1, close to the inflection point of the velocity the values were underestimated. In both cases, for Reynolds number 400 and 1000, that corresponds to the laminar regime, a certain discrepancy is observed among the results resultant, perhaps, due to some anomaly of CVFEM with the fractional step method. The Fig. 6 illustrates the velocity profile U for Reynolds number 3200 of the fractional step method of the method 1, described previously, for steady state. The results are compared with Ghia *et al*. (1982). In this case, the results are wavy and they did not present agreement. This way the influence of the viscosity of Smagorinsky, still has to be better investigated. This Reynolds 3200 corresponds to the transition strip of laminar to turbulent regime. The application the fractional step with CVFEM for high Reynolds numbers flows has to be enhanced as was pointed out by Campos (2005). Modifications in the form of implementation shall be investigated to verify if the solution converge for high Reynolds numbers flows and if for other geometries the fractional step is suitable.

The implementation of fractional step method, used for the solution of the equation of Navier-Stokes, was shown to be quite complex. Although, since the decade of 60, it haas been exhaustively studied in CFD literature and several implementations have been proposed, studies involving theoretical fundamentation and aspects of application still continue in process, according to very recent works like Codina and Blasco (2004) and Vijalapura (2005). The fractional step method possesses only characteristics when compared with all the other methods for the numerical solution of the Navier-Stokes. One of them is the fact that the boundary conditions imposed in the incompressible phase of the calculation are different depending on the way assumed by the equations and of the type of adopted space discretization. Basically, it is necessary to be distinguished the discreet representation if the variable pressure is or not defined at the contour. If the pressure is defined at the contour, a condition of additional contour is requested to determine the value of the contour of this variable. The other alternative is when the variable pressure is not defined at the contour that happens, for instance, when we use approaches with finite elements representing the field pressure through a continuous polynomial by parts and that is discontinuous in the contours. The choice of the lid-driven cavity flow for the test of the method was due to fact that this satisfies the restrictions at contours of the domain, besides the mass flow to be null through the walls. With effect, this problem is framed in the restricted group of problems that satisfies the restrictions in the contours of the domain for the application of the method. In this work, in an of the steps of the methodology the velocity field from the Navier-Stokes equations is obtained with the correction of the velocity field obtained through the solution of the Burges equation, that is the simplified version of the Navier-Stokes equations for the cases that the gradient of the pressure can be despised.

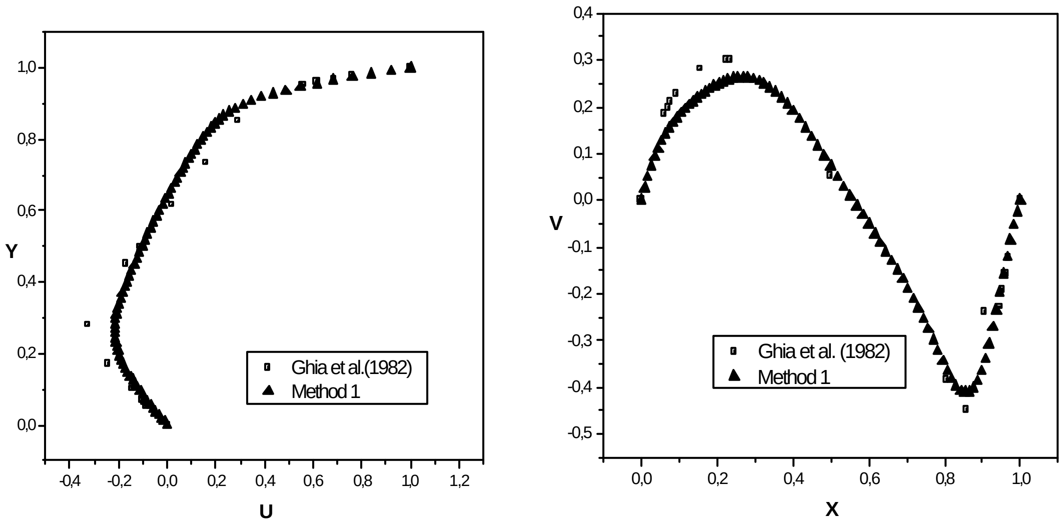

Figure 2: Method 1, Re= 400 (a) U-velocity at X = 0,5 (b) V-velocity at Y = 0,5

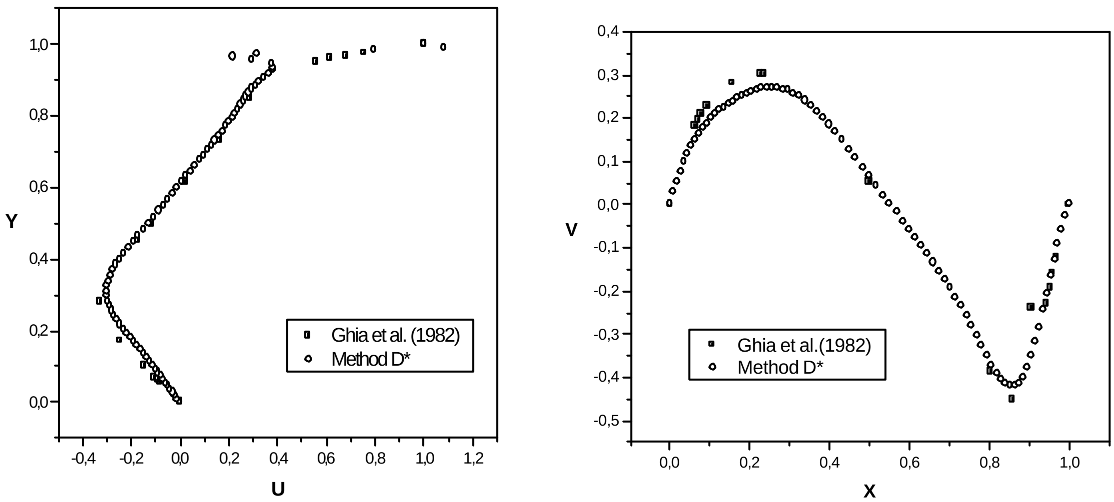

Figure 3: Method D*, Re= 400 (a) U-velocity at X = 0,5 (b) V-velocity at Y = 0,5

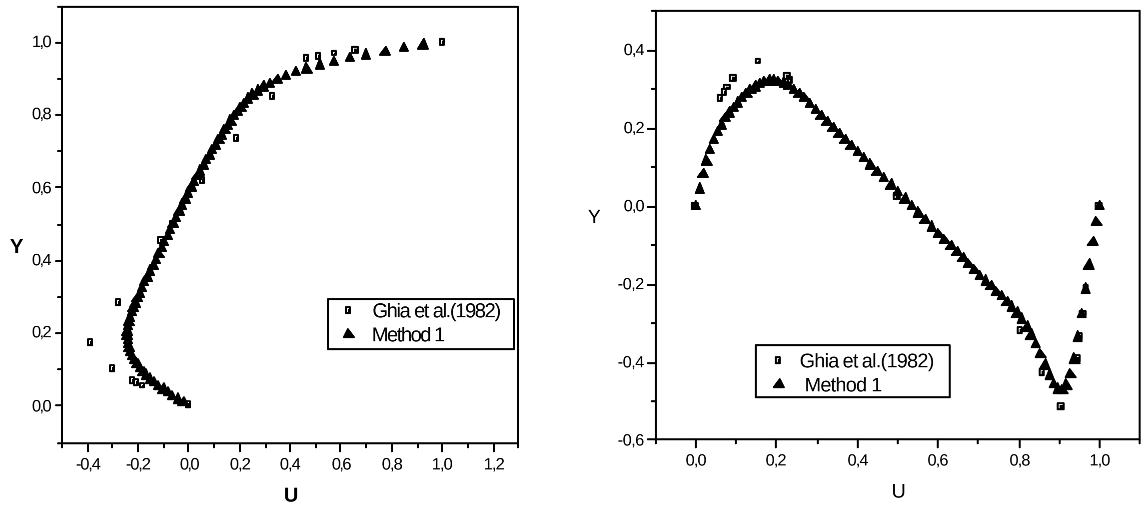

Figure 4: Method 1, Re= 1000 (a) U-velocity at X = 0,5 (b) V-velocity at Y = 0,5

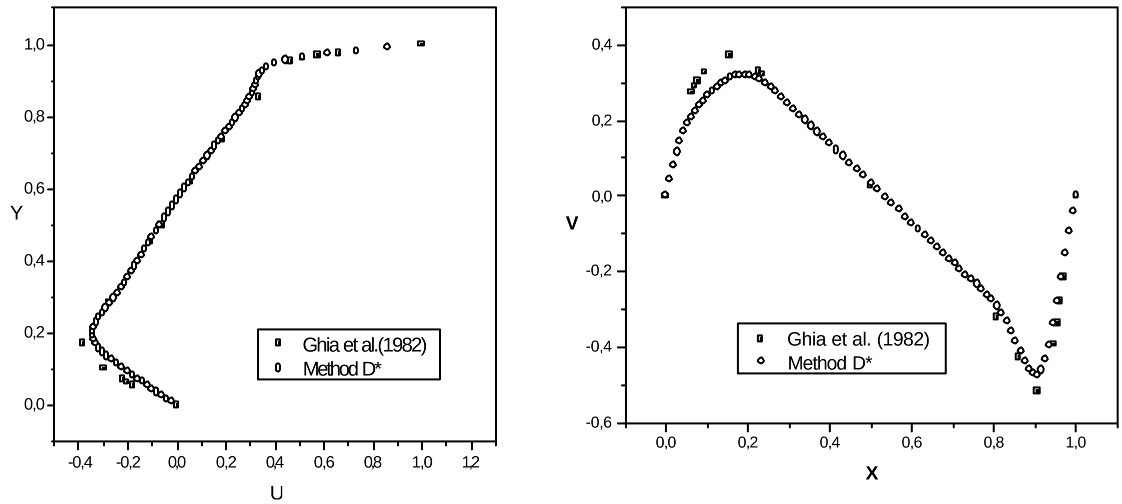

Figure 5: Method D*, Re= 1000 (a) U-velocity at X = 0,5 (b) V-velocity at Y = 0,5

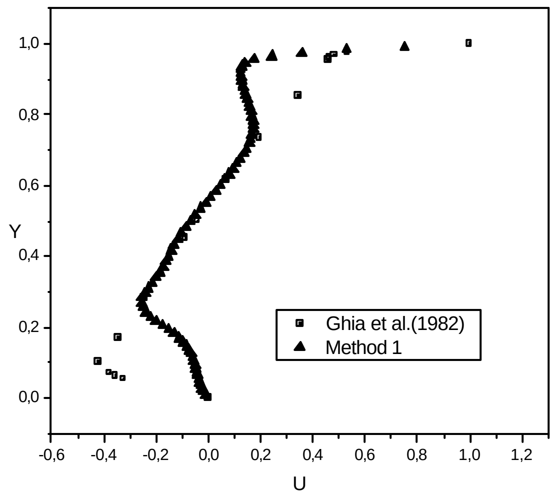

Figure 6: Method 1, Re= 3200, U-velocity at X = 0,5

5. CONCLUSIONS

A CVFEM with fractional step was proposed in this work to solve the Navier-Stokes equations. The case of the lid-driven cavity flow was analyzed and showed that the numerical method needs to be enhanced for convergence of the solution for high Reynolds flows. The fractional step method of time discretization posses advantages, once it simplifies the differential equations to be solved, however, with the CVFEM method it has to more investigated. The accomplished test allowed the investigation of some aspects of the method that would influence to the solution, such as the relationship among the quality of the numeric solution the refine of the mesh of the discretization and the use of boundary conditions adapted to the numeric solution of the equation of Poisson for the pressure. Other options of fractional steps can be more appropriated with the CVEFM method, they should be implemented in future works.

6. REFERENCES

Bell, J.B. et al., 1989, "A Second-Order Projection Method for the Incompressible Navier-Stokes Equations", Journal of Computational Physics, Vol. 85, pp. 257-283.

Bruneau, C. H. and Saad, M., 2006, "The 2D Lid-driven Cavity Problem Revisited", Computer & Fluids, Vol. 35, pp. 326-348.

Buggraf, O. R., 1966, "Analytical and Numerical Studies of the Structure of Steady Separated Flows", Journal of Fluid Mechanics, Vol. 24, pp. 113-151.

Campos, M.D., 2005, "Simulação Numérica de Escoamentos de Fluidos pelo Método de Elementos Finitos Baseado em Volumes de Controle e Usando a Técnica de Passo Fracionado", (Dissertação de Mestrado), Departamento de Engenharia Mecânica, Faculdade de Engenharia de Ilha Solteira, Brasil.

Chorin, A.J., 1968a, "Numerical Solution of the Navier-Stokes Equations", Mathematics of Computation, Vol. 22, pp. 745-762.

Chorin, A.J., 1968b, "Numerical Solution of Incompressible Flow Problems", Studies in Numerical Analysis, Vol. 2, pp. 64-71.

Chorin, A.J., 1969, "On the Convergence of Discrete Approximations to the Navier-Stokes Equations", Mathematics of Computation, Vol. 23, pp. 341-353.

Codina, R. and Blasco, J., 2004, "Error Estimates for an Operator-Splitting Method for Incompressible Flows", Applied Numerical Mathematics, Vol.51, pp.1-17.

Ghia, U. et al., 1982, "High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method, Journal of a Computational Physics, Vol. 48, pp. 387-411.

Kim, Y. B., Lee, M. J., 2002, "Boundary-Condition Pairs for Fractional-Step Method and Compatibility with the Pressure Poisson Equation", Computational Fluid Dynamics Journal, Vol. 11, pp.323-334.

Ramaswamy, B., 1988, "Finite-Element Solution for Advection and Natural Convection Flows", Computer & Fluids, Vol. 16, pp. 349-388.

Ramaswamy, B. et al., 1992, "Semi-Implicit and Explicit Finite Element Schemes for Coupled Fluid/Thermal Problems", International Journal for Numerical Methods in Engineering, Vol. 34, pp. 675-696.

Temam, R., 1984, "Navier-Stokes Equations: Theory and Numerical Analysis", Elsevier Science Publishing Company, Amsterdan , 526 p.

Temam, R., 1969a, "Sur L'approximation de la Solution des Equations de Navier-Stokes par la Méthode des Pas Fractionaires(I) ", Archive for Rational Mechanics and Analysis, Vol. 32, pp. 135-153.

Temam, R., 1969b, "Sur L'approximation de la Solution des Equations de Navier-Stokes par la Méthode des Pas Fractionaires(II)", Archive for Rational Mechanics and Analysis, Vol.33, pp. 377-385.

Vijalapura, P.K. et al., 2005, "Fractional Step Methods for Index-1 Differential-Algebraic Equations", Journal of Computational Physics, Vol. 203, pp. 305-320.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.