Blucher Mechanical Engineering Proceedings
May 2014, vol. 1 , num. 1
www.proceedings.blucher.com.br/evento/10wccm

10th World Congress on Computational Mechanics
8-13 July 2012 • São Paulo • Brazil

INCOMPRESSIBLE FLUID FLOW BY THE MACCORMACK METHOD

M. M. Martins[1] , J. D. Bressan[2], S. T. Button[3]

[1]Department of Materials Engineering, Faculty of Mechanical Engineering, State University of Campinas (demaram@fem.unicamp.br)

[2]Department of Mechanical Engineering, Centre for Technological Science, University of Santa Catarina State

[3]Department of Materials Engineering, Faculty of Mechanical Engineering, State University of Campinas

Abstract. *The laminar incompressible fluid flow by computational numerical simulation often appears in numerical analysis in academic and industrial activities. In order to solve this kind of flow, it is necessary to determine the velocity and pressure fields which are the variables of Navier-Stockes equations [11,15,21]. However, to solve the equations of fluid flow with losses there is no simple equation to carry out velocity and pressure coupling, hence, it is necessary to use a coupling method to obtain velocity and pressure fields consistent [1,2,3]. This work deals with the presentation of a numerical method to calculate velocity and pressure fields to computational numerical simulation of laminar fluid incompressible flow with losses. The Navier-Stockes equations were discretized by the Finite Volume Method [11,15,21], using explicit MacCormack Method [21] in co-localized and structured mesh [11,15,21], where velocity and pressure coupling was made by SIMPLE method [11,21]. The MacCormack method is a two-steps method (predictor-corrector) of second-order accuracy in both space and time and this method is commonly utilized in the resolution of compressible fluids problems [21]. The numerical results of velocity fields were obtained for bi-dimensional case and it was compared with analytical results for parallel plates.*

Keywords: *incompressible fluid; finite volume method; MacCormack Method; SIMPLE; velocity field.*

1. INTRODUCTION

The computational numerical simulation, nowadays, is an indisputable reality in academic and industrial environment. How can be seen through of number scientific paper published in appropriate magazine or through commercial software used in specific projects of industries [11]. In the fluid dynamic the reality is not different. Because, the quantity of scientific paper and industries that use private or commercial software to solve problems of compressible or incompressible fluid flow show this.

By [19] a viscous fluid is defined when the viscosity effects or internal shear stresses cannot be neglected in the fluid flow. According [19] incompressible fluid is defined by density must remain constant in the fluid flow. [11] define incompressible fluid as a fluid that doesn't have a specific equation for pressure, this is, density is not function of pressure.

In incompressible fluid flow analysis, usually, tries to determine the temperature, velocity and pressure fields [21]. These informations are represented, mathematically, by mass, momentum and energy conservation equations. Inside of this equations appear the primitive variables, that are: density, velocity components, temperature and pressure [11,21].

According [11] compressible fluid flow equations are solved using a state equation. But, incompressible fluid flow there is not state equation, according [11]. The challenge in the incompressible fluid flow is extract the pressure of the momentum equation and the velocities obtained should satisfy the continuity equation.

According [21] there are two approaches to solve an incompressible fluid flow. The first approach is called *"Coupled Approach"*, where the aim is insert in the equations an artificial compressibility and this strategy was presented by [2]. The second approach is called *"Pressure-Correction Approach"*, where the main feature is to obtain a pressure correction equation from continuity or momentum equation. This strategy was presented by [14].

In the pressure-correction approach some proceedings were developed. [11,15,21] presented and discussed the main proceeding that were developed, such as: SIMPLE Method, SIMPLER Method, PRIME Method, etc.

Among these pressure-velocity coupling method, the SIMPLE (Semi-Implicit Method for Pressure Linked Equation) method have some highlight, because it is large applied. How can be seen in the works [3,4,13].

According [11] the velocity correction equations are obtained from momentum equation and pressure correction equation is obtained from continuity equation, this last equation are a Poisson Equation and that should be solved with boundary condition appropriated.

In numerical analysis of incompressible fluid flow the position of velocity and pressure on the control volume is an important aspect in the Finite Volume Method (FVM). The natural position, apparently simpler is storing all variables in the centre of control volume, according [11]. This arrangement kind is called *"Collocated grid or Nonstaggered grid"*. According [11] the advantage this arrangement is simplicity of index control in the computational implementation, but, in this approach arise *"Checkerboard Oscilations"*.

According [11] to solve the *"Checkerboard Oscillations"*, when is used collocated grid, can be made dislocating the velocity from control volume centre to control volume faces. This dislocating is made through an interpolation scheme. [21] suggest the *"Momentum Interpolation Method"* presented by [20] and can be seen in the works [3,4,13].

The alternative variables arrangement in incompressible fluid flow is *"Staggered Grid"*, according [11,21]. Where, pressure and velocity have their own control volume. According [11] this arrangement is physically consistent, however, the index control in the computational implementation is harder. Being the first arrangement appears in several works, such as, [12, 16]. The staggered grid arrangement is described in [11,15,21].

The time formulation is other important aspect in the discretization of flow equation. [11] suggest the explicit formulation, where, all neighbour values of calculated point were calculated in the before instant, generating a set of algebraic equations. Others formulations are called, respectively, implicit and totally implicit and both generate a linear equation system.

How commented before the explicit formulation generates a set of algebraic equations, that are solved one by one in the process very fast. However, a numerical method to obtain a consistent solution it must satisfy the consistency, stability and convergence condition.

According [21] a numerical method is called consistent when the numerical solution gets close of analytical solution with refining mesh. [21] also say that numerical method has stability if any errors decrease of time step to next time step.

[11,21] say that a numerical method is convergent if this method is consistent and stable. This affirmation is guaranteed by *"Lax theorem"*, given by:

"Given a properly initial value problem and finite-difference approximation to it that satisfies the consistency condition, stability is that necessary and sufficient condition for convergence [21]"

According [11] all numerical models developed from equation in the conservative form and used FVM is said consistent. Therefore, an explicit numerical method that used discretized equations in the conservative form by FVM will be consistent. However, the numerical method stability will be given by CFL (Courant-Friedrichs-Lewy) condition, where the CFL condition depend on time step, grid size, among others information. So, an explicit numerical method is called *"Conditionally stable"*.

[21] suggest some methods to solve the governing equations of fluid flow, such as: Euler Method, Lax-Wendroff Method, Brailovskaya Method, Range-Kutta Method, MacCormack Method, etc. When is made a comparison among these method some have advantages and disadvantages in some features.

The MacCormack Method has some important features and is large applied to solve compressible fluid flow problems.

The MacCormack is a variation of two steps Lax-Wendroff Method, but, more efficient to solve PDEs non-liners, generating good solutions to discontinuities of this equations [21].

According [21] the MacCormack Method is a two steps method, this is, it calculates a *"Predictor Step"* and followed it calculates a *"Corrector Step"*, defining the final value from an arithmetic mean. If predictor step used a forward differential operator and corrector step use backward differential operator. The method must follow this scheme to ensure the main feature a second-order of accuracy for time and space.

The MacCormack Method doesn't require a control oscillations method, but, the practice show it needs one to obtain good solutions. These control oscillations method can be viscosity artificial method or TVD (Total Variation Diminishing) scheme, how may be seen in the works [5,9,17,18,22].

This work treats of numerical method that applied explicit MacCormack Method in the solution of governing equations of incompressible viscous fluid flow, through computational numerical simulation by FVM to structured and collocated mesh, where the pressure-velocity coupling is made by SIMPLE method and by Moment Interpolation Method.

2. MATHEMATICAL MODEL FOR A INCOMPRESSIBLE VISCOUS FLUID

2.1. Governing Equations

The incompressible viscous fluid flow is given, mathematically, by conservations law. The conservations law are general principles that governing the movement of bodies submitted by external forces. The conservations law are given by continuity, momentum and energy equations [21], given below:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \tag{1}$$

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v}\vec{v}) = \nabla \cdot \sigma + \rho b \tag{2}$$

$$\frac{\partial (\rho c T)}{\partial t} + \nabla \cdot (\rho c T \vec{v}) = \mathrm{tr}(\sigma \cdot \dot{\varepsilon}) + \rho r - \nabla \cdot \dot{q} \tag{3}$$

Where, ρ is the density, $\vec{v}$ is the velocity vector, t is time, σ are surface forces, b are body forces, c is the specific heat in pressure and volume constant, T is temperature, $\dot{\varepsilon}$ is strain rate tensor, $\mathrm{tr}(\sigma \cdot \dot{\varepsilon})$ is rate of plastic work, r is heat per unit of mass e $\dot{q}$ is the flux heat vector [21].

2.2. Constitutive Equations

The relationship among the material behaviour in mechanical loading condition and/or thermal through variables of static (by stresses), kinematic (by displacement, strain and velocity) and thermal (by thermal flow and temperature) are given through mathematical expressions called constitutive equations [7]. The constitutive equations used in this model are presented by Equation (4), which makes the relation between stresses with strain rate and is given by generalized Stocks Law, the strain rate tensor is given by Equation (5) and the Fourier's Law is given by Equation (6).

$$\sigma = -pI + 2\mu\dot{\varepsilon} \tag{4}$$

$$\dot{\varepsilon} = \frac{1}{2}\left[\mathrm{grad}\, v + (\mathrm{grad}\, v)^T\right] \tag{5}$$

$$\dot{q} = -k\nabla \cdot T \tag{6}$$

Where, σ is the stress tensor, $\dot{\varepsilon}$ is the strain rate tensor, p is the pressure, I is identity tensor, μ is the dynamic viscosity, $\vec{v}$ is the velocity vector, T is the temperature and k is the thermal conductivity.

3. NUMERICAL METHOD

3.1. Finite Volume Method

Consider the Equations (1), (2) e (3), that represent the differential form of governing equations, written in a compact vector form and given by (7).

$$\frac{\partial Q}{\partial t} + \nabla \cdot F = S \tag{7}$$

Where, Q is the primitive variable vector, F is a flux vector and S is source term. Integrating the Equation (7) over control volume and following the deductions presented in [21], have the Equation (8), that represent the discretization by FVM to bi-dimensional, quadrilateral and stationary in the space control volume, represented by Figure 1 (a).

$$\frac{\partial Q_{mn}}{\partial t} = -\frac{1}{V_{mn}}\left[\left(\vec{F}\cdot\vec{s}\right)_{m-\frac{1}{2},n} + \left(\vec{F}\cdot\vec{s}\right)_{m+\frac{1}{2},n} + \left(\vec{F}\cdot\vec{s}\right)_{m,n-\frac{1}{2}} + \left(\vec{F}\cdot\vec{s}\right)_{m,n+\frac{1}{2}}\right] + S_{mn} \tag{8}$$

Where, $\vec{F}$ is the flux vector to bi-dimensional case, $\vec{s}$ is the outward normal vector control volume surface and V is the volume of control volume, that in the bi-dimensional case represent the control volume area.

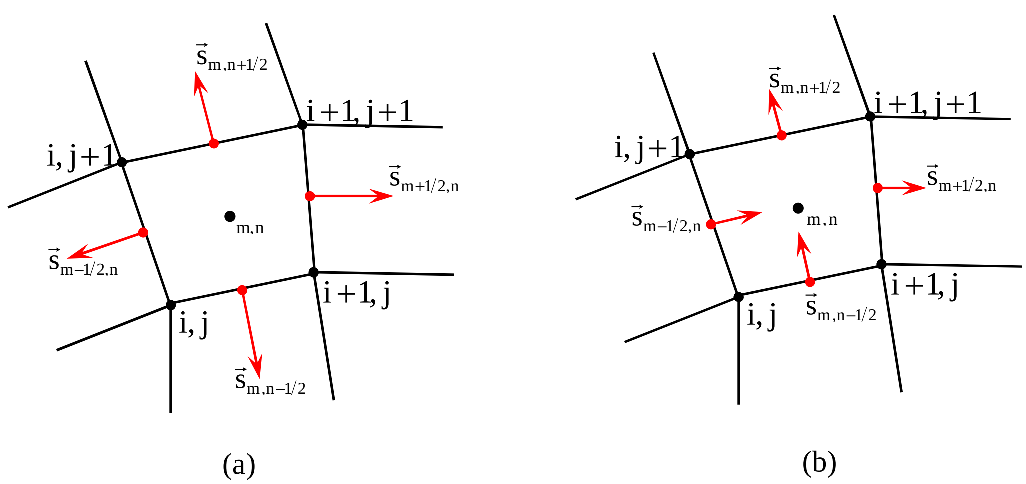

Figure 1 – Representation of quadrilateral control volume. (a) specifying outward normal vector control volume. (b) specifying flux way to flow.

3.2. MacCormack Method

The Equation (8) was solved by MacCormack explicit scheme, present in reference [21]. Considering the control volume given by Figure 1 (b), have equations for predictor step, corrector step and update step, given respectively, by Equations (9), (10) e (11).

$$\overline{Q_{mn}^{t+1}} = Q_{mn}^{t} - \frac{\Delta t}{V_{mn}} \left\{ \begin{matrix} \left[\left(F_x\right)_{m+1,n} \cdot \left(s_x\right)_{m+\frac{1}{2},n} + \left(F_y\right)_{m+1,n} \cdot \left(s_y\right)_{m+\frac{1}{2},n} \right] - \\ \left[\left(F_x\right)_{m,n} \cdot \left(s_x\right)_{m-\frac{1}{2},n} + \left(F_y\right)_{m,n} \cdot \left(s_y\right)_{m-\frac{1}{2},n} \right] + \\ \left[\left(F_x\right)_{m,n+1} \cdot \left(s_x\right)_{m,n+\frac{1}{2}} + \left(F_y\right)_{m,n+1} \cdot \left(s_y\right)_{m,n+\frac{1}{2}} \right] - \\ \left[\left(F_x\right)_{m,n} \cdot \left(s_x\right)_{m,n-\frac{1}{2}} + \left(F_y\right)_{m,n} \cdot \left(s_y\right)_{m,n-\frac{1}{2}} \right] \end{matrix} \right\}^{t} + \Delta t\, S_{mn}^{t} \tag{8}$$

$$\overline{\overline{Q_{mn}^{t+1}}} = Q_{mn}^{t} - \frac{\Delta t}{V_{mn}} \left\{ \begin{matrix} \left[\left(F_x\right)_{m,n} \cdot \left(s_x\right)_{m+\frac{1}{2},n} + \left(F_y\right)_{m,n} \cdot \left(s_y\right)_{m+\frac{1}{2},n} \right] - \\ \left[\left(F_x\right)_{m-1,n} \cdot \left(s_x\right)_{m-\frac{1}{2},n} + \left(F_y\right)_{m-1,n} \cdot \left(s_y\right)_{m-\frac{1}{2},n} \right] + \\ \left[\left(F_x\right)_{m,n} \cdot \left(s_x\right)_{m,n+\frac{1}{2}} + \left(F_y\right)_{m,n} \cdot \left(s_y\right)_{m,n+\frac{1}{2}} \right] - \\ \left[\left(F_x\right)_{m,n-1} \cdot \left(s_x\right)_{m,n-\frac{1}{2}} + \left(F_y\right)_{m,n-1} \cdot \left(s_y\right)_{m,n-\frac{1}{2}} \right] \end{matrix} \right\}^{t} + \Delta t\, S_{mn}^{t} \tag{9}$$

$$Q_{mn}^{t+1} = \frac{1}{2}\left(\underline{Q_{mn}^{t+1}} + \underline{\underline{Q_{mn}^{t+1}}} \right) \tag{10}$$

Where, t is the current time step, $t+1$ is next time step, Δt is the time step, $\overline{Q_{mn}^{t+1}}$ is the predictor step, $\overline{\overline{Q_{mn}^{t+1}}}$ is the corrector step and Q_{mn}^{t+1} is the final time step. The vectors $\vec{s}$ that represent the outward normal control volume surface are calculated from computational mesh coordinates, following the direction presented in the Figure 1 (b). The flux vector $\vec{F}$ that appears inside of Equations (8) e (9) should be discretized in the correct way and presented in reference [21], to be ensured second-order accuracy of time and space, that is the main feature of MacCormack scheme.

3.3. Velocity-Pressure Coupling Method

According [11,21] in an collocated mesh there is checkerboard oscillations. This mesh requires a pressure-velocity coupling method. In this work was used a pressure-velocity coupling method made from SIMPLE method, that made the pressure and velocities corrections, and momentum interpolation method, to make the dislocation of velocities of control volume centre to control volume faces.

According [2,4] the SIMPLE method, for bi-dimensional case, use the Equations (11) till (13) to make the pressure and velocities corrections.

$$p = p_0 + p' \tag{11}$$

$$u = u_0 + u' \tag{12}$$

$$v = v_0 + v' \tag{13}$$

Where, "p", "u" e "v" are current values of pressure and velocities, respectively, "p_0", "u_0" e "v_0" are estimated values of pressure and velocities and "p'", "u'" e "v'" are variables corrections.

The cell given in Figure 2, the corrections of velocities and pressure are given by Equations (14) till (18).

$$u_e = u_e^* - A\left(\frac{p'_E - p'_P}{\partial x_e}\right) \tag{14}$$

$$u_w = u_w^* - A\left(\frac{p'_P - p'_W}{\partial x_w}\right) \tag{15}$$

$$v_n = v_n^* - A\left(\frac{p'_N - p'_P}{\partial y_n}\right) \tag{16}$$

$$v_s = v_s^* - A\left(\frac{p'_P - p'_S}{\partial y_s}\right) \tag{17}$$

$$p'_P = \left(A_E p'_E + A_W p'_W + A_N p'_N + A_S p'_S + B\right) / A_P \tag{18}$$

Where, the coefficients A, A_P, A_E, A_W, A_N, A_S and B are describe in details in the reference [2]. The lowercase subscripts are the positions of control volume faces and uppercase subscripts are the position of the control volume centre. According [11] the pressure correction equation tends to calculate super-estimated values of pressure, because of this the Equation (11) can be exchanged for Equation (19).

$$p = p_0 + \alpha p' \tag{19}$$

Where, α is an over-relaxation constant. [1,15] suggest that α values are in the range $0 < \alpha < 1$.

3.4. Boundary condition

Some boundary conditions were imposed over the control volume faces. These boundary conditions were imposed using the scheme called *"Ghost Volume"*. In these ghost volumes can be applied boundary condition of *"Dirichlet Kind"* and *"Neumann kind"*. According [8,11] the boundary condition applied for incompressible fluid flow should be imposed in this way:

- **Inlet:** the velocity field should be prescribed and pressure should be zero gradient.
- **Outlet:** the velocity should be zero gradient and pressure should be prescribed.

- **Symmetry line:** should be prescribed zero gradients for normal surface gradient and parallel surface components should use domain values.
- **Wall-solid:** pressure gradient is zero and velocity is also zero.

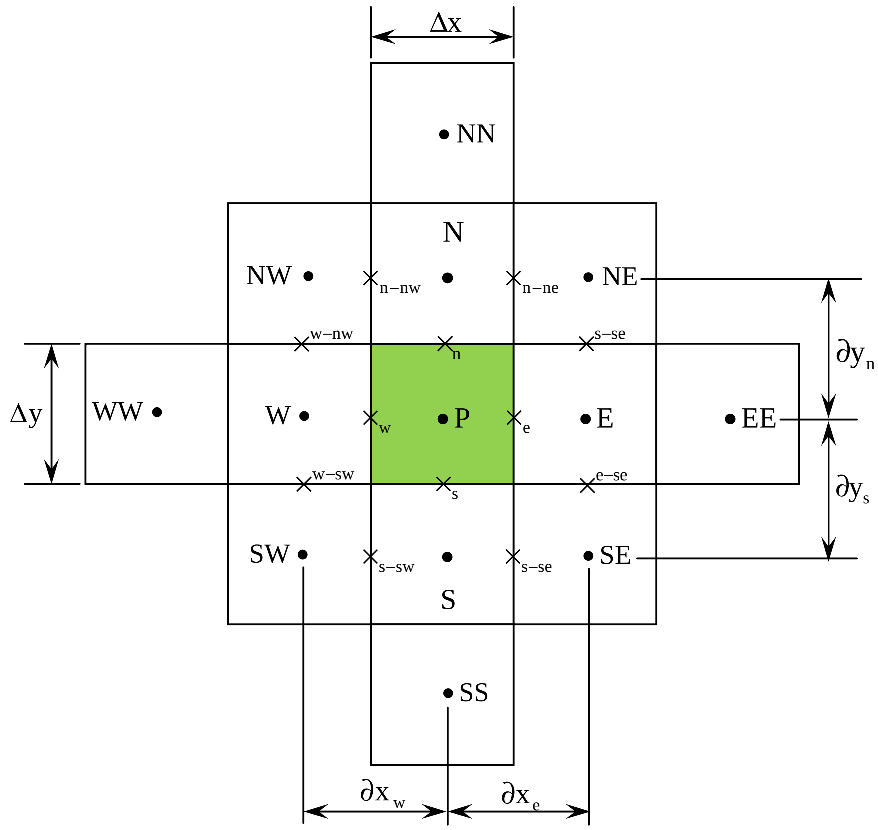

Figure 2 – Representation of computational mesh cell, defining the main control volume and his neighbours [22].

4. RESULTS

4.1. Case Discription

The flow studied in this work was bi-dimensional flow and is given by Figure 3. Where, inlet velocity profile is constant and represented by V_0, the channel height is given by 2h and L is the channel length. The channel length is 10h, this length ensures that flow is totally developed in channel outlet. The mesh is structured, quadrilateral and uniform, with aspect rate about 1 and 110 control volumes distributed in the axial direction and 10 control volumes in the radial direction. The inlet velocity is constant and equal $V_0 = 1.0$ m/s, defining a Reynolds Number equal Re = 27 ensuring a laminar flow. The incompressible viscous fluid proprieties used in the numerical simulation are in Table 1.

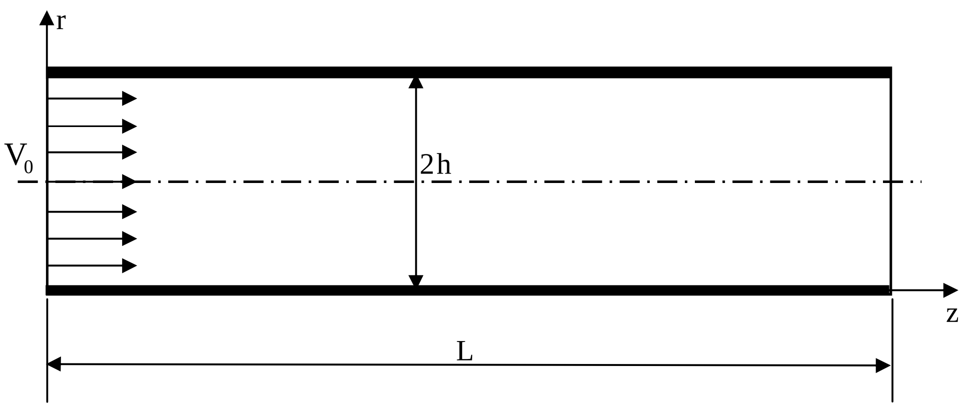

Figure 3 – Representation of geometric scheme of bi-dimensional flow.

Table 1. Incompressible Fluid proprieties

Proprieties	Value
Density	1261 $kg.m^{-3}$
Dynamic Viscosity	0.934 Pa.s
Specific heat	2430 $J.kg^{-1}.K^{-1}$
Thermal Conductivity	0.30 $J.m^{-1}.s^{-1}.K^{-1}$

4.1. Discussion Results

The velocity field was calculated by numerical code developed in FORTRAN language from numerical method presented before. The code is represented in the flowchart in the Figure 4. The axial velocity (Vz) results were compared with axial analytical velocity from analytical equation given in [19].

The comparison between numerical and analytical axial velocity for outlet channel is represented in the Figure 5. How can be seen there is a good agreement between of results, considering a mean square error about 0.01.

The Figure 6 and 7 represent axial velocity convergence and radial velocity convergence, respectively, in the mesh position 55x8. How can be seen there was velocity convergence during the number of iterations. The Figure 8 shows the analysed flow in the last graphics, how can be seen the flow is developed and profile velocity was achieved.

The Figure 9 shows the graphic of comparison between numerical and analytical axial velocity, considering mesh refinement or the increase of control volume number.

How can perceive the mesh refinement became the numerical results closer of analytical results. The Figure 10 shows the mean square error reduction with mesh refinement. Analysing the graphics of Figure 9 and 10 can perceive that computational code developed produces consistent results.

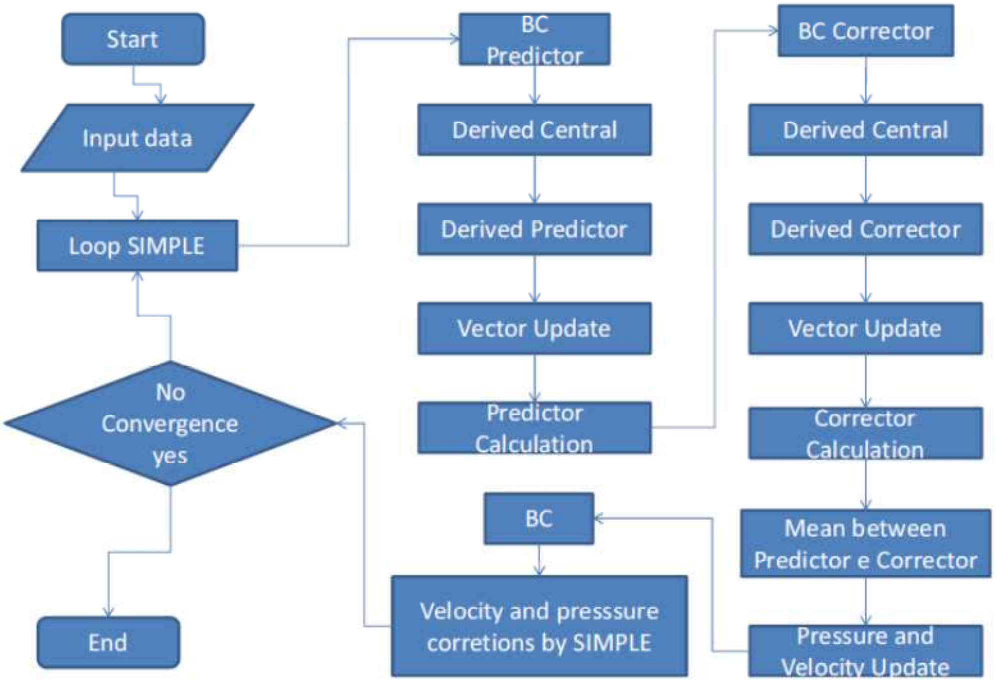

Figure 4. Flowchart of numerical code.

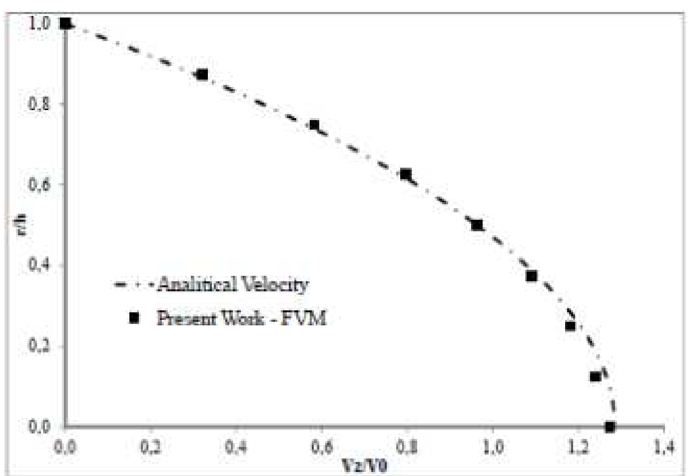

Figure 5. Comparison between numerical and analytical axial velocity in the channel outlet.

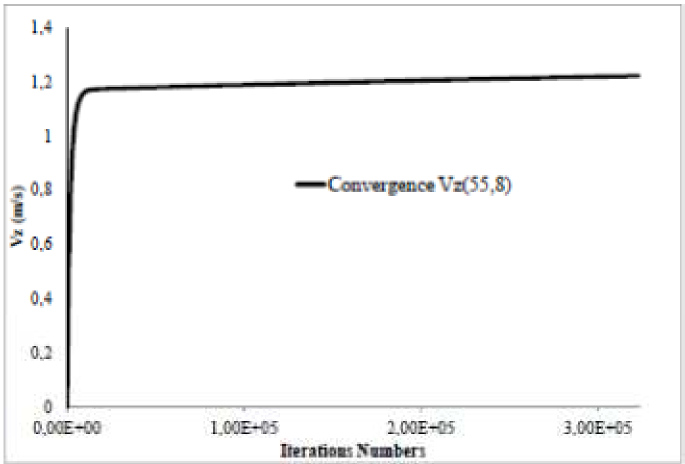

Figure 6. Axial velocity (Vz) convergence in mesh position 55x8.

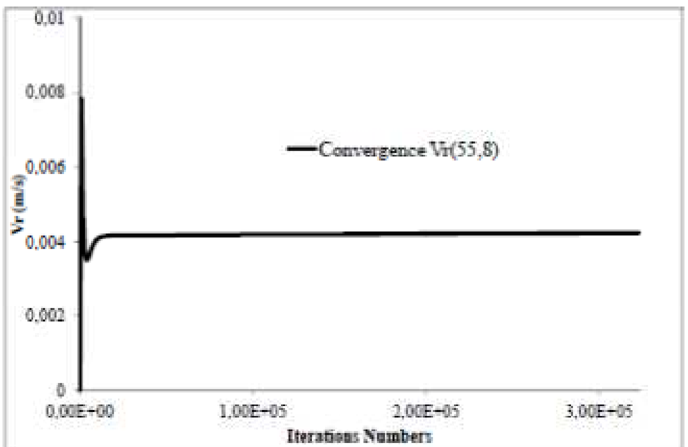

Figure 7. Radial velocity (Vr) convergence in mesh position 55x8.

Figure 8. Axial velocity totally developed in the fluid flow.

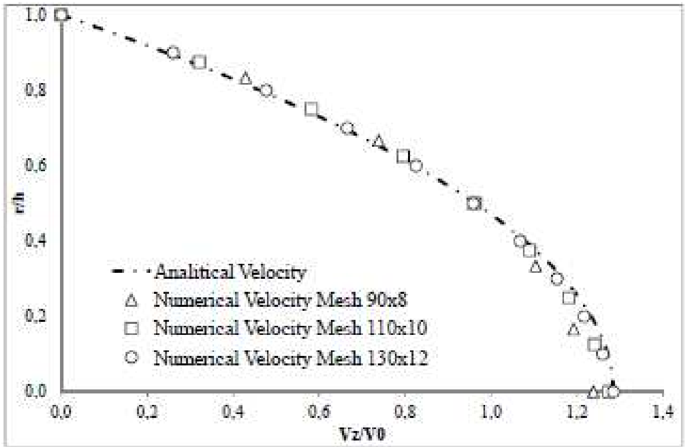

Figure 9. Comparison between mesh refinement and approximation of analytical results of axial velocity.

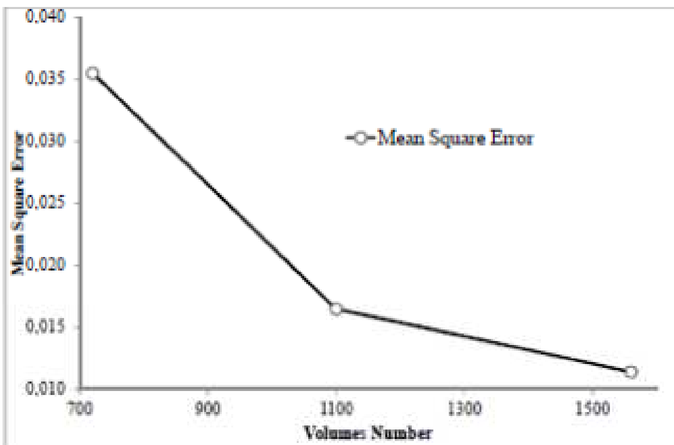

Figure 10. Show reduction of mean square error with increase of control volume number.

6. CONCLUSIONS

The incompressible fluid flow solved by numerical method presented in this work produced the following conclusions:

- The numerical method generated good agreement between numerical and analytical results of velocities;
- The numerical axial velocity profile was parabolic how suggest of literature;
- The numerical method calculated consistent numerical results no artificial viscosity addition;
- The governing equation were solved by explicit method, because of this, the numerical method required a large iterations number to convergence;
- The CFL condition required a time step in the minimum about 10^{-6};
- The over-relaxation coefficient of correction pressure equation was equal 1 and no prejudice to numerical solution;
- The MacCormack Method can be applied incompressible fluid flow beyond compressible fluid flow.

Acknowledgements

The authors would like to thank the support of The National Council for Scientific and Technological Development (CNPq) – Brazil, State University of Campinas (UNICAMP) – Brazil, State University of Santa Catarina (UDESC) – Brazil and University Catholic Centre of Santa Catarina – Brazil.

7. REFERENCES

[1] BASIC H., DEMIRDŽIC I.,DŽAFEROVIC E., “Finite volume method for simulation of extrusion processes”. *Inter. Journ. for Num. Meth. in Eng.* 62(4), 475–494, 2005.

[2] CHORIN A.J., “A Numerical Method for Solving Incompressible Viscous Flow Problems”. *Journ. of Comp. Phys.* 2, 12-26, 1967.

[3] CHOW P., CROSS M., PERICLEOUS K., “A Natural Extension of the Conventional Finite Volume Method into Polygonal Unstructured Meshes in CFD Application”. *Inter. Journ. of Heat and Mass Trans.* 20, 170-183, 1996.

[4] DARWISH M., SRAJ I., MOUKALLED F., “A Coupled Finite Volume Solver for the Solution of Incompressible Flow on Unstructured Grids”. *Journ. of Comp. Phys.*, 228, 180-201, 2009.

[5] FORT J., FURST J., HALAMA J., KOZEL K., PRAHA, “Numerical Solution of 3D Transonic Flow Through Cascade”. *Math. Bohem*. 126, 353-361, 2001.

[6] Hans P.L., Kent-Andre M., Ragnar W., “Numerical method for incompressible viscous flow”. *Adv. Wat. Res.* 25, 1125-1146, 2002.

[7] INCROPERA F.P., DEWITT D.P., BERGMAN T.L., LAVINE, A.S., “Fundamentos de Transferência de Calor e de Massa”. Editora: LCT. Rio de Janeiro, 2007.

[8] JASAK H. “Error Analysis and Estimation for the Finite Volume Method with Applications to Fluid Flows”. 1996. 394p. Thesis – Department of Mechanical Engineering, Imperial College of Science, Technology and Medicine, London.

[9] LIN G.F., LAI J.S., GUO W.D., “Finite Volume Component-wise TVD Schemes for 2D Shallow Water Equations”. *Adv. in Water Resour.* 26, 861-873, 2003.

[10] LOMAX H., PULLIAN T.H., ZINGG D.W., “Fundamentals of Computational Fluid Dynamics”. Editora: Springer, New York, 2003.

[11] MALISKA C.R., “Transferência de Calor e Mecânica dos Fluidos Computacional”. Editora: LCT, Rio de Janeiro, 2004.

[12] MOMPEAN G., THAIS, L., “Finite Volume Numerical Simulation of Viscoelastic Flows in General Orthogonal Coordinates”. *Math. and Comp. in Sim*. 80, 2185-2199, 2010.

[13] NEOFYTUL P., “A 3rd Order Upwind Finite Volume Method for Generalized Newtonian Fluid Flows”. *Adv. in Eng. Soft.* 36, 664-680, 2005.

[14] PATANKAR S.V., SPALDING D.B., “A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flow”. *Inter. Journ. of Heat and Mass Trans*. 15, 1787-1806, 1972.

[15] PATANKAR S.V., “Numerical heat Transfer and Fluid Flow”. Editora: Taylor&Francis, London, 1980.

[16] PILLER M., STALIO, E., “Finite-Volume Compact Schemes on Staggered Grids”. *Journ. of Comp. Phys*. 197, 299-340, 2004.

[17] PORÍZKOVA P.P., FURST J. HORÁCEK J., KOZEL, K., “Numerical Solution of Unsteady Flows with Low Inlet Mach Numbers”. *Math. and Comp. in Sim*. 80, 1795-1805, 2010.

[18] PORÍZKOVA P.P., KOZEL K., HORÁCEK J., “Simulation of Unsteady Compressible Flow in a Channel with Vibrating Walls-Influence of the Frequency”. *Comp. & Fluids*. 46, 404-410, 2011.

[19] POTTER, M., WIGGERT, D.C., “Mecânica dos Fluidos”. Editora: Thomson, São Paulo, 2004.

[20] RHIE, C.M., CHOW, W.L., “A Numerical Study of the Turbulent Flow Past an Isolated Airfoil with Trailing Edge Separation”. *Am. Inst. of Aero. and Astron.* 21, 1525-1532, 1983.

[21] TANNEHILL J.C., ANDERSON D.A., PLETCHER, R.H. “Computational Fluid Mechanics and Heat Transfer”. Editora: Taylor&Francis. London, 1997.

[22] VIMMR J., “Mathematical Modelling of Compressible Inviscid Fluid Flow Through a Sealing Gap in the Screw Compressor”. *Math. and Comp. in Simul*. 61, 187-197, 2003.

[23] Zdanski, P.S.B., VAZ JR, M., INÁCIO, G.R., “A Finite Volume Approach to Simulation of Polymer Melt Flow in Channels”. *Eng. Comp*. 25, 233-250, 2008.