COMPUTATIONAL MODELING OF NAVIER STOKES EQUATIONS USING A PENALIZED FINITE ELEMENT FORMULATION

Liad Paskin
LAMCE/PEC/COPPE/UFRJ
liadpaskin@gmail.com

José Luis Drummond Alves
LAMCE/PEC/COPPE/UFRJ
jalves@lamce.coppe.ufrj.br

Carlos Eduardo da Silva
LAMCE/PEC/COPPE/UFRJ
kadu@lamce.coppe.ufrj.br

Abstract. *Engineering problems involving fluid flows are widely found in industrial fields, as for example: aeronautics, naval, chemistry, offshore, subsea, among others. Evaluation of physical quantities in a flow, such as pressure and velocities is a common practice in structure and equipment design. There are cases where the flow may exist in a level of complexity that doesn´t fit on the simplifications where analytical models rely, showing an engineering demand for experimental and numerical procedures. In this context, this work presents a numerical modelling technique for fluid mechanics, with particular attention to incompressible, two dimensional flows and Newtonian fluids. A numerical model was developed using a penalized, finite element formulation, able to treat mathematical difficulties of null divergence that translates the kinematic constraints of incompressibility. The implemented code employs quadratic triangular elements with reduced integration used for the penalized term, while full integration is used for the remaining. The convective term of the momentum conservation equation was linearly treated by the Picard method. Time discretization is accomplished by the finite difference method of Crank Nicolson. Finally, numerical applications solved by the proposed model are presented, with particular attention to inner flow in a cavity and outer flow through the von Karman Cylinder. Those applications showed good agreement to the available results in bibliography.*

Keywords: *Penalized method, Incompressible flows, Computational Fluid Dynamics, von Karman Vortex Street, Cavity flow*

1. INTRODUCTION

Fluid mechanics studying is an activity of great importance on the historical scientific and technological development (Eckert, 2007). Filling the medium we live, fluid mechanics is closely-related to other basic engineering studies, like movement. By expanding scientific knowledge about nature, deeper understanding of fluid behavior is acquainted, inducing the exploiting of its proprieties by technological applications. This knowledge is now applied into numerous industrial sectors, e.g., chemical, civil, mechanical, automotive, aeronautic, naval, offshore and subsea. Evaluation of physical quantities in a flow such as, pressure, velocities, temperature, friction, vorticity and kinetic energy is a common practice in the design of structures and equipment that compose those sectors.

The procedures that support fluid mechanics studies are analytical, experimental or numerical. The analytical models that are capable of solving the equations that describes fluid motion are based on simplified hypothesis about its behavior, e.g., homogeneous, symmetrical, non-transient, pure viscous or non-viscous flows. In most of the applications found by engineering, the flow has such a complexity that doesn´t fit on the simplifications on which analytical models rely, showing an engineering demand for experimental and numerical procedures.

A discussion about analytical, experimental and numerical procedures is carried by (Griebel, et al., 1998) that explains how experimental procedures are limited by its technological equipment, mostly incapable of measuring variables that involves extremely high or low, quantities or space-time scales. On these procedures, the number of measures is limited and operational considerations relative to security may also lead to the unavailability of the experiment. Sometimes to become possible, those procedures must assume simplifications that doesn´t describe the real application with sufficient accuracy. Numerical procedures are capable of virtual reproducing those experiments and are taking a bigger part of engineering procedures as processing units becomes more powerful.

By numerical procedures, different methods are capable of approximately solve physical equations by discretizing space and time domains. Among the most adopted methods for solving Navier Stokes equations are the Finite Differences (FDM), Finite Volumes (FVM) and Finite Elements (FEM) methods. Although its use in solid mechanics has stablished FEM as the best approximation method for elliptic problems in arbitrarily complex domains (Gresho & Sani, 1998), historically finite element method has been less applied into computational fluid dynamics (CFD) than FVM and FDM. This historical delay shown by (Gresho & Sani, 1998), that also discusses the advantages of FEM in this context, is being compensated by the new finite elements methodologies being developed to treat fluid simulations difficulties, e.g. the non-symmetric operator that translate convective terms and incompressibility's kinematic constraint.

Among the mathematical difficulties that involve computational fluid dynamics, one of the greatest is related to the kinematic constraint of incompressibility, mathematically expressed by zero divergence of the velocity field (Gresho & Sani, 1998). In order to satisfy this constrain, the approximating field must be constructed by having zero divergence a priori, a characteristic only achieved by solenoids functions, as concluded by (Karam, 1989) who states that the only formulations able to apply the solenoids functions as the approximating field for the Navier Stokes equations are based on streamline functions as the main variable, which is an approach limited to two-dimensional problems that requires the velocity field to be post-processed by the derivatives of the solution, compromising its precision. Lagrange Multipliers technique is an alternative to this problem, leading to the Galerkin`s mixed method. Another alternative is the penalty formulation, adopted herein and described by (Hughes, et al., 1979) as the "*simplest, effective, finite element implementation of incompressibility*".

On the mathematical constrained models, as the case of incompressibility, the proprieties of the continuum model are not transferred to the discrete model (Karam, 1989), meaning that existence and uniqueness conditions of the discrete model given by Brezzi theorem (Brezzi, 1974) must be evaluated for each approximating field chosen. The penalty method substitutes the incompressible model for a slightly compressible one, turning unnecessary to attend a priori the incompressibility constraint. However, convergence of the discrete model isn´t automatically achieved, demanding the reduced integration technique on the penalized term, which was shown by (Malkus & Hughes, 1978) to be mathematically equivalent to the mixed methods.

This paper presents a methodology of numerical modelling fluid dynamics by the finite elements method, applied to incompressible two-dimensional flows of Newtonian fluid. This paper employs triangular quadratic elements. Reduced integration evaluates the penalized term while full integration evaluate the others, as shown by (Hughes, et al., 1979) for the case of biquadratic element application. The convective term on the momentum conservation equation is linearly treated by the Picard iteration method (Engelman, et al., 1981). Time stepping is achieved by the finite difference method of Crank-Nicolson (Hughes, 1987).

On the next section, governing equations are introduced. The most relevant algorithms adopted by the presented methodology are shown in section three. On the fourth section, two test cases are presented with the primarily objective of demonstrating the capabilities of this methodology while evaluating its implementation. Finally conclusions will be expressed on the fifth section.

2. GOVERNING EQUATIONS

2.1. Newtonian fluid and incompressible flow

Let $\Omega \subset R^{n_{sd}}$ be the spatial domain, where n_{sd} is the number of spatial dimensions, let Γ denote the boundary of Ω and the interval $[0,t_f]$ denote the time domain of the problem. We consider the Navier-Stokes equations governing the incompressible flow of a Newtonian fluid in a differentiable form and an Eulerian description defined over the space-time domain $\Omega\times [0,t_f]$:

$$\rho\ (\partial \boldsymbol{u}/\partial t + \boldsymbol{u}\cdot\nabla \boldsymbol{u} - \boldsymbol{b}) - \nabla \cdot \boldsymbol{\sigma} = 0, \tag{1}$$

$$\nabla \cdot \boldsymbol{u} = 0, \tag{2}$$

where ρ and $\boldsymbol{u}$ are the density and velocity field, respectively, and $\boldsymbol{b}$ represents the force vector carrying the gravity acceleration effect. Equation (1) represents the conservation of momentum and Eq. (2) the conservation of mass, also called continuity equation and here representing the kinematic constraint of incompressibility. The Cauchy stress tensor $\boldsymbol{\sigma}$ for a Newtonian fluid in an incompressible flow is defined as:

$$\boldsymbol{\sigma} = - p\, \boldsymbol{I} + 2\, \mu\, \boldsymbol{D}, \tag{3}$$

where p is the pressure, $\boldsymbol{I}$ the identity tensor, μ the dynamic viscosity and $\boldsymbol{D}$ the rate of deformation tensor given as:

$$\boldsymbol{D} = 1/2\cdot[\ \nabla \boldsymbol{u} + (\nabla \boldsymbol{u})^T\ ] \tag{4}$$

2.2. Penalized formulation

The penalized formulation of the Navier-Stokes equations is achieved by stating a relationship between the pressure and the velocity field, such that:

$$p = - \lambda\ (\ \nabla\cdot \boldsymbol{u}\ ) \tag{5}$$

where λ is the penalty parameter. It may be seen from Eq. (5) that for maintaining finite pressure while setting λ to infinite, the divergence of the velocities field must be zero. Convergence of the penalty formulation applied to the Navier-Stokes equations has been proved by (Temam, 1977). Parameter λ must be chosen sufficiently large so that the compressibility introduced will be minimum, but not so big that numerical ill conditioning ensues. According to (Hughes, et al., 1979), the parameter λ might be given as:

$$\lambda = c\,\mu \tag{6}$$

where c is a constant depending on the computer word length. According to (Hughes, et al., 1979), numerical studies reveal that for a floating-point word length from 60 to 64 bits, an appropriate choice for c is 10^7.

Applying the relations between the elastic constants given by (Shames & Cozzarelli, 1997), an expression relating the parameter λ, the dynamic viscosity μ and the Poisson coefficient ν might be stated:

$$\lambda = 2\,\mu\,\nu \,/\, (\,1 - 2\,\nu\,) \tag{7}$$

In the present formulation ν and μ are the parameter setted by the user, while λ is post processed from Eq. (7). On the following test cases: $\nu = 0.5\ (\,1 - 10^{-7}\,)$, corresponding to $c = 10^{7}$.

By introducing the penalty formulation on the Navier-Stokes equations, we relate the pressure to the velocities field, thus abandoning the need of satisfying the continuity equation. The equations are solved for the velocities field, from which the pressures are post processed.

3. SOLUTION TECHNIQUE

Finite element spatial discretization applied to Eq. (1) leads to:

$$\boldsymbol{M}\dot{\boldsymbol{u}} + \boldsymbol{K}(\boldsymbol{u})\,\boldsymbol{u} = \boldsymbol{F} \tag{8}$$

where $\boldsymbol{M}$ is the mass matrix, $\boldsymbol{K}\,(\boldsymbol{u})$ the non-linear stiffness matrix, $\boldsymbol{F}$ the force vector and $\dot{\boldsymbol{u}}$ the local acceleration field.

3.1. Time stepping method

The Crank-Nicolson method is part of a more general family of time stepping methods called the "generalized trapezoidal family of methods" (Hughes, 1987). Stating Δt as the time interval between each time step, and the subscript i standing for the current time step, considering $\alpha = 0.5$ on the next equations leads to the Crank-Nicolson method on the form it was implemented. This is achieved by first calculating $\boldsymbol{u}_{i+\alpha}$, called the predictor of $\boldsymbol{u}_{i+1}$:

$$\boldsymbol{u}_{i+\alpha} = \boldsymbol{u}_i + (1 - \alpha)\,\Delta t\,\dot{\boldsymbol{u}}_i \tag{9}$$

$$\boldsymbol{u}_{i+1} = \boldsymbol{u}_{i+\alpha} + \alpha\,\Delta t\,\dot{\boldsymbol{u}}_{i+1} \tag{10}$$

Substituting Eq. (10) in Eq. (8) evaluated at time step $i+1$, leads to a system that can be solved for the current acceleration field $\dot{\boldsymbol{u}}_{i+1}$:

$$(\boldsymbol{M} + \alpha\,\Delta t\,\boldsymbol{K})\dot{\boldsymbol{u}}_{i+1} = \boldsymbol{F}_{i+1} - \boldsymbol{K}\boldsymbol{u}_{i+\alpha} \tag{11}$$

After obtaining the acceleration field, the velocities field can be updated by Eq. (10).

3.2. Equations solving

The system given by the Eq. (11), must be solved for $\dot{\boldsymbol{u}}_{i+1}$. A direct approach was taken, implementing the Crout Factorization (Hughes, 1987). Mesh reordering uses Cuthill Mckee algorithm for matrix band width reduction.

3.3. Nonlinearity evaluation

Picard iteration method (Engelman, et al., 1981) is used and the stiffness matrix of Eq. (8) is evaluated considering the already known velocities field taken from the previous time step. Applying the subscript i that indicate the current time step leads to the time discretized equation on the form:

$$\boldsymbol{M}\dot{\boldsymbol{u}}_{i+1} + \boldsymbol{K}(\boldsymbol{u}_i)\,\boldsymbol{u}_{i+1} = \boldsymbol{F} \tag{12}$$

This method is modified by the current methodology where a criterion based on the Courant Number (CFL) avoids matrix assembly and factorization at every time step. The system will only be assembled and factorized if the maximum

CFL of the current step is greater than the maximum CFL of the previous step, otherwise, only back-substitution takes place. The maximum CFL is evaluated element by element:

$$CFL_{max} = \max_{e} \left\{ u_{ei} \cdot \frac{\Delta t}{\Delta x_e} \right\} \ , e=1, n_{el} \tag{13}$$

where e denote the element identification, Δx_e the characteristic length of the element and u_{ei} its characteristic velocity on time step i. The characteristic length is taken as the average of the element`s edges lengths and the characteristic velocities considered are evaluated as follows:

$$u_{ei} = \frac{1}{3} \sum_{n_c=1}^{3} \sqrt{u_{1n_c}^2 + u_{2n_c}^2} \tag{14}$$

where n_c represents each one of the corners of the triangular element and u_{1n_c} and u_{2n_c} the components of the velocity vector evaluated at this corner.

4. TEST CASES

In this section, are presented two test cases with the purpose of establishing the validation and capabilities of the proposed methodology. The first case considered is that of a two-dimensional cavity, having the inner flow induced by the top edge movement. The second case is about an outer flow through a cylinder, giving rise to the well-known von Karman vortex street.

4.1. Lid-driven cavity

Here, the motion of an incompressible viscous fluid is induced inside the cavity showed on the Figure 1 by the movement of its top edge, while all other walls are held motionless. The cavity has unitary sides and the boundary conditions of the flow are given by:

$$\begin{aligned} &On\ \{x_1=0,\ or\ x_1=1,\ or\ x_2=0\} \rightarrow \{u_1=0;\ u_2=0\} \\ &On\ \{x_2=1\} \qquad\qquad\qquad\qquad\quad\ \rightarrow \{u_1=1;\ u_2=0\} \end{aligned} \tag{15}$$

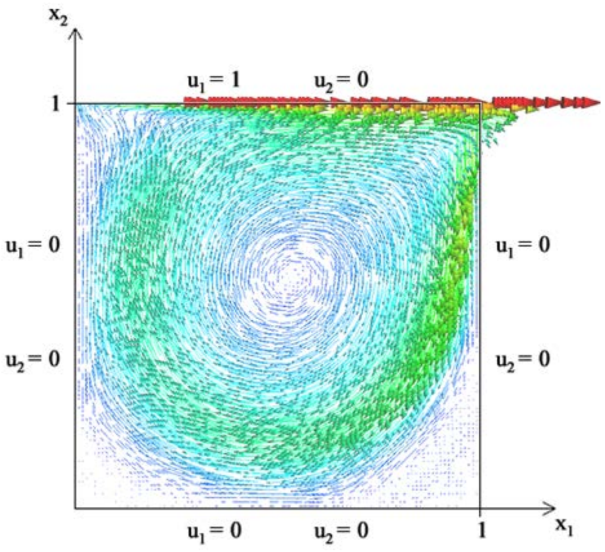

Figure 1. Cavity model geometry and boundary conditions superposed to the velocity profile obtained as the solution for Re 1000.

The uniform mesh considered was constructed by subdividing each one of the model´s edges into 100 equally spaced cells, leading to 100×100 squares that were subdivided each one into two quadratic triangular elements. Thus, it has 40,401 nodes and 20,000 elements. Reynolds number is given for this model considering both the characteristic length and the characteristic velocities equal to 1.

The steady-state result obtained for this model at Reynolds 1000 is compared to the one published by (U.Ghia, et al., 1982) at the Figure 2, where the results here obtained are superimposed to the other. This comparison has showed great agreement between both models.

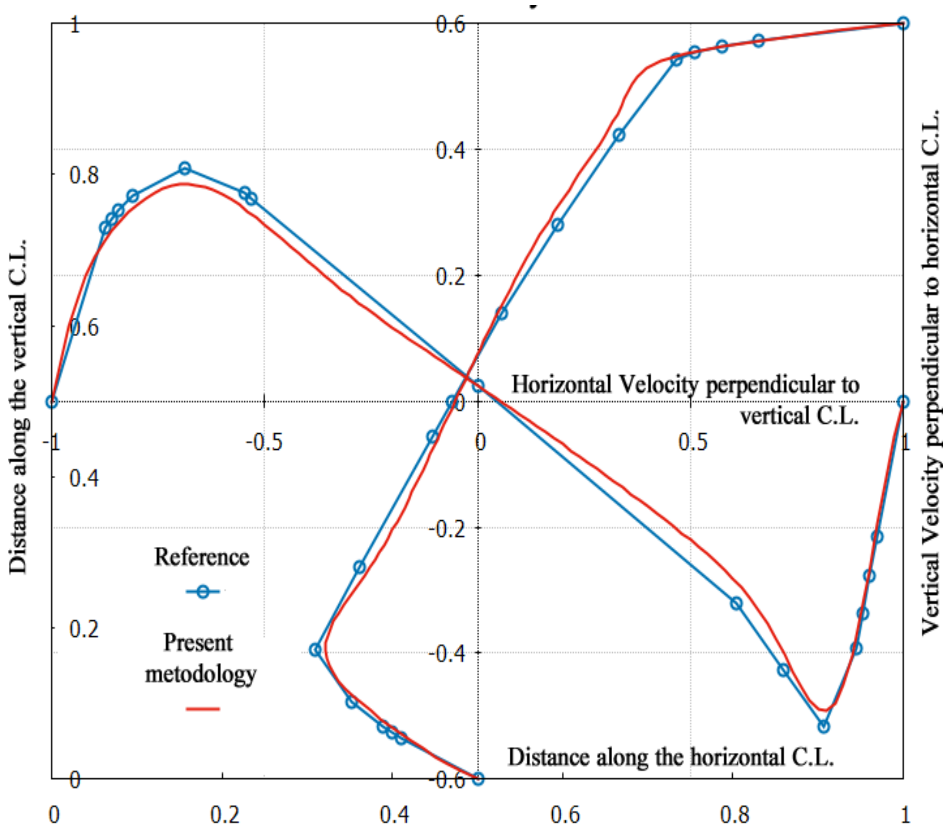

Figure 2. Profiles of perpendicular velocities along center lines (C.L.) of the lid-driven cavity model. Re 1000 compared to (U.Ghia, et al., 1982).

Steady state`s streamlines obtained by the present formulation and (U.Ghia, et al., 1982) are illustrated in Figure 3.

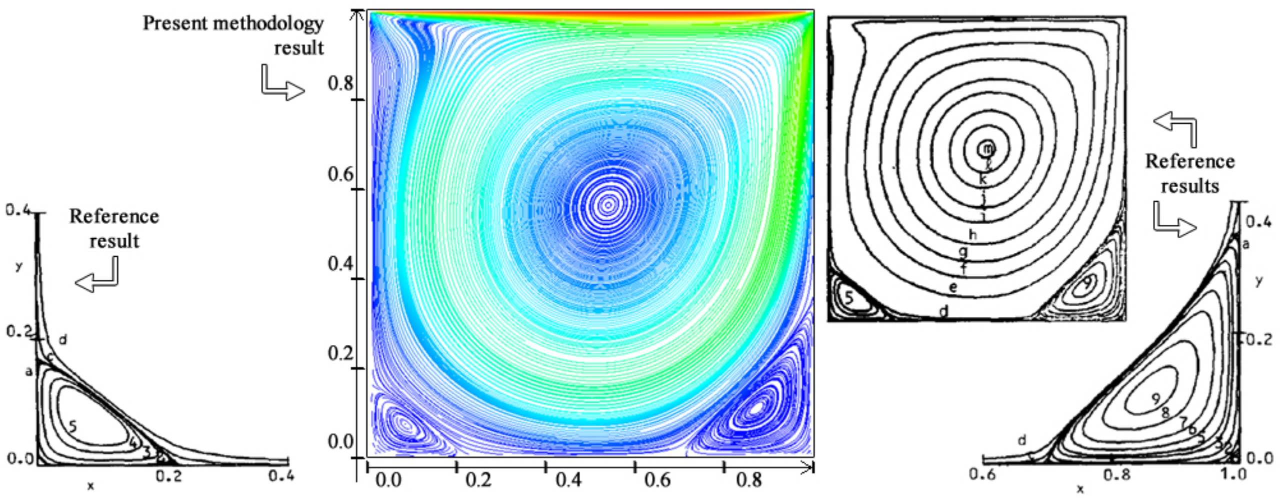

Figure 3. Streamlines obtained from the solution of the lid-driven cavity model. Re 1000 compared to (U.Ghia, et al., 1982).

4.2. Flow past a circular cylinder

According to (Brooks & Hughes, 1982) this is "*one of the most challenging problems for numerical solution methods*", since "*all of the terms in the governing equations are significant*". It consists of a circular cylinder immersed in a flowing viscous fluid. As the fluid starts to flow past the cylinder, a pair of symmetrical eddies forms downstream. When the flow reaches higher Reynolds numbers those eddies become unstable and are carried downstream with the flow, giving rise to the von Karman vortex street, a condition observed on the following results.

The model, its dimensions and boundary conditions are showed in Figure 4. Left edge represents the inlet, with unitary velocity being prescribed perpendicular to the edge. Right edge receives outlet condition. Top and bottom edges get no-flow boundary condition, while the cylinder gets no-flow and no-slip boundary condition.

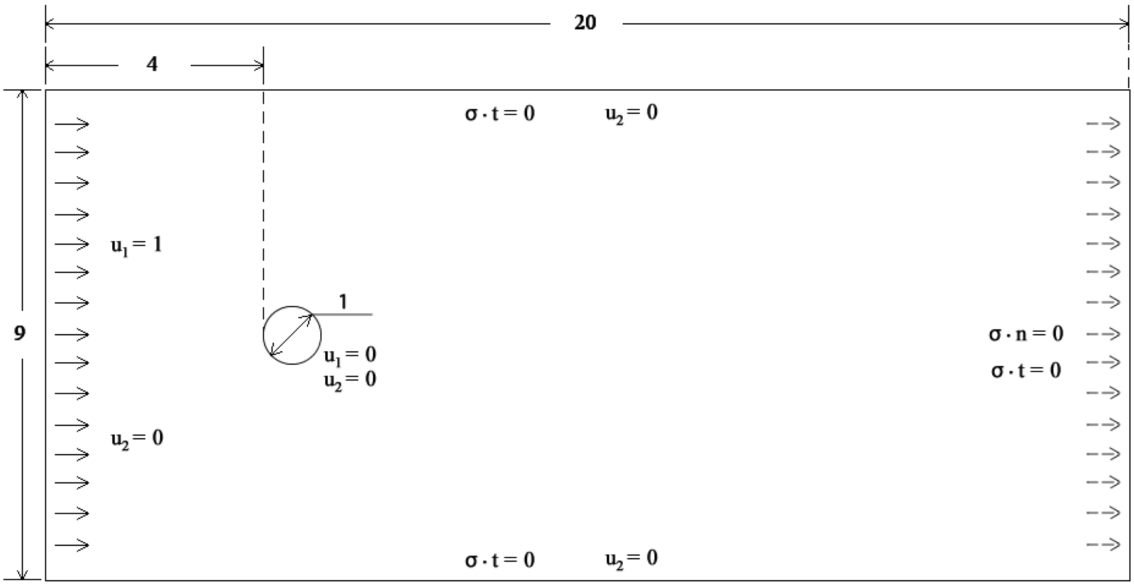

Figure 4. von Karman model geometry and boundary conditions

The structured mesh here considered is shown in Figure 5. It has 45,296 nodes and 22,400 elements. The time interval adopted is 0.001 time units.

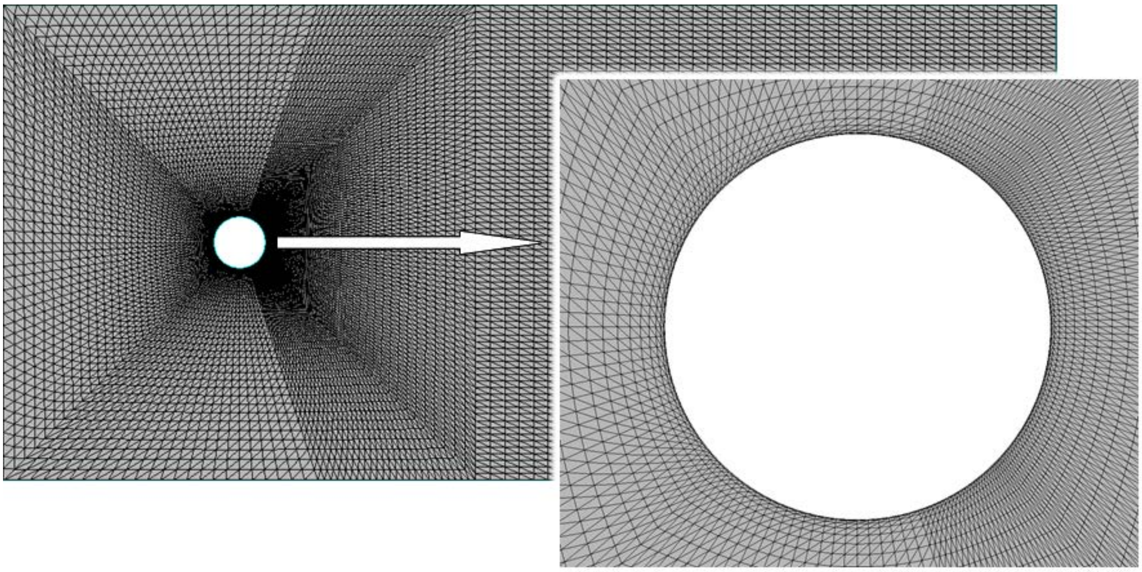

Figure 5. von Karman model mesh

Reynolds number is given for this model considering both the characteristic length and the characteristic velocities equal to 1. Following results have been observed for Reynolds 100.

The Period of the vortex shedding frequency on the present formulation results is 6.13 time units, close from the empirical data that predicts 6.25 time units.

In order to validate the amplitudes of the velocity field`s results obtained, the same model was simulated in the commercial Finite Volume Method`s software, Fluent. Figure 6 shows the comparison between the solutions obtained by Fluent and by the proposed methodology in four graphs. Each one of those graphs represents a different point, positioned on the horizontal center line of the domain, where the vertical velocity is plotted against time. Results shows good agreement between the amplitudes of the velocities obtained at each point. The frequency of the signal differs substantially between the models though. The period observed for the vortex shedding frequency obtained by Fluent was 8.36 time units. The difference observed between the results on the first graphic, i.e., the one that represents the point closest from the inlet, might be consequence of quadratic element interpolation at the present formulation that introduces small disturbances on the inlet. For its magnitude in comparison to the other graphs, this difference have been considered a small error that should be explored by future works.

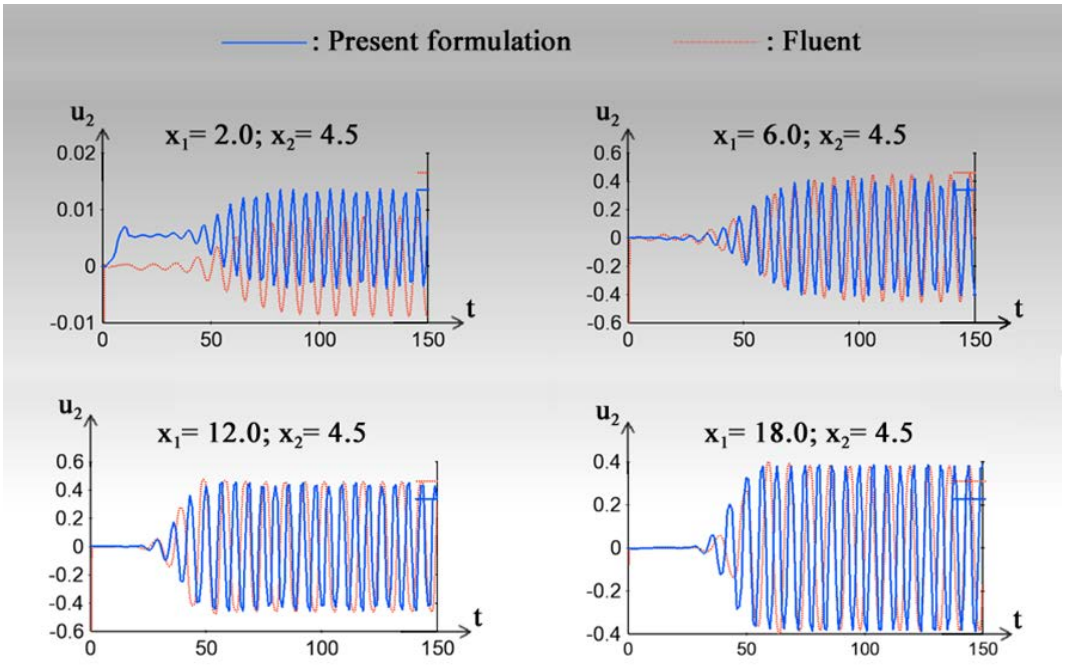

Figure 6. von Karman model results comparing to Fluent. Reynolds 100

Figure 7 illustrate the results obtained for Reynolds 100, coloring the domain by velocity magnitude and tracing contour lines on equal velocities spots.

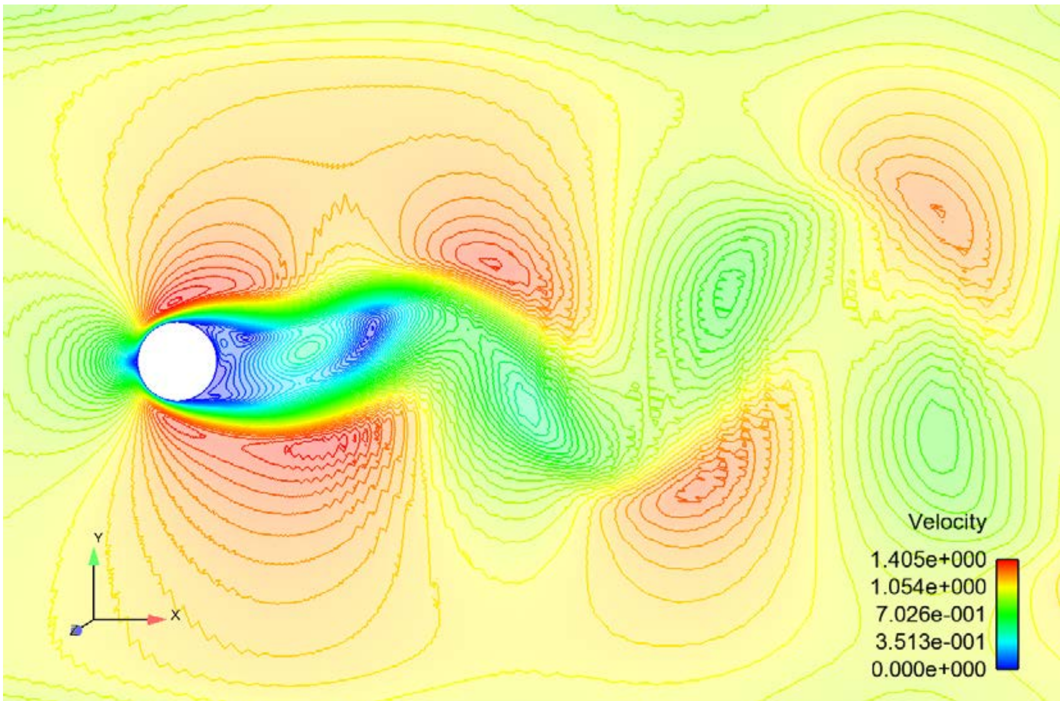

Figure 7. von Karman model, Re 100 results

Figure 8 relates the maximum CFL number observed in each step with the correspondent time. Each one of the time steps where factorization occurs is marked on the curve. It may be seen that factorization occurs whenever CFL gets higher. Since the factorization procedure is by far the most time consumable, this methodology has led to relevant time saving, without sacrificing its reliability.

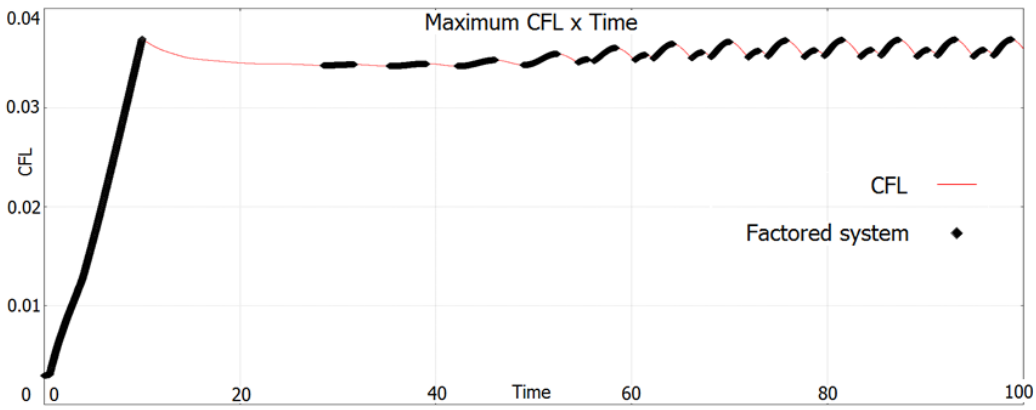

Figure 8. Maximum CFL against time and the time steps which occurs factorization. von Karman model, Re 100.

5. CONCLUSIONS

This paper presents a penalized finite element methodology capable of modeling two-dimensional incompressible flows of a Newtonian fluid. Fluid mechanics was contextualized for its importance in engineering, as well as its computational modeling. The main difficulties about computational fluid dynamics were introduced as well as the methods that deal with it. The solution techniques used by this methodology were described.

The main objective achieved was to develop a formulation without the use of parameter settings of stabilization for fully convenient and consistent numerical modeling of Navier-Stokes equations. The results show good agreement with those found in literature, demonstrating some of the capabilities of the chosen methods. The developed method for nonlinearity evaluation showed effective in saving CPU time. The criterion chosen for its implementation is considered conservative as it introduces no visible differences in the solution. However, this criterion should be further investigated as it can be even more effective, i.e., reducing factorization steps while preserving solution quality.

Future work involves further developments in the code, extending its capabilities to address two-fluid flows, fluid-structure interaction and three-dimensional problems.

6. BIBLIOGRAPHY

Brezzi, F., 1974. On the existence, uniqueness and approximation of sadle-point problems arising from lagrange multipliers. *ESAIM: Mathematical Modelling and Numerical Analysis - Modélisation Mathématique et Analyse Numérique*.

Brooks, A. N. & Hughes, T. J., 1982. Streamline upwind/Petrov-Galerkin formulations for convection dominated flows with particular emphasis on the incompressible Navier-Stokes equations. *Computer methods in applied mechanics and engineering*.

Eckert, M., 2007. *The dawn of fluid dynamics: a discipline between science and technology*. s.l.:John Wiley & Sons.

Engelman, M., Strang, G. & Bathe, K. J., 1981. The application of quasi-Newton methods in fluid mechanics. *International Journal for Numerical Methods in Engineering*.

Gresho, P. M. & Sani, R. L., 1998. *Incompressible flow and the finite element method*. New York: John Wiley and Sons, Inc..

Griebel, M., Dornseifer, T. & Neunhoeffer, T., 1998. *Numerical simulation in fluid dynamics: a practical introduction*. Philadelphia: Society for Industrial and Applied Mathematics.

Hughes, T. J. R., 1987. *The finite Element method: linear static and dynamic finite element analysis*. Englewood Cliffs, N.J.: Prentice-Hall, Inc..

Hughes, T. J. R., Liu, W. K. & Brooks, A., 1979. Finite element analysis of incompressible viscous flows by the penalty function formulation. *Journal of Computational Physics*.

Karam, F. J., 1989. *Uma nova formulação de elementos finitos mistos para escoamentos incompressiveis*. Rio de Janeiro(RJ): Universidade Federal do Rio de Janeiro.

Malkus, D. S. & Hughes, T. J. R., 1978. Mixed finite element methods - reduced and selective integration techniques: a unification of concepts. *Computer Methods in Applied Mechanics and Engineering*.

Shames, I. H. & Cozzarelli, F. A., 1997. *Elastic and Inelastic Stress Analysis*. Philadelphia: Taylor & Francis Ltd..

Temam, R., 1977. Navier-Stokes Equations. *North Holland*.

U.Ghia, Ghia, K. & Shin, C. T., 1982. High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method. *Journal of Computational Physics*, 15 January.

7. RESPONSIBILITY NOTICE

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