ON THE USE OF STABILIZATION SCHEMES IN COLLOCATED GRIDS FOR THE INCOMPRESSIBLE NAVIER-STOKES EQUATIONS

Gustavo Trindade Choaire
Antonio Fábio Carvalho da Silva
Clovis R. Maliska
Gustavo Paul Exel
Universidade Federal de Santa Catarina
gtchoaire@gmail.com, afabiocs@gmail.com, clovis.maliska@gmail.com, gustavoexelgpe@gmail.com

Abstract. *The solution of the momentum conservation equations coupled with the mass conservation equation for incompressible flows plays an important role on Computational Fluid Dynamics (CFD). Numerous studies have shown different ways of dealing with the pressure-velocity coupling inherent to these set of equations while using segregated methods of solution and collocated variables. It is well documented that the pressure checkerboard problem arises when the collocated variable arrangement is used, since the nodal velocities are not located where they are need for the mass conservation balance. Thus, several stabilization schemes are reported in the literature with the purpose of preventing the oscillatory pressure fields. It is not clear, however, if these stabilization schemes are really needed, opposed to a simple nodal averaging velocity, if the system is solved simultaneously. It is fully reported in the literature the need of introducing the pressure in the mass conservation equation to avoid a zero diagonal in the linear system of equation. The first part of this study, shown in this paper, is concerned with the analysis of using weighted average of nodal velocities or a stabilization scheme for calculating the convecting velocity (mass flow) in conjunction with simultaneous solution. For the analysis, as prototype of a stabilization scheme, the Physical Influence Scheme (PIS) is used. The bidimensional lid-driven cavity flow problem, a widely known benchmark, was used. The results shows a good agreement of the velocity fields using average nodal velocities for the calculation of the advecting velocities. The results were also compared with the staggered grid arrangement, known for eliminating the spurious pressure oscillations.*

Keywords: *Incompressible Flow, Collocated grid, Fully coupled, Weighted average velocities, Cell-face velocities*

1. INTRODUCTION

In Computational Fluid Dynamics (CFD) the Incompressible Navier-Stokes (INS) model is a well-known mathematical model composed by the momentum and mass conservation equations. In the numerical solution, this model may have its unknowns represented through the collocated grid, where all variables were displayed at the center of the control volume. This introduces a key difficulty in performing the mass balance, since the velocity which calculates the mass flow at the boundaries of the control volume does not exist. In addition, the fact that pressure and velocities were held together at the center, and both variables needed to be interpolated at the faces of the control volume, the balance for pressure would not include the nodal pressure of that particular volume as shown in Fig. 1. The balance for the control volume centered in P is described in Eq. 1. Therefore, the presence of oscillatory pressure fields may arise and it was first reported in (Patankar, 1980).
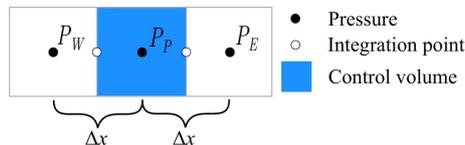

Figure 1. Control volume for pressure.

$$P_e - P_w = \frac{P_E + P_P}{2} - \frac{P_W + P_P}{2} = \frac{P_E - P_W}{2}. \tag{1}$$

Harlow and Welch (1965) introduced a strategy for solving this difficulty, introducing the staggered grid in which all variables have their own control volume, placing the velocities at the faces of the control volume, contributing for stabilizing the pressure fields. Despite the high accuracy of the scheme, it brings difficulties when unstructured grids and, especially, tridimensional problems, are solved, due to index control and storage aggravation. Figure 2 presents the two different variable arrangements.
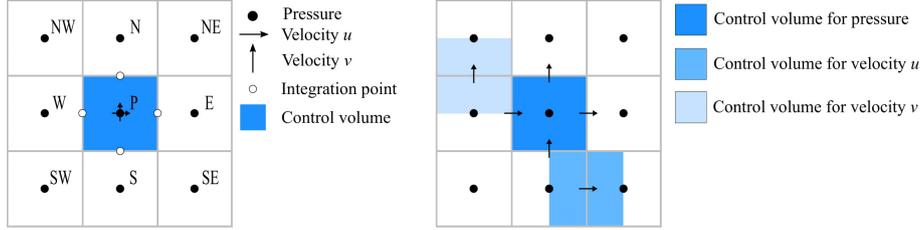
Figure 2. Collocated and staggered grids.

The difficulties encountered by using the staggered scheme forced the numerical analysts to be back to the co-located arrangement of variables, creating stabilization methods to fulfill the role of preventing oscillatory pressure fields. Schneider and Raw (1987) presented the Physical Influence Scheme (PIS), that consists of using the momentum equations as auxiliary equations for approximating cell-face velocities. This choice is based on the fact that this auxiliary equation carries all the physical information of the problem.

While segregated solution of the equation set was used mainly because of the lack of computer resources, the always increasing computational capabilities allowed the development of schemes using simultaneous solution of the equations, no longer requiring algorithms for pressure-velocity coupling. Differently from segregated methods, the simultaneous solutions solves all variables simultaneously providing the required coupling. Nowadays the number of studies employing simultaneous solution is constantly increasing (Alisadeghi and Karimian, 2011a);(Alisadeghi and Karimian, 2011b);(Honório, 2013). Despite of using a simultaneous solutions approach, if a co-located arrangement is used, the need of calculating the convecting velocity at the control volume interface still remains. It is intensively reported in the literature that a simple average of the nodal velocities to calculate the interface velocities (convecting velocity) does not suffic, and stabilizing schemes which introduces the pressure in the mass conservation equation must be devised. Preliminary results obtained in the *Laboratório de Simulação Numérica em Mecânica dos Fluidos e Transferência de Calor* (SINMEC), evaluating the convecting velocity by a simple average of the node velocities weighted by the distance from the interface, and calculating the convected velocity, which requires an interpolation function, using a CDS scheme were promising (Dal Pizzol and Maliska, 2012). Excellent results were achieved on a cartesian equally spaced grid. This raised the attention of discussing the need of stabilization schemes on collocated arrangements when simultaneous solution is employed. At the first glance it looks that the use of co-located arrangement with segregated solution, and the convecting velocities by averaging the nodal velocities, could be the factor requiring stabilization schemes. For trying to bring some lights to these doubts is the main contribution of this paper. In this way, both formulations with and without stabilization schemes are presented in Sec. 2, following in Section 3, the introduction of the numerical method employed for discretizing the governing equations on Cartesian grids. In Section 4, a benchmark is performed in order to validate both formulations. Finally, Section 5, points out few important remarks.

2. MATHEMATICAL MODEL

The mathematical model comprises the bidimensional transient momentum and mass conservation equations for a incompressible fluid. The mathematical model reads,

$$\frac{\partial}{\partial t}(\rho \mathbf{v}) + \nabla \cdot (\rho \mathbf{v}\mathbf{v}) = -\nabla \cdot \overline{\overline{P}} + \nabla \cdot (\mu \nabla \mathbf{v}),$$ (2)

$$\nabla \cdot \mathbf{v} = 0,$$ (3)

in which $\mathbf{v}$ is the velocity vector, ρ is density, μ is viscosity and $\overline{\overline{P}}$ is the pressure tensor represented by,

$$\overline{\overline{P}} = \left[\begin{array}{c} \mathbf{P}^x \\ \mathbf{P}^y \end{array} \right] = \left[\begin{array}{cc} p & 0 \\ 0 & p \end{array} \right].$$ (4)

The momentum equations are known by its nonlinear features clearly seen in the second term of Eq. 2. These two velocities play different roles in the numerical solution, the first one being the convecting velocity which represents the mass flow, while the second one is the convected velocity, that is, the transported variable. Indeed, it is important to highlight that the approach used to evaluate convecting velocities must be the same adopted in the mass conservation equation, given the fact that those are the same mass conserving velocities. It looks obvious, but this is not always followed when developing numerical schemes for fluid flows solution.

2.1 The algorithms

As already mentioned, the convecting velocities represents the mass conserving fluxes that transports a certain convected variable. Therefore, the proposed work consists of two different approaches for evaluating cell-face velocities

referred, herein as the convecting term. Firstly, employing a weighted average for evaluating mass conserving velocities, taking into account the distance between nodal point variables. In the cartesian grid, it is simple matter to obtain the expression for the cell-face velocity. Secondly, using the PIS – Physical Influence Scheme, a stabilization scheme, which plays the role of stabilizing the pressure field, since the general belief is that a simple averaging of the nodal velocities does not suffice. The expressions for the integration points (points at the surface) in collocated cartesian grid can be read as follows according to Fig. 3,

$$u_e = \frac{U_P + U_E}{2}, \tag{5}$$
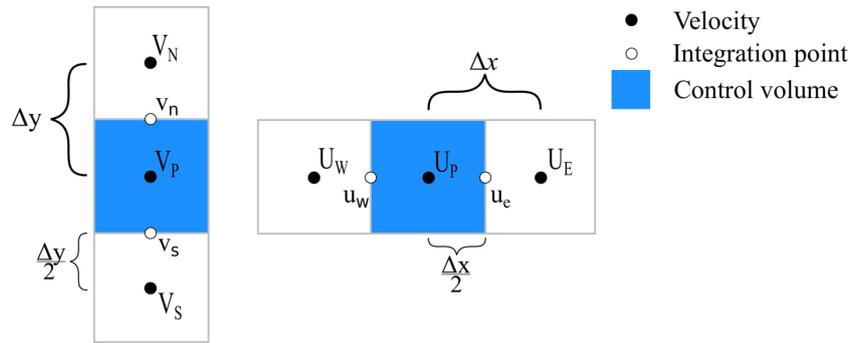
$$v_n = \frac{V_P + V_N}{2}. \tag{6}$$

Figure 3. Collocated arrangement of variables for Cartesian grid.

The second algorithm consists on applying the PIS scheme to obtain the convecting velocity, what provides the appearance of pressure in the mass conservation equation, an argument strongly stated as the reason for avoiding pressure instabilities. The PIS scheme uses the momentum conservation equation as an auxiliary equation, given by

$$\rho |\mathbf{V}| \frac{\partial \mathbf{v}}{\partial s} + \frac{\partial P}{\partial x_i} - \mu \nabla^2 \mathbf{v} = 0, \tag{7}$$

where $|\mathbf{V}|$ is represented in streamwise direction, as follows,

$$|\mathbf{V}| = \sqrt{(u^2 + v^2)}. \tag{8}$$

In order to evaluate the velocities upstream the SUDS-NO (Skew Upwind Difference Scheme-Node) introduced by (Souza, 2000) is employed.

3. NUMERICAL FORMULATION

3.1 Average of Nodal Values for the Convecting Velocities

The calculation of the mas flow involves, of course, the use of the convecting velocity at the boundaries of the control volume. This calculation can be made through a weighted average of the nodal values, what means that pressure will not be involved in the mass conservation, or using a so-called stabilization scheme, which, at the end, introduces pressure in the mass conservation equation. Putting this clear, it is understood that the pressure present in the mass conservation equation introduces the required ingredient to avoid pressure oscillations. The evaluation of mass fluxes (involving the convecting velocities) at the boundaries of the control volume using weighted average results in the following expression for example, at the east face of the control volume centered at P in Fig. 2,

$$\dot{m}_e = \rho u_e \Delta y. \tag{9}$$

The velocity in the east integration point is approximated according to Eq. 5,

$$\dot{m}_e = \rho \left(\frac{U_P + U_E}{2} \right) \Delta y. \tag{10}$$

3.2 PIS - Physical Influence Scheme

On the other hand, employing the PIS stabilization scheme at the integration point equations according to Eq. (8), it results the following expressions for cartesian grid,

$$u_e = \left(\frac{P_P - P_E}{\Delta x} \right) \frac{1}{C} + \frac{\mu}{C} \left(\frac{4U_E + 4U_P}{\Delta x^2} + \frac{2U_P + 2U_E + U_N + U_S + U_{NE} + U_{SE}}{\Delta y^2} \right) + \rho |\mathbf{V}| \left(\frac{U_{ups}}{\Delta s_e} \right) \frac{1}{C}, \quad (11)$$

where all nodal variables are known from the previous nonlinearity iteration. Differently from the momentum equations, nodal variables that arise from integration point expressions in the continuity equation are not known from previous time iteration and terms which multiply those variables must be coefficients. The C coefficient is determined by,

$$C = \left(\frac{\rho |\mathbf{V}|}{\Delta s_e} + \frac{8\mu}{\Delta x^2} + \frac{8\mu}{\Delta y^2} \right). \quad (12)$$

In the global matrix of variables, every line from the matrix addresses the coefficients corresponding to each particular equation, in which the first upper index designates the model equation and the other relates the variable included in that equation. As the first algorithm, the one which uses the average of the nodal velocities, the pressure does not appear on the continuity equation leading to a zero diagonal according to Eq. 13.

$$
\begin{matrix}
Momentum_x & \longrightarrow \\
Momentum_y & \longrightarrow \\
Continuity & \longrightarrow
\end{matrix}
\begin{bmatrix}
A^{uu} & 0 & A^{up} \\
0 & A^{vv} & A^{vp} \\
A^{pu} & A^{pv} & 0
\end{bmatrix}_{3N_x X 3N_y}
\begin{bmatrix}
U \\
V \\
P
\end{bmatrix}_{3N_x N_y}
=
\begin{bmatrix}
B^u \\
B^v \\
0
\end{bmatrix}_{3N_x N_y}, \quad (13)
$$

where N_x and N_y represents the number of volumes in the x and y direction respectively. The employment of PIS as a stabilization scheme contribute to a larger stencil together with pressure terms emerging in the continuity equation. The appearance of a new submatrix is highlighted in Eq. 14, providing a complete fulfillment of matrix diagonal.

$$
\begin{matrix}
Momentum_x & \longrightarrow \\
Momentum_y & \longrightarrow \\
Continuity & \longrightarrow
\end{matrix}
\begin{bmatrix}
A^{uu} & 0 & A^{up} \\
0 & A^{vv} & A^{vp} \\
A^{pu} & A^{pv} & A^{pp}
\end{bmatrix}_{3N_x X 3N_y}
\begin{bmatrix}
U \\
V \\
P
\end{bmatrix}_{3N_x N_y}
=
\begin{bmatrix}
B^u \\
B^v \\
B^p
\end{bmatrix}_{3N_x N_y}. \quad (14)
$$

4. RESULTS

Regarding the software implementation technique, both algorithms were developed in Python through Spyder (Raybaut, 2009), which is an open-source cross-platform for scientific programming. Also, plots shown in this section were assembled using *Matplotlib* (Hunter, 2007). In order to evaluate the different algorithms for the convecting velocities calculation, and also analyze the behavior of pressure fields with and without stabilization schemes, the well-known lid-driven cavity benchmark is appropriate, since it contains all the numerical details one wants to analyse. Figure 4 illustrates the 1 *m* x 1 *m* cavity and its boundary conditions. The Reynolds parameter is determined by the expression in the same figure, density (ρ) is defined as $1kg/m^3$, velocity at the top $U = 1$ for all cases and therefore viscosity follows the expression $\mu = \frac{1}{Re}$.

Figure 4. Lid-driven cavity.

The linear system was solved using a direct sparse solver provided by SciPy, which is a Python-based open-source software for mathematics, science and engineering. The solver adopt LU decomposition to determine the solution of

sparse linear system $Ax = B$, where A is the matrix illustrated in Eqs. 13 and 14, x is the solution vector and B is the independent term. Also, all results should reach the steady state and therefore it is not necessary to achieve convergence at each time level. In this way, the nonlinearities were uptaded only once at each iteration. Lastly, the convergence criterion was defined by the following expression,

$$\in = \max(S - S^o), \tag{15}$$

where S is the current solution vector and S^o refers to the previous time level solution. The tolerance to achieve the steady state was set to $1x10^{-6}$ and all algorithms reached this parameter.

4.1 Velocity fields

The results for velocity were achieved employing different grids and Reynolds Number parameter, investigating distinct flow behaviors. The staggered grid solution was used as reference solution, since it is known not to produce oscillations. In this clean problem one is sure there are no other possibilities of introducing instabilities by any other numerical pathology. The solution was obtained for Reynolds number values of 100, 400 and 1,000. Additionally, there were also three different grids used in each Reynolds case, consisting on meshes with 900, 3600 and 4900 volumes for the results shown in Fig. 7, compared with the solutions presented by (Ghia *et al.*, 1982).
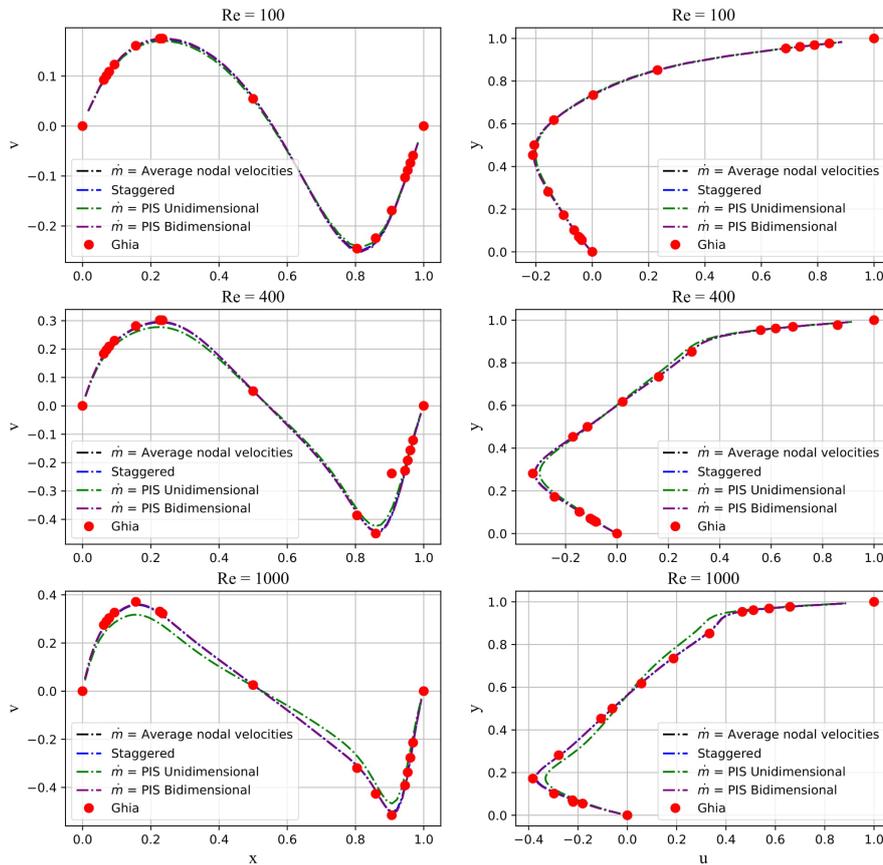
Figure 5. Velocity along horizontal and vertical lines in $y = 0.5m$ and $x = 0.5m$ respectively.

The results agree very well with the staggered arrangement even for higher Reynolds, validating the approach of calculating the convective velocity by the weighted averaging of nodal velocity values. This confirms the original findings of the research group at SINMEC, findings that motivated the present study in arguing if stabilization schemes are really needed (Maliska and Honório, 2019); (Maliska *et al.*, 2017). By its turn, the bidimensional stabilization scheme also provided good results, and as expected proved to perform better than the unidimensional one. Still on this figure, it is important to mention that in all results a CDS interpolation function was employed for the convected velocities.

4.2 Pressure fields

Another important analysis consists in evaluating the effect on pressure fields, taking into account the approximation made for the pressure gradients. As can be seen in Figure 6, the staggered algorithm produced a well-behaved pressure field confirming the stability provided by this arrange. On the other hand, even with a remarkable well calculated velocity profile, the use of the average of the nodal velocities still presents minor oscillatory pressure fields confirming the existence of pressure oscillations, what was firstly discussed in (Patankar, 1980). In the other hand, both stabilization schemes, PIS 1D and PIS 2D, were able to provide steady pressure fields (no oscillations) confirming the importance of having the presence of the pressure in the mass conservation equation.

Moreover, it should be noted that in all algorithms and in all cases run, the pressure at the northeast volume was set to zero, avoiding a Neumann problem. This explain the negative pressures in Figs. 6 and 7.
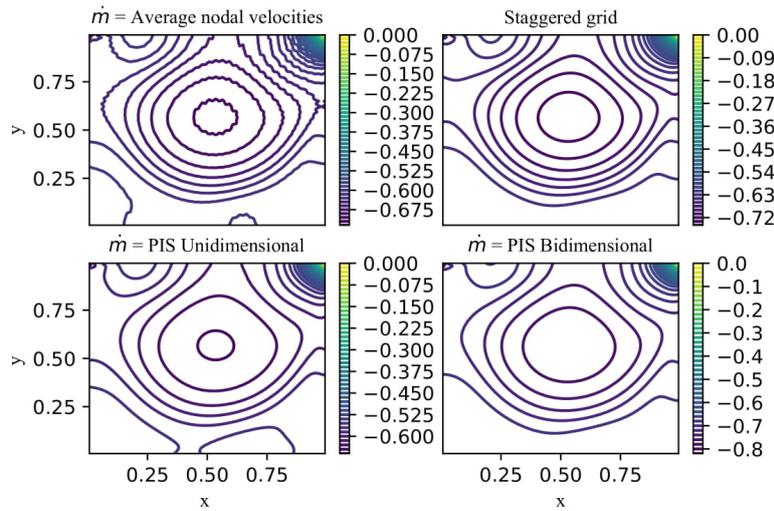
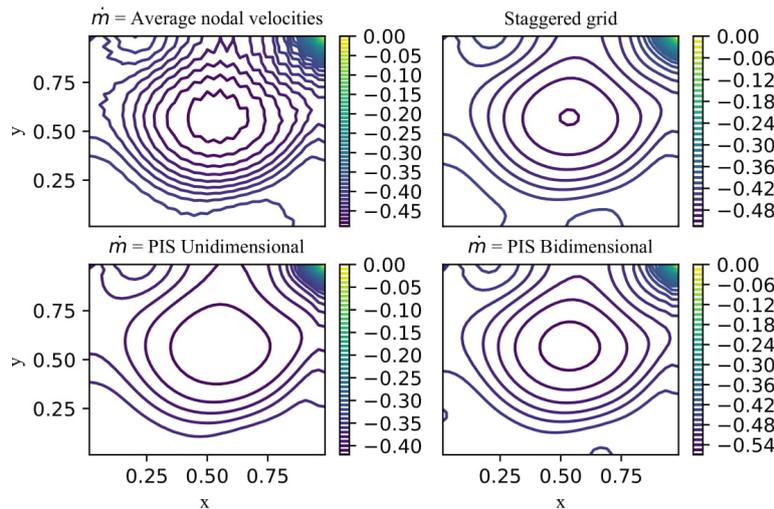Figure 6. Pressure fields for Re = 1000 in cartesian evenly spaced grid with 4,900 volumes.

Figure 7. Pressure fields for Re = 1000 in cartesian evenly spaced grid with 1,369 volumes.

Additionally, with the aim of analyzing the performance of stabilization schemes on coarser grids, Figure 7 illustrates how those cases behave in high Reynolds number case. It is noticed that both schemes accomplish their role of providing stable pressure fields along with contributing to the fulfillment of continuity submatrix as outlined in Eq. 14.

Finally, as can be seen in Figure 8, oscillatory pressure fields are noticeable when the mass flow is calculated by averaging the nodal neighboring velocities (black line), which is not the case when stabilization schemes are employed. Furthermore, the results that made use of the two-dimensional PIS scheme and the staggered arrangement showed the

best results for velocity profiles when compared to (Ghia et al., 1982), carrying the same pressure gradients that inflict the velocity calculation.

Since it is an incompressible problem, pressure does not have a single level, therefore it is important to note pressure gradients from both purple and blue lines. The staggered grid (blue line) is known to be the most stable arrangement for pressure, therefore the similarity between both pressure gradients highlights the efficiency of stabilization schemes in providing stable results for pressure.
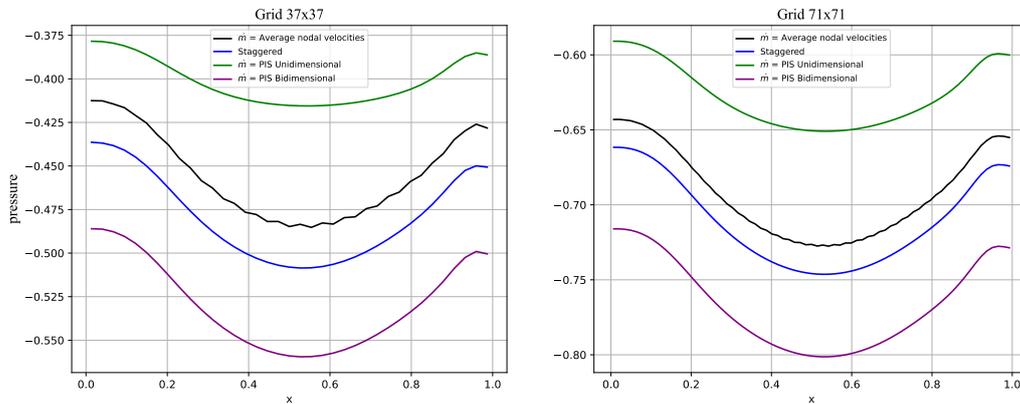
Figure 8. Pressure fields along x for Re = 1000 in cartesian coarser and refined grid.

5. CONCLUSIONS

This paper discussed and compared two approaches for treating the advecting velocities in the solution of a fluid flow problem. It is common for this particular problem to solve the pressure-velocity coupling in a segregated manner, or in a simultaneous way employing a stabilization scheme to prevent oscillatory pressure fields. Therefore, the main goal of this work was to analyze the two approaches and their influence on the pressure fields.

A comparison between the two proposed approaches revealed that both provided excellent results regarding the velocity fields for all three Reynolds numbers. However, the use of average nodal velocities for calculating the convective velocity was unable to fully prevent oscillatory pressure fields. The PIS algorithm has proven to be an efficient scheme in order to obtain stable pressure results for the cartesian evenly spaced grid. As this research is in progress, results for the unstructured grids is underway and will be published in a coming paper. Since in an unstructured grid one has several pressure and velocity nodes around a control volume, it is expected that using the nodal velocity values for calculating the convected velocity may reduce even more the instabilities in the pressure field.

6. REFERENCES

Alisadeghi, H. and Karimian, S., 2011a. "Comparison of different solution algorithms for collocated method of mcim to calculate steady and unsteady incompressible flows on unstructured grids". *Computers & fluids*, Vol. 46, No. 1, pp. 94–100.

Alisadeghi, H. and Karimian, S., 2011b. "Different modelings of cell-face velocities and their effects on the pressure–velocity coupling, accuracy and convergence of solution". *International Journal for numerical methods in fluids*, Vol. 65, No. 8, pp. 969–988.

Dal Pizzol, A. and Maliska, C.R., 2012. "A finite volume method for the solution of fluid flows coupled with the mechanical behavior of compacting porous media". In *AIP Conference Proceedings 4*. American Institute of Physics, Vol. 1453, pp. 205–210.

Ghia, U., Ghia, K.N. and Shin, C., 1982. "High-re solutions for incompressible flow using the navier-stokes equations and a multigrid method". *Journal of computational physics*, Vol. 48, No. 3, pp. 387–411.

Harlow, F.H. and Welch, J.E., 1965. "Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface". *The physics of fluids*, Vol. 8, No. 12, pp. 2182–2189.

Honório, H.T., 2013. *Análise de métodos segregados e acoplado de solução de escoamentos incompressíveis utilizando malhas não-estruturados híbridas*. Master's thesis, Universidade Federal de Santa Catarina, Florianópolis, Santa Catarina.

Hunter, J.D., 2007. "Matplotlib: A 2d graphics environment". *IEEE Annals of the History of Computing*, Vol. 9, No. 03, pp. 90–95.

Maliska, C.R. and Honório, H.T., 2019. "Pressure-displacement coupling in poroelasticity. further details of a stable

finite volume formulation". In *COUPLED VIII: proceedings of the VIII International Conference on Computational Methods for Coupled Problems in Science and Engineering*. CIMNE, pp. 574–585.

Maliska, C.R., Honório, H.T. and Coelho Jr, J., 2017. "A non-oscillatory staggered grid algorithm for the pressure-displacement coupling in geomechanics". In *IACM 19th international conference in flow problems–FEF*.

Patankar, S.V., 1980. *Numerical heat transfer and fluid flow*. Taylor & Francis.

Raybaut, P., 2009. "Spyder-documentation". *Available online at: https://docs.spyder-ide.org/current/index.html*.

Schneider, G. and Raw, M., 1987. "Control volume finite-element method for heat transfer and fluid flow using colocated variables—1. computational procedure". *Numerical Heat Transfer, Part A Applications*, Vol. 11, No. 4, pp. 363–390.

Souza, J.A., 2000. *Implementação de um método de volumes finitos com sistema de coordenadas locais para a solução acoplada das equações de Navier-Stokes*. Master's thesis, Universidade Federal de Santa Catarina, Florianópolis.