

COBEM-2017-0938

GRID CONVERGENCE ANALYSIS USING FINITE VOLUME METHOD FOR NONLINEAR CASE OF FLUID FLOW BETWEEN TWO PARALLEL FLAT PLATES

Wilcker Neuwald Schinestzki
Leonardo Barros da Luz
José Carlos Ignácio Gonçalves Zart
Giuliano Demarco

Universidade Federal de Santa Maria, Brazil, 97105-900, 1000 Roraima Avenue
wilcker.formula@gmail.com
leobdl.formula@gmail.com
zart.formula@gmail.com
giuliano.demarco@ufsm.br

Abstract. *In nowadays engineering, Computational Fluid Dynamics (CFD) has been widely used for improving and developing new technologies. When it comes to complex heat transfer and fluid flow problems, it is necessary to use CFD for solving nonlinear differential equations which cannot be solved analytically. Although there are many numerical methods capable of solving this class of equations, Finite Volume Method (FVM) has been commonly used in the most well-established CFD codes. This method consists on solving discretized conservation equations in central points of each element of a grid, whose number of elements is determined by a base size. In order to acquire reliable solutions, it is important to insure the grid is converged, which means the numerical solution satisfactorily converges to the real answer of the problem. In the present work, the FVM is applied to a nonlinear fluid flow case between two parallel flat plates with the purpose of analyzing grid convergence. Therefore, maximum relative error between different grids was monitored until the defined stopping criteria has been satisfied. The developed code has shown good stability and convergence and it has been validated comparing numerical to analytical Navier-Stokes solution when considered a particular linear case. In this way, this work presents a useful technique for preliminary analysis of problems which can be approached as a nonlinear flow between two parallel flat plates, for instance, in wind tunnels' test section or even in rotors lubrication analysis.*

Keywords: *CFD, grid, convergence, nonlinear, flow.*

1. INTRODUCTION

In fluid mechanics, the concept of control volume is used to derive some conservative equations. In order to determine the velocity field, it is used momentum conservation equations, also known as Navier-Stokes equations. In this case, for a one-dimensional fluid flow, it is only necessary to consider one equation, which corresponds to u component of velocity (Kundu and Cohen, 2011). The general form of this equation for x direction is shown Eq. (1).

$$\rho \left(\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = \rho g_x - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left[-\frac{2}{3} \mu (\vec{\nabla} \cdot \vec{V}) + 2\mu \frac{\partial u}{\partial x} \right] + \frac{\partial}{\partial y} \left[\mu \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right] + \frac{\partial}{\partial z} \left[\mu \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) \right] + S_{Mx} \quad (1)$$

During transient state, boundary layer is yet in development due to viscous effects. Once fluid flow is well-developed, it can be assumed as a steady state, hence $u = u(y)$ (Kundu and Cohen, 2011). In addition, velocity equals zero on the boundaries due to no-slip condition, so $u(0.3) = u(-0.3) = 0$.

The chosen hypothesis for solving the problem were Newtonian ($\mu = \text{constant}$) and incompressible ($\vec{\nabla} \cdot \vec{V} = 0$) fluid with high viscosity ($Re_x \ll 2000$), what makes it a laminar flow (Kundu and Cohen, 2011). An illustration of the problem is shown in Fig. 1.

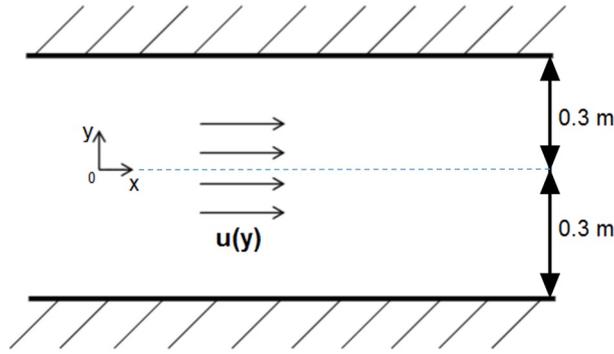


Figure 1: Fluid flow between two parallel flat plates

Simplifying Eq. (1) with all taken hypothesis, it becomes:

$$-\frac{\partial p}{\partial x} + \mu \frac{\partial^2 u(y)}{\partial y^2} - \gamma u^2(y) = 0 \quad (2)$$

Where $\gamma u^2(y)$ is the nonlinear term referent to force fields, which could represent, for example, a magnetic force per unit of volume acting in a ferromagnetic fluid flow (Ray *et al.*, 2012).

2. COMPUTATIONAL PROCEDURE

The first step is to define the number of grid elements N to calculate Δy , which represents the height of each control volume. Grid elements were represented in Fig. 2.

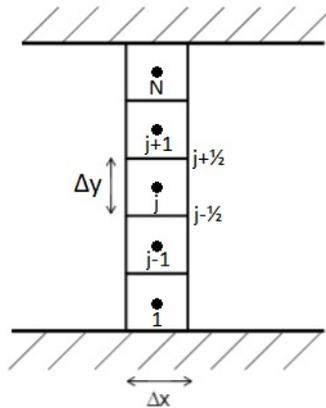


Figure 2: Grid representation

Then, it is possible to integrate Eq. (2) in control volumes for each element j .

$$\int_{CV_j} -\frac{\partial p}{\partial x} + \mu \frac{\partial^2 u(y)}{\partial y^2} - \gamma u^2(y) dV_j = 0 \quad (3)$$

The term $\frac{\partial p}{\partial x}$ is constant due to the parallelism of the flat plates and $u(y)$ can be considered constant for each element. For the variable term, it is necessary to use divergence theorem to analyse the net fluxes through each surface of control (Versteeg and Malalasekera, 2007), which are illustrated in Fig. 3.

So, Equation (3) becomes:

$$-\frac{\partial p}{\partial x} \Delta x \Delta y + \int_{CS_2} \nabla u \cdot \hat{n} dS_2 + \int_{CS_4} \nabla u \cdot \hat{n} dS_4 + \gamma u^2 \Delta x \Delta y = 0 \quad (4)$$

It is possible to notice that net fluxes in surfaces 1 and 3 are zero, since flow is assumed to be well-developed and velocity u is only function of height y . Developing the integrals in the respective surfaces, it becomes:

$$-\frac{\partial p}{\partial x} \Delta x \Delta y - \left(\frac{du}{dy} \right)_{j-\frac{1}{2}} \Delta x + \left(\frac{du}{dy} \right)_{j+\frac{1}{2}} \Delta x + \gamma u^2 \Delta x \Delta y = 0 \quad (5)$$

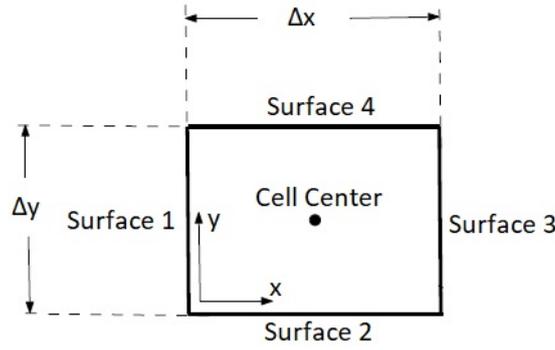


Figure 3: Surfaces of control

Using first-order finite differences to rewrite the differentiations and the velocity component, Equation (5) is now discrete.

$$-\frac{\partial p}{\partial x} + \mu \frac{u_{j-1} - 2u_j + u_{j+1}}{\Delta y^2} - \gamma(2u_{gj}u_j - u_{gj}^2) = 0 \quad (6)$$

The term u_{gj} represents the initial guess for velocity to start the iterative process. The numerical method chosen for solving the linear system was Gauss-Seidel for the reason of faster convergence (Ruggiero and Lopes, 1996). The iterative method was implemented using MATLAB and the parameters were chosen in a way results visualization has become clearer: $\frac{\partial p}{\partial x} = -1$, $\mu = 1$, $\gamma = 20$ and N was started in 5 elements and incremented by 5 units until maximum relative error between $u_j^{(N)}$ and $u_j^{(N+5)}$ has become less or equal to 1%. Maximum relative error E was calculated as shown in Eq. (7).

$$E = \frac{\max(|u_j^{(N)} - u_j^{(N+5)}|)}{|u_j^{(N+5)}|} \leq 1\% \quad (7)$$

An example of the linear system in matrix form for $N = 5$ is shown in Eq. (8).

$$\begin{pmatrix} \beta - 1 & 1 & 0 & 0 & 0 \\ 1 & \beta & 1 & 0 & 0 \\ 0 & 1 & \beta & 1 & 0 \\ 0 & 0 & 1 & \beta & 1 \\ 0 & 0 & 0 & 1 & \beta - 1 \end{pmatrix} \begin{pmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{pmatrix} = -(\Delta y)^2 \begin{pmatrix} \psi \\ \psi \\ \psi \\ \psi \\ \psi \end{pmatrix} \quad (8)$$

Where $\beta = -2 - 2\gamma u_{gj} \Delta y^2$ and $\psi = \gamma u_{gj}^2 + 1$.

According to Strang (2007), in the discretization of second order differential equations, periodic, symmetric, sparse and also tridiagonal matrices are obtained, so as shown in Eq. (8). Due to boundary conditions applied to the walls, the first and last terms of principal diagonal are different from the rest ones.

As stopping criteria for the loop, it was used a relative residual boundary of 10^{-3} , which was defined as shown in Eq. (9).

$$R = \frac{\sum_{k=1}^N |u_j^{(k)} - u_j^{(k-1)}|}{\sum_{k=1}^N |u_j^{(k-1)}|} \leq 10^{-3} \quad (9)$$

Where k is the current iteration, hence $u_j^{(0)} = u_{gj}$.

It was used 10^{-3} to limit relative residual because when γ was chosen equal to zero (linear case), it has shown a good convergence-time ratio when compared to analytical Navier-Stokes solution.

3. CODE VALIDATION

In order to validate the used code, γ was set to zero and automatically the differential equation has become linear as shown in Eq. (10).

$$-\frac{\partial p}{\partial x} + \mu \frac{\partial^2 u(y)}{\partial y^2} = 0 \tag{10}$$

With no presence of the nonlinear term, Equation (10) can be analytically solved to the chosen boundary conditions, and its solution is shown in Eq. (11).

$$u(y) = -\frac{y^2}{2} + 0.045 \tag{11}$$

Then, numerical solution for linear case was overlapped and compared to the analytical solution in Fig. 4.

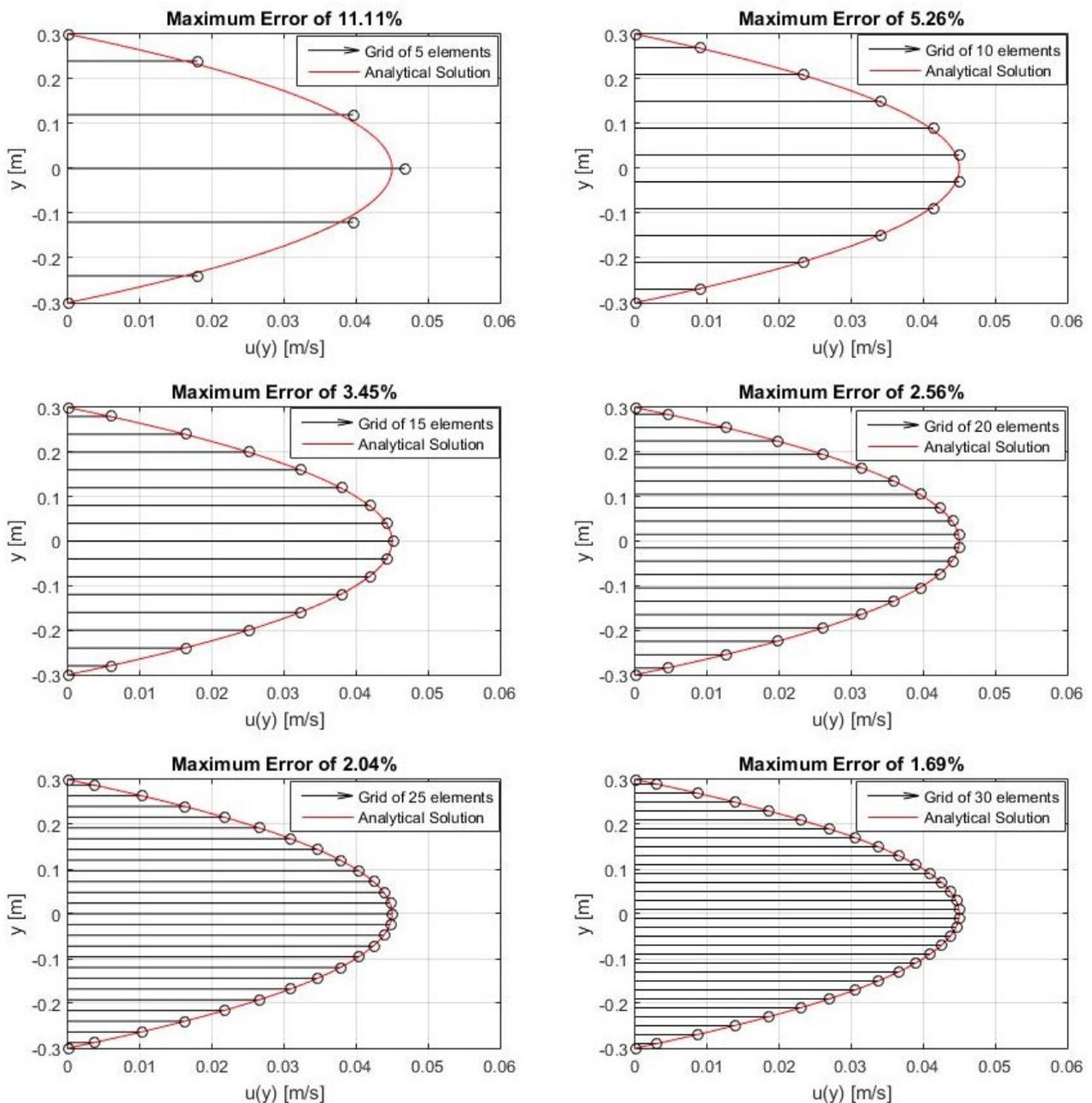


Figure 4: Comparison between numerical and analytical solutions

Analyzing the comparison done in Fig. 4, it is clear that the higher the number of grid elements, the lower the maximum relative error. This happens due to the necessity of many grid elements for numerical solution to fit better analytical solution curve.

Once known that numerical solution converges to the correct solution with the chosen relative residual limit, the next step was to verify grid convergence between different grids for linear case. As already explained in computational procedure section, number of elements was increased by 5 units until 1% or less of maximum relative error was achieved, as shown in Fig. 5. It was noticed that 35 elements were necessary for achieving the desired grid convergence.

Also, relative residual has been plot in function of iteration number in order to verify the solution stability along iterations, as can be seen in Fig. 6. The required number of iterations for achieving convergence in the last simulation was about $2.9 \cdot 10^4$. In addition, a good stability could be observed for the applied method, since the plotted curve is smooth, indicating no solution oscillation.

After ensured convergence and stability, the nonlinear grid convergence case was done.

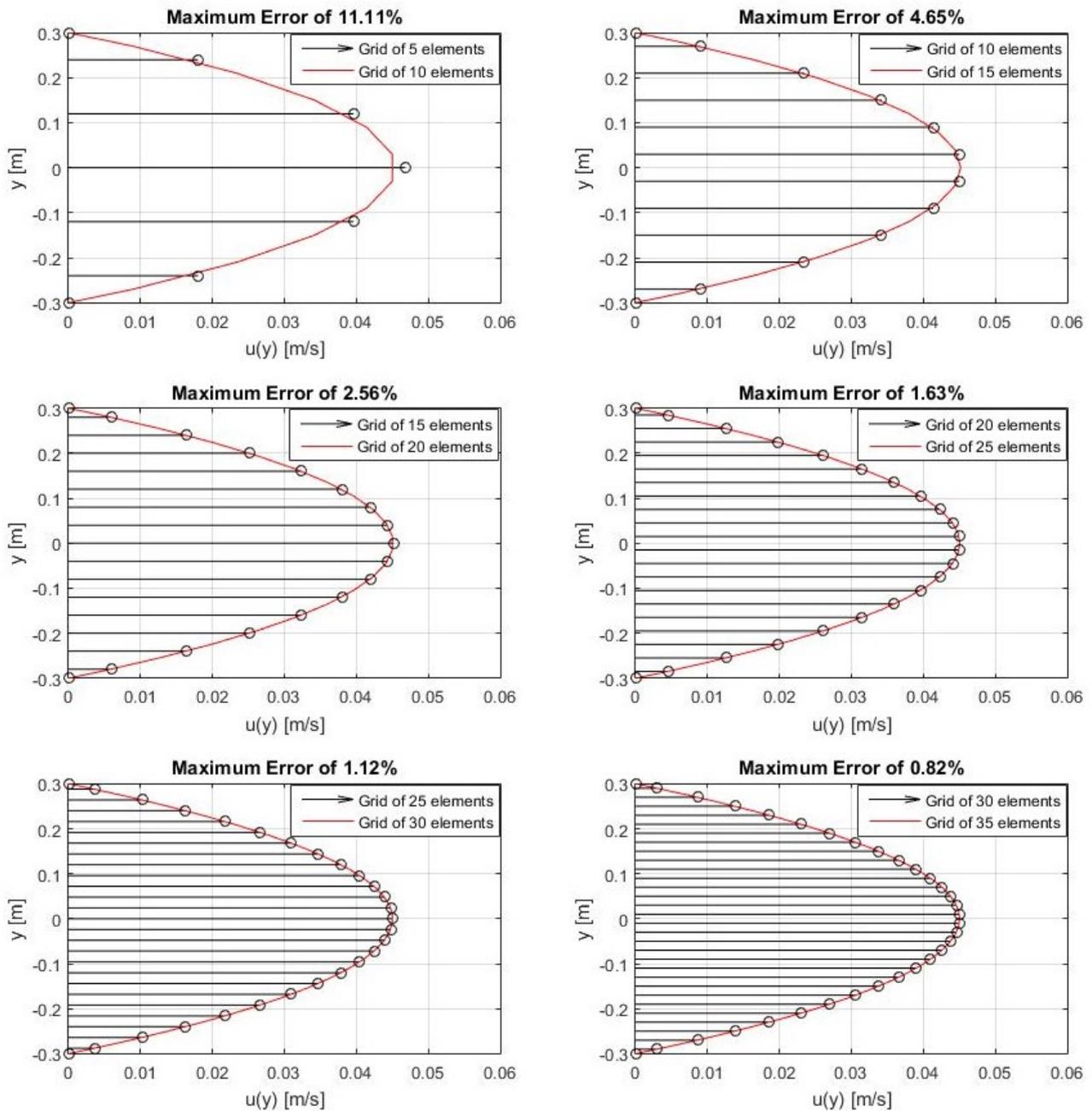


Figure 5: Grid independence for linear case

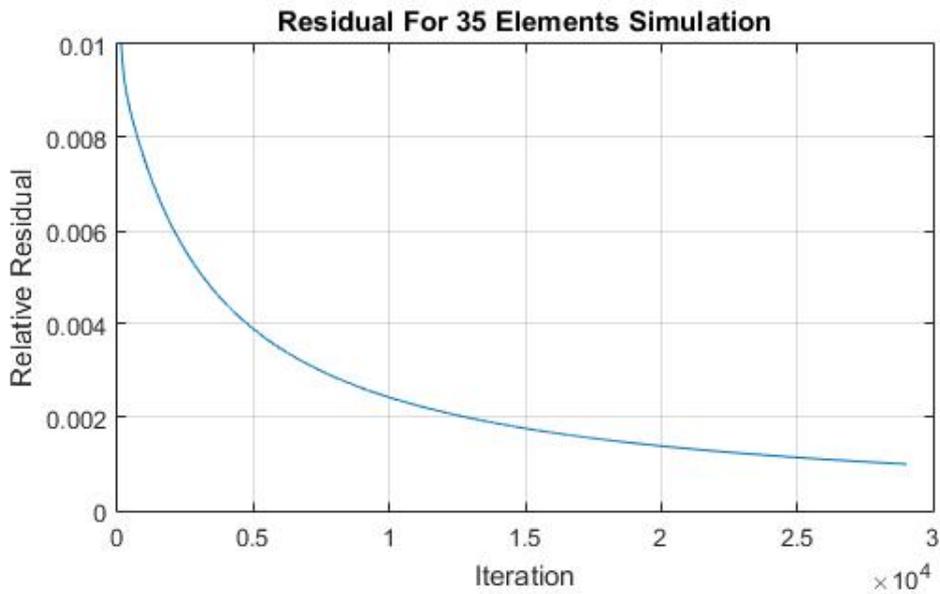


Figure 6: Relative residual for linear case

4. RESULTS AND DISCUSSION

Once the method was applied to the nonlinear case, grid independence analysis has required about $3.1 \cdot 10^4$ iterations to converge last simulation, which is higher than the number of iterations required to converge linear case. Nevertheless, relative residual still has decreased smoothly, converging stably to a numeric steady state solution, as shown in Fig. 7.

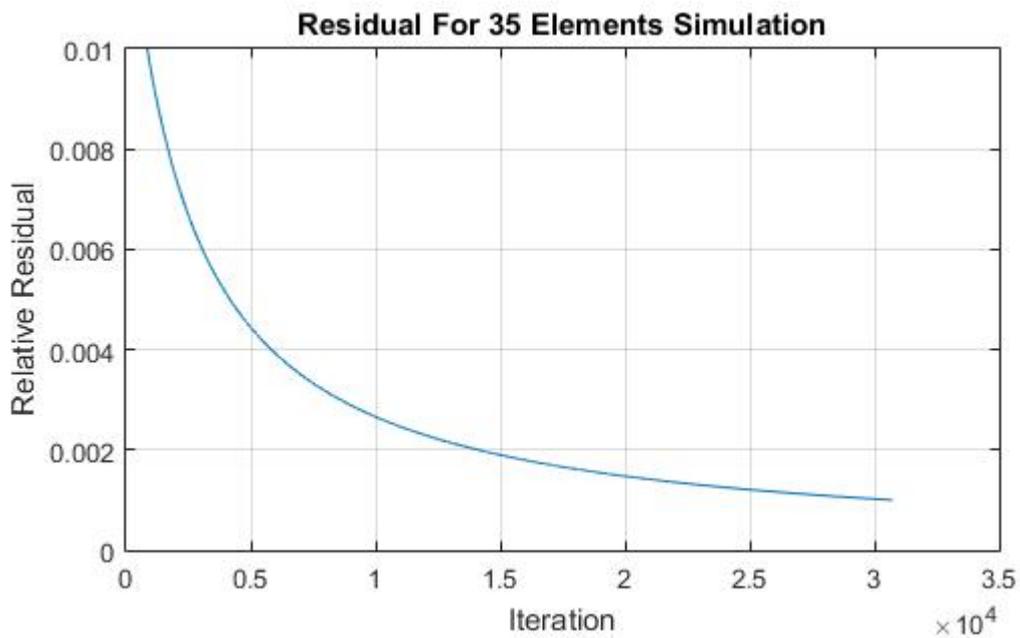


Figure 7: Relative residual for nonlinear case

It is important to notice that both linear and nonlinear case have achieved grid independence for maximum relative error less or equal to 1% with 35 grid elements. When analyzing Fig. 8, which represents grid independence for nonlinear case, it is noted that for the same number of elements, maximum relative error is a little bit higher than in nonlinear case, since there are extra terms to be taken into account while solving the linear system.

Remembering maximum relative error has been calculated and normalized in horizontal direction, it is hard to perceive significant error visually when $N \geq 15$.

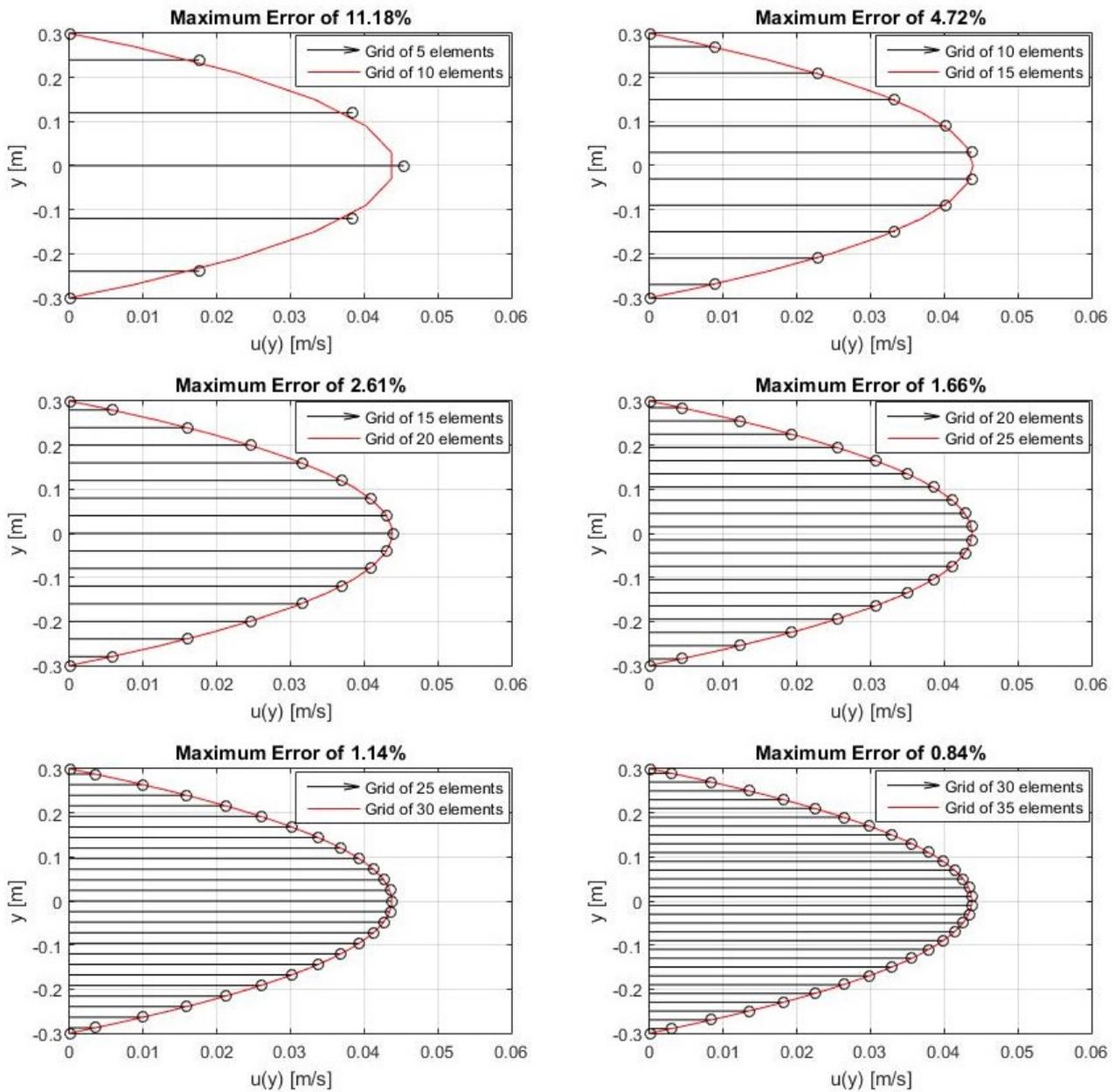


Figure 8: Grid independence for nonlinear case

When it comes to solution dependency on the nonlinear term, it is expected velocity vectors to be stretched or to be shortened depending on sign and absolute value of γ , since it is the proportionality constant between force field per volume unit and square velocity. In this case, comparing Fig. 5 and Fig. 8, it can be seen that velocity vectors have been shortened.

5. CONCLUSION

It was possible to conclude that a coarse grid is not able to satisfactorily approximate analytical solution. Also, an excessively refined grid is not necessary, since it increases considerably computational cost. So, in order to efficiently apply CFD in engineering, it is essential to analyze whether grid is converged or not and if it is not excessively refined, so that a good cost-benefit ratio can be achieved.

Another thing to highlight is that the developed code has shown good stability and convergence. It is robust enough for comparing relative error between different velocity fields independent on their number of grid elements and for allowing different settings for nonlinear force fields and boundary conditions for the parallel flat plates. In this way, it has many applications in preliminary analysis which can be approached as a flow between parallel flat plates, for instance, in wind tunnels test section or even rotors lubrication analysis.

Also, during the development of this work, it was noticed that a fixed step for increasing grid elements is not the best option, since this method can overshoot stopping criteria, refining the grid more than necessary. In this way, it is suggested for future works the code to be optimized with a variable step, in order to achieved the best solution for the chosen relative error limit.

6. REFERENCES

- Kundu, P.K. and Cohen, I.M., 2011. *Fluid Mechanics*. Academic Press, Massachusetts, 5th edition.
- Ray, B., Bhaskaran, R. and Collins, L., 2012. *Introduction to CFD*. Cornell University, Ithaca, NY.
- Ruggiero, M.A.G. and Lopes, V.L.R., 1996. *Cálculo Numérico: aspectos teóricos e computacionais*. Pearson Makron Books, São Paulo, 2nd edition.
- Strang, G., 2007. *Computational Science and Engineering*. Wellesley-Cambridge Press, Wellesley MA, 1st edition.
- Versteeg, H.K. and Malalasekera, W., 2007. *An Introduction to Computational Fluid Dynamics*. Pearson Education Limited, United Kingdom, 2nd edition.

7. RESPONSIBILITY NOTICE

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