

Comparing the Solution of the Navier-Stokes Equations Using a Fixed Point Iterative Method and Using COMSOL Multiphysics

Blanca, BERMÚDEZ JUÁREZ

Facultad de Ciencias de la Computación, Benemérita Universidad Autónoma de Puebla
14 Sur y San Claudio, Ciudad Universitaria, Puebla, Puebla, México

Beatriz, BONILLA CAPILLA

Facultad de Ciencias Físico Matemáticas, Benemérita Universidad Autónoma de Puebla
18 Sur y San Claudio, Ciudad Universitaria, Puebla, Puebla, México

José David, ALANIS URQUIETA

División Tecnologías de la Información, Universidad Tecnológica de Puebla
Antiguo Camino a la Resurrección 1002-A Col. Zona Industrial Oriente
Puebla, Puebla, México

Alejandro, RANGEL HUERTA

Facultad de Ciencias de la Computación, Benemérita Universidad Autónoma de Puebla
14 Sur y San Claudio, Ciudad Universitaria, Puebla, Puebla, México

Wuiyebaldo Fermín, GUERRERO SANCHEZ

Facultad de Ciencias Físico Matemáticas, Benemérita Universidad Autónoma de Puebla
18 Sur y San Claudio, Ciudad Universitaria, Puebla, Puebla, México

ABSTRACT

In this paper, we compare the solutions for the Navier-Stokes equations with moderate and very high Reynolds numbers obtained using a Fixed Point Iterative Method with those obtained using COMSOL Multiphysics. Despite the advantages of COMSOL, we want to show that our results, using a Fixed Point Iterative method agree as much as possible, with those obtained with COMSOL. Results for viscous incompressible flows in 2D are presented, using the Stream Function-vorticity formulation of the Navier-Stokes equations. The Fixed point Iterative Method uses Finite Differences and a uniform mesh; COMSOL uses the Finite Element Method and the formulation in primitive variables and the mesh is refined in some places; streamline and crosswind diffusion are also used. Results are reported, in the case of the lid-driven cavity problem for Reynolds numbers in the range of $5000 \leq Re \leq 100000$.

As the Reynolds number increases, the time and the step mesh have to be refined, both for time and space in order to capture the fast dynamics of the flow and numerically, because of stability reasons. The advantages of our code are: it is “transparent” and easily modifiable, so, it can be used for solving other problems. We are looking forward to parallelize it.

Keywords: Stream function- vorticity formulation, primitive variables formulation, Reynolds number, Fixed Point Iterative Process, COMSOL Multiphysics, parallelization.

1. INTRODUCTION

The fixed-point iterative method described in this work has already been used for solving the Navier-Stokes equations and the Boussinesq system under different formulations [2], [3], [11], [4], [5], [6]. With this iterative method, the idea was to work with a symmetric and positive definite matrix.

The scheme worked very well, as shown in [2], [3], [4], [5] [6], [11] but the processing time was, in general, very large, especially for high Reynolds numbers.

To show our scheme works well for moderate and high Reynolds numbers, we report results for the lid-driven cavity problem and Reynolds numbers in the range of $5000 \leq Re \leq 100000$.

As the Reynolds number increases, the mesh has to be refined and a smaller time step has to be used, to capture the fast dynamics of the flow and, numerically, because of stability reasons, as mentioned in [11].

In [12], they give some transient solutions at $Re=100000$ in the lid-driven cavity problem and give a numerical confirmation of the presence of an attractor starting with two very different initial data. They are using a Finite Difference Method and the primitive variables formulation. They used very fine grids (2048 x 2048) for arriving to $Re=100000$. As they said, there are very few computations for such a high Reynolds number in the literature, since it is difficult to represent correctly the flow in the boundary layer. In [13], some results give a view of the solution in the transient starting from rest. In [16], they explore the possibility of implicit Large Eddy Simulation (LES) using the laminar interface of COMSOL Multiphysics. They examine the lid-driven cavity flow Reynolds numbers up to 1000000.

In [14], various turbulence models were used to simulate internal turbulent flow with lid-driven cubic box. They arrive to $Re=100000$ using a uniform mesh of 2500 x 2500.

In [15], computer simulation results of steady incompressible flows in a 2-D square lid-driven cavity up to $Re=65000$ are presented. Quadratic Upstream Interpolation for convective Kinematics (QUICK) is used for the approximation of the convective term.

The idea in this work is mainly to compare our results, when using a Fixed Point Iterative Method codified in FORTRAN by ourselves, with those obtained using a very robust commercial software, COMSOL Multiphysics.

2. INTRODUCTION

Let $\Omega \subset R^N$ ($N = 2, 3$) be the region of a viscous, incompressible, non-stationary flow and Γ its boundary.

$$\begin{cases} u_t - \nabla^2 u + \nabla p + (u \cdot \nabla) = f \\ \nabla \cdot u = 0 \end{cases} \quad \begin{matrix} (a) \\ (b) \end{matrix} \quad (1)$$

These are the Navier-Stokes equations in primitive variables, and this system must be provided with appropriate initial and boundary conditions.

The initial conditions are given by:

$$u(x, 0) = u_0(x) \text{ in } \Omega, \quad (2)$$

and the boundary conditions by:

$$u = g \text{ on } \Gamma \quad (3).$$

Restricting equations (1a-b) to a two dimensional region Ω , taking the curl in both sides of the equation (1a) and taking into account that:

$$u_1 = \frac{\partial \psi}{\partial y}, \quad u_2 = -\frac{\partial \psi}{\partial x}, \quad (4)$$

followed by (1b), with ψ the Stream function and u_1, u_2 , the two components of the velocity, we arrive to the Stream function-vorticity formulation of the Navier-Stokes equations.

The following system of equations is then obtained:

$$\begin{cases} \nabla^2 \psi = -\omega, & (a) \\ \omega_t - \nabla^2 \omega + u \cdot \nabla \omega = f_\omega, & (b) \end{cases} \quad (5)$$

where ω is the vorticity and $\omega = \frac{\partial u_2}{\partial x} - \frac{\partial u_1}{\partial y}$.

This system represents the Navier-Stokes equations in the Stream function-vorticity formulation. The incompressibility condition (1b), by (3) is automatically satisfied and the pressure does not appear any more.

This system represents the Navier-Stokes equations in the Stream function-vorticity formulation. The incompressibility condition (1b), by (3) is automatically satisfied and the pressure does not appear any more.

3. Fixed Point Iterative Method

For the time derivative, the following second order approximation is used:

$$\omega_t(x, (n+1)\Delta t) \approx \frac{3\omega^{n+1} - 4\omega^n + \omega^{n-1}}{2\Delta t},$$

where $n \geq 1$, $x \in \Omega$ y $\Delta t > 0$ is the time step size.

Then, at each time level, the following non-linear system has to be solved:

$$\begin{cases} \nabla^2 \psi = -\omega, \quad \psi|_{\Gamma} = \psi_{bc}, \\ \omega_t - \nabla^2 \omega + u \cdot \nabla \omega = f_\omega, \quad \omega|_{\Gamma} = \omega_{bc}, \end{cases} \quad (a) \quad (b)$$

where $\alpha = \frac{3}{2\Delta t}$ and $f_\omega = \frac{4\omega^n - \omega^{n-1}}{2\Delta t}$

For solving this system of equations, two strategies were used in this work: first, we used a fixed-point iterative method described in [1]. Denoting:

$$R_\omega(\omega, \psi) = \alpha\omega - \frac{1}{Re} \nabla^2 + u \cdot \nabla \omega - f_\omega, \text{ in } \Omega,$$

our system is equivalent to:

$$\begin{cases} \nabla^2 \psi = -\omega, \quad \psi|_{\Gamma} = \psi_{bc} \\ R(\omega, \psi) = 0, \quad \omega|_{\Gamma} = \omega_{bc} \end{cases} \quad (a) \quad (b)$$

Then, at each time level, the following iterative process [1] is used:

Given ω^0 and ψ^0 solve until convergence in ω and ψ

$$\begin{cases} \nabla^2 \psi^{m+1} = -\omega^m & \text{in } \Omega \\ \psi^{m+1} = \psi_{bc} & \text{over } \Gamma \\ \psi^{m+1} = \omega^m - \rho_\omega (\alpha I - \epsilon \nabla^2)^{-1} R_\omega(\omega^m, \psi^{m+1}) & \text{in } \Omega \\ \psi^{m+1} = \omega_{bc}^{m+1} & \text{in } \Gamma, \rho_\omega > 0 \end{cases}$$

and then take $(\omega^{n+1}, \psi^{n+1}) = (\omega^{m+1}, \psi^{m+1})$.

4. Numerical Stability in COMSOL Multiphysics.

Equation (1a) is unstable when using the Galerkin Finite Element Method. Stabilized Finite Element Methods are then necessary in order to obtain physical solutions. There are three types of stabilization methods available in COMSOL for the Navier-Stokes equations: streamline diffusion, crosswind diffusion and isotropic diffusion (see Reference Guide COMSOL Multiphysics 4.3a [17]). Below we discuss streamline and crosswind diffusion.

4.1 Streamline Diffusion

It is necessary when convection is the dominating part of the flow and it is supported by COMSOL Multiphysics. It is active by default when convection is dominating. The unstabilized incompressible Navier-Stokes equations are subject to the Babuska-Brezzi conditions, the basic functions for pressure must be of lower order than those for velocity. If they are stabilized by Galerkin Least Squares (GLS), it is possible to use equal-order interpolation.

4.2 Crosswind Diffusion

Crosswind diffusion, when applied to the Navier-Stokes equations becomes a shock-capturing operator. Incompressible flows do not contain shock waves, but crosswind diffusion is still useful for introducing extra diffusion in sharp boundary layers and shear layers that otherwise would require a very dense mesh to resolve.

The tuning parameter, C_k controls the amount of crosswind diffusion introduced in the model. The recommended range for low Mach number flows and incompressible flows is $0 < C_k < 1.0$ ($C_k = 0$ means no diffusion at all). The value must not depend on space or time. We use the default value C_k ($C_k = 1$).

5. Numerical Experiments

With respect to the lid-driven cavity problem, and using the Stream function-vorticity formulation, $\Omega = [0,1] \times [0,1]$ the top wall is moving with a velocity given by $(0, 1)$ and $(0, 0)$ for the other three walls, the velocity is given by $(0, 0)$, ψ is over determined at the boundary ($\frac{\partial \psi}{\partial n}$ at the boundary is also known) and there is no boundary condition for ω . In our case, we have followed the alternative proposed by Goyon [9], $\psi = 0$ is chosen over Γ . A translation of the boundary condition in terms of the velocity (primitive variable) has to be used. By a Taylor series expansion of equation (3), we obtained:

$$\begin{aligned}\omega(0, y, t) &= -\frac{1}{2h_x^2}[8\psi(h_x, y, t) - \psi(2h_x, y, t)], \\ \omega(a, y, t) &= -\frac{1}{2h_x^2}[8\psi(a - h_x, y, t) - \psi(a - 2h_x, y, t)], \\ \omega(x, 0, t) &= -\frac{1}{2h_y^2}[8\psi(x, h_y, t) - \psi(x, 2h_y, t)], \\ \omega(x, b, t) &= -\frac{1}{2h_y^2}[8\psi(x, b - h_y, t) - \psi(x, b - 2h_y, t)],\end{aligned}$$

where h_x, h_y denote the spatial step size in the directions of x, y respectively. In Figure 1 we show results for $Re = 7500$. We show the isovorticity contours obtained using the Fixed Point Iterative Method, with $h = h_x = h_y = 1/128$ and the results obtained with COMSOL Multiphysics. The steady state is shown. In the following figures we only shows results for $t_{final} = 5$ due to the computational cost needed to arrive to a greater time. Moreover for the Reynolds Numbers shown there is no steady state. In Figure 2 and Figure 3 we show the isovorticity contours for $Re = 25000$ with $h = h_x = h_y = \frac{1}{728}$ for $T_{final} = 5$, with both methods. In Figure 4 we show the isovorticity contours for $Re = 50000$ with $h = h_x = h_y = \frac{1}{1024}$ for $T_{final} = 5$, using also both methods. In Figure 5 we show the isovorticity contours for $Re = 100000$ with $h = h_x = h_y C = \frac{1}{1280}$ for $T_{final} = 5$, using both methods. Results with COMSOL Multiphysics are shown in the Figures 2 to 5 and were calculated without stabilization and using stabilization (streamline and crosswind diffusion). The mesh used in COMSOL Multiphysics was an extremely fine mesh; it has 162225 domain elements and 600 contour elements.

We must say that for the Fixed Point Iterative Method, mesh independence studies have been made, for example in Nicolás et al [10], in order to verify the convergence of the method.

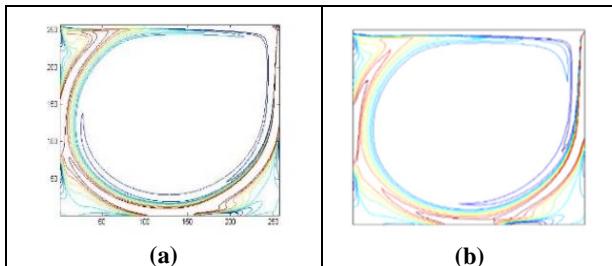


Figure 1: Isovorticity contours for $Re = 7500$, $h_x = h_y = 1/256$ with the F. P. I.M. (Finite Point Iterative Method) (a), and with COMSOL and an extremely fine mesh (b)

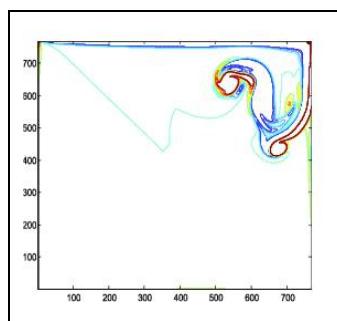


Figure 2: Isovorticity contours for $Re = 25000$, $h_x = h_y = 1/728$ with the F. P. I.M.

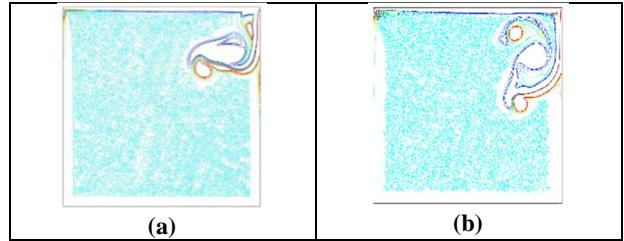


Figure 3: Isovorticity contours for $Re = 25000$, $h_x = h_y = 1/728$ with COMSOL and an extremely fine mesh without stabilization (a), and with streamline diffusion and crosswind diffusion (b).

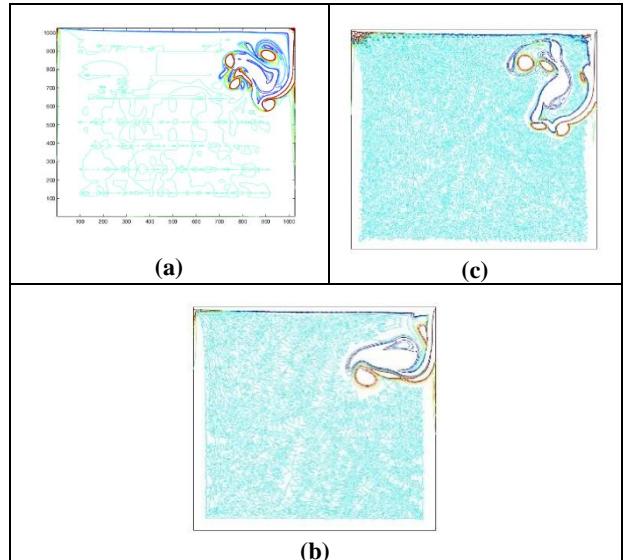


Figure 4: Isovorticity contours for $Re = 50000$, $h_x = h_y = 1/1024$ with the F. P. I.M. (a) and with COMSOL and an extremely fine mesh without stabilization (b) and with streamline diffusion and crosswind diffusion (c).

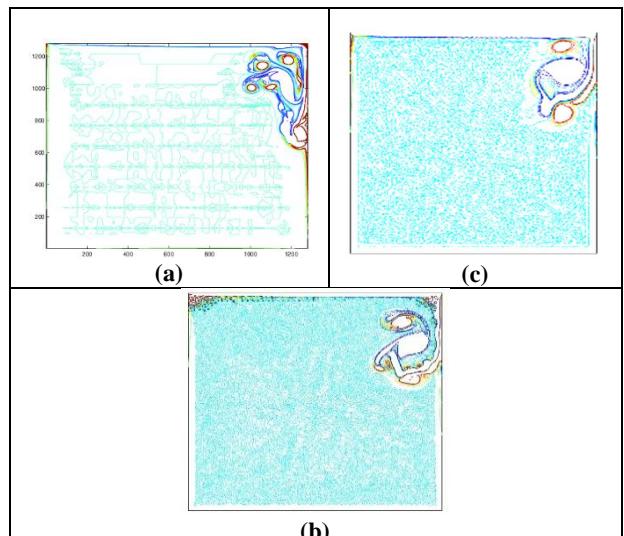


Figure 5: Isovorticity contours for $Re = 100000$, $h_x = h_y = 1/1280$ with the F. P. I.M. (a) and with COMSOL and an extremely fine mesh without stabilization (b) and with streamline diffusion and crosswind diffusion (c).

6. Conclusions

For the lid-driven cavity problem, results agree very well with those reported in the literature [2], [3], [4], [6], [7], [8], and [10]. As can be seen in Figure 1, oscillations occurred, given the high Reynolds numbers used, in such a way that it is necessary to use smaller values of h [11], numerically because of stability of the method, and physically, in order to capture the fast dynamics of the flow.

For high Reynolds numbers and small values of h , the computational work takes even days, in the case of the Fixed Point Iterative method, so reducing computing time becomes very important.

We must recognize that a disadvantage of the Fixed Point Iterative Method is time. Computations for high Reynolds numbers take a lot of time and we have still not been able to parallelize the code, since it is not an easy task. COMSOL Multiphysics uses Primitive Variables Formulation and we are using the Stream Function-vorticity Formulation. In this latter case, the incompressibility condition is automatically satisfied and it does not include the pressure explicitly.

Results with both methods agree very well up to $Re = 10000$. For Reynolds greater than $Re = 10000$, we observe some differences due to the meshes used. In our case we are able to handle such fine meshes (up 1024 x 1024), and COMSOL Multiphysics is not able to handle them. Moreover COMSOL Multiphysics does not use uniform meshes.

Nevertheless, the results with the Fixed Point Iterative Method shows better structure on turbulent vortices. In this case, with COMSOL Multiphysics, you have to include a stabilization method such as streamline and crosswind diffusion.

Even though turbulence is a tri-dimensional phenomenon, two dimensional flows at high Reynolds numbers give some clues of transition to real turbulence. Some examples of turbulent flows are the boundary layers in the atmosphere, the ocean currents, the wake of a reactor, boundary layers around plane wings, the trail of ships, cars, airplanes, submarines, the flow on a river, etc. In COMSOL Multiphysics, you cannot add some code by yourself, and our code is, let us say “transparent” and that is an advantage of our code, which we are looking forward to parallelize.

7 Acknowledgements

The authors would like to acknowledge the support received by the National Laboratory of Supercomputing from of the southeast of Mexico BUAP-INAOEP-UDLAP (Laboratorio Nacional de Supercómputo del Sureste de México) in running our programs.

8 References

- [1] Nicolás A., A finite element approach to the Kuramoto-Sivashinsky equation. *Advances in Numerical Equations and Optimization*, Siam (1991).
- [2] Bermúdez B., Nicolás A., Sánchez F.J., Buendía E., Operator Splitting and unwinding for the Navier-Stokes equations, *Computational Mechanics* (1997) 20 (5): 474-477.
- [3] Nicolás A., Bermúdez B., 2D incompressible viscous flows at moderate and high Reynolds numbers, *CMES* (2004): 6(5): 441-451.
- [4] Nicolás A., Bermúdez B., 2D Thermal/Isothermal incompressible viscous flows, *International Journal for Numerical Methods in Fluids* (2005) 48: 349-366.
- [5] Bermúdez B., Nicolás A., Isothermal/Thermal Incompressible Viscous Fluid Flows with the Velocity Formulation. *Información Tecnológica* (2010) 21(3): 39-49
- [6] Bermúdez B. and Nicolás A. The Taylor Vortex and the Driven Cavity Problems by the Velocity-Vorticity Formulation, *Proceedings 7th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics* 2010.
- [7] Glowinski R., Finite Element methods for the numerical simulation of incompressible viscous flow. Introduction to the control of the Navier-Stokes equations. *Lectures in Applied Mathematics* (1991), (AMS), 28.
- [8] Ghia U., Ghia K.N. and Shin C.T., High-Re Solutions for Incompressible Flow Using the Navier-Stokes equations and Multigrid Method, *Journal of Computational Physics* (1982): 48, 3817-411.
- [9] Goyon, O., High-Reynolds numbers solutions of Navier-Stokes equations using incremental unknowns, *Computer Methods in Applied Mechanics and Engineering* 130 (1996): 319-355.
- [10] Nicolás-Carrizosa A., Bermúdez Juárez B., Onset of two dimensional turbulence with High Reynolds Numbers in the Navier-Stokes equations, *Proceedings IV International Conference on Computational Methods for Coupled Problems*, (2011).
- [11] Nicolás A., Bermúdez B., Viscous incompressible flows by the Velocity-Vorticity Navier-Stokes equations, *CMES* (2007) Vol. 20, No. 2, 73-83.
- [12] Bruneau Charles-Herni, Mazen Saad, The Behaviour of High Reynolds flows in a driven Cavity, *Computational Fluid Dynamics Journal*, (2006) 15(3).
- [13] Kuwahara K., Computational Fluid Dynamics: Cavity flow at Reynolds Number 100000, <http://www2.icfd.col.jp>
- [14] Nagapetyan Hakop J., Wray Thimothy J., Agarwal Ramesh K., Computation turbulent Flow in a lid-driven 2-D Cavity and a 3-D Box using a number of Turbulence Models, 54th. AIAA Aerospace Sciences Meeting, AIAA SciTech Forum, (2016) 1114.
- [15] Yapıcı k., Uludag Y., Finite Simulation of 2-D Steady Square Lid-Driven Cavity Flow at High Reynolds Numbers, *Brazilian Journal of Chemical Engineering*, Vol. 30 No. 4 923-937.
- [16] Hashiguchi M., Possibility of Implicit LES for Two Dimensional Incompressible Lid Driven Cavity Flow Based on COMSOL Multiphysics, Accessed online 11, 2015.
- [17] COMSOL, COMSOL Multiphysics User’s Guide, COMSOL 4.3, USA, 2012.