25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

COB-2019-1826
TOWARDS FLEXIBLE AND OPTIMIZED DIRECT NUMERICAL SIMULATIONS OF INCOMPRESSIBLE TURBULENT FLOWS

Ednir Luís Pedro Nigra Júnior
School of Mechanical Engineering, University of Campinas, 200 Mendeleyev Street, Barão Geraldo District, Campinas, SP, Brazil
enigra@fem.unicamp.br

Jan Mateu Armengol
Laboratoire EM2C, CNRS, CentraleSupélec, Université Paris-Saclay, 3, rue Joliot Curie, 91192 Gif-sur-Yvette cedex, France
jan.mateu@ecp.fr

Rogério Gonçalves dos Santos
School of Mechanical Engineering, University of Campinas, 200 Mendeleyev Street, Barão Geraldo District, Campinas, SP, Brazil
roger7@fem.unicamp.br

Abstract. *This work aims to present an ongoing development of a high-order code capable of determining the unsteady velocity and pressure fields of incompressible turbulent flows. In the current study, a fourth-order centered finite-difference scheme is used to discretize the spatial derivatives of the incompressible Navier-Stokes equations on a collocated grid arrangement with a uniform structured mesh. An explicit fourth-order Runge-Kutta method is employed to advance in time the solution. The pressure-velocity coupling is obtained through a Poisson equation for the pressure. The Gauss-Seidel iterative method is used to solve the implicit Poisson equation. The classical laminar lid-driven cavity flow for the Reynolds number 100, 400, and 1000 is here used to assess and to verify the code. Comparisons with the results available in the literature are given. Future works are planned regarding the parallelization of this solver based on a domain decomposition using the open library PETSc.*

Keywords: *2D lid-driven cavity flow, high-order finite-difference method, incompressible Navier-Stokes equations, Poisson equation for pressure*

1. INTRODUCTION

The study of fluids in motion plays an important role in a wide range of industrial and non-industrial application areas. The understanding of the physics of fluid flow is crucial to develop new products and optimize existing products, e.g., the combustion process in an internal combustion engine and cooling of electronic devices. In the fluid flow, some physical phenomena are encountered, such as diffusion, convection, dissipation, slip surfaces, boundary layers, shock waves, and turbulence (Lomax *et al.*, 2001). For Newtonian fluids, these phenomena may be modeled mathematically by a set of partial differential equations known as Navier-Stokes equations. However, these equations have a general no analytical solution. These difficulties motivate the researchers to adopt the numerical simulations of the Navier-Stokes equations.

The study of fluid flows by numerical simulations is known as Computational Fluid Dynamics (CFD). This technique has become an important tool in several industries due to its advantages over the experiment-based approach. The costs of a CFD capability is usually cheaper than a high-quality experimental facility. Moreover, there is a reduction in project time and the ability to isolate a specific phenomenon to study. CFD is developed by research centers of industries and universities. Usually, these research centers use numerical simulations to develop fundamental researches in fluid flows as well as to investigate the physical phenomena that occur in specific applications. This kind of study requires numerical simulation data with high accuracy. For this end, the physical model chosen must represent the behavior of the fluid, and the algorithms used must introduce no more than a low-level error (Sagaut, 2006). Before using numerical simulations on application problems, the algorithm used needs to be verified and validated. A test case (benchmark) with experimental results or analytical solution is reproduced by the CFD, and their results are compared. This procedure is called validation. Another way is to compare the CFD results with other numerical studies. This procedure is called verification. In this work, the verification process is adopted using the numerical results of some lid-driven cavity flow benchmarks.

Since the work of Burggraf (1966), the lid-driven cavity flow has been used by many authors as the classical test problem for the assessment of numerical methods and the verification of Navier-Stokes codes (Botella and Peyret, 1998; Fortuna, 2000). Not being different, the present work employs this same problem to evaluate and verify an in-house two-dimensional Fortran code in developing that is capable of determining velocity and pressure fields of unsteady, incompressible, viscous, and Newtonian fluid flows.

Burggraf (1966) pioneered the study of the classical lid-driven cavity problem. In his work, the finite-difference method was used to discretize the spatial derivatives of the Navier-Stokes equations formulated in the stream function-vorticity variables. He obtained numerical solutions for flows inside a driven cavity for Reynolds number $0 \leq Re \leq 400$ on a mesh of 40×40 nodes. Many authors have investigated this problem. The previous numerical results were reviewed by AbdelMigid *et al.* (2017), and a physical review of the internal flow in cavities is presented in the work of Shankar and Deshpande (2000).

The present study is a first step towards an efficient solver for Direct Numerical Simulations (DNS) of incompressible turbulent flows. Thus, future works will treat the parallelization of this solver based on a domain decomposition using the open library PETSc (The Portable, Extensible Toolkit for Scientific Computation).

2. PHYSICAL AND MATHEMATICAL MODEL

In this section, the physical and mathematical model of the two-dimensional lid-driven cavity problem studied here are presented.

2.1 Physical model

Figure 1 shows a schematic representation of the two-dimensional lid-driven cavity problem studied in the present work. The dimension of the square cavity is $\mathrm{H_x} = \mathrm{H_y} = 1$ m, and its interior is filled with a Newtonian fluid. All boundaries are impermeable and adherent solid wall, i.e., no-slip conditions are applied. The top wall moves continuously to the right with a uniform velocity while the other walls are stationary. The lid motion drags the fluid adhered to it, and due to the viscous stress the flow is induced (Fortuna, 2000; Hirsch, 2007).

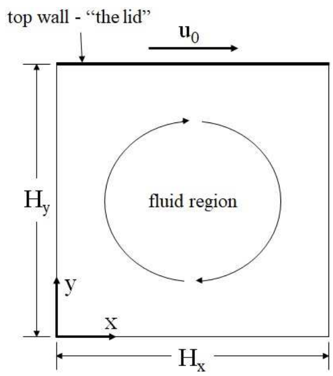


Figure 1: Schematic representation of the lid-driven square cavity problem.

2.2 Mathematical model

The fluid motion in the lid-driven cavity is modeled by the Navier-Stokes equations in primitive variables considering an incompressible, viscous, and Newtonian fluid flow. The mass conservation equation states:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

where u_i is the i^{th} component of the fluid velocity, and x_i is the i^{th} component of the space. The momentum conservation equation may be expressed as:

$$\frac{\partial \rho u_j}{\partial t} = A_j - \frac{\partial p}{\partial x_j} \tag{2}$$

where p is the pressure, ρ is the fluid density, and A_j is the j^{th} component of the convective and diffusive fluxes defined as:

$$A_j = \mu \frac{\partial^2 u_j}{\partial x_i^2} - \frac{\partial \rho u_i u_j}{\partial x_i} \tag{3}$$

in which μ denotes fluid dynamic viscosity.

Besides the Navier-Stokes equations, an addition one is needed to solve the flow pressure field. In this work, a Poisson equation for pressure is used. The Poisson equation may be obtained by taking the divergence of the momentum equation, Eq. 2, and its final form is expressed as (Hirsch, 2007):

$$\frac{\partial^2 p}{\partial x_j^2} = \frac{\partial A_j}{\partial x_j} - \frac{\partial}{\partial t}\left(\frac{\partial u_j}{\partial x_j}\right) \tag{4}$$

In this work, it is studied a two-dimensional lid-driven cavity with a domain extension of $[0, \mathrm{H_x}]\mathrm{x}[0, \mathrm{H_y}]$. For this geometry, the boundary conditions for the velocities are set as zero for all walls except for the lid where $u(x, H_y) = u_0$ with $x \in [0, H_x]$. The lid velocity, u_0, may be defined by the Reynolds number:

$$Re = \frac{u_0 H_x}{\nu} \tag{5}$$

where ν is the kinematic viscosity.

3. METHODOLOGY FOR THE NUMERICAL SOLUTION

In order to solve the conservation equations that model the fluid motion inside the lid-driven cavity (Eqs. 1, 2, and 4), a high-order two-dimensional Fortran code is developed. The Navier-Stokes equations are solved numerically on a collocated grid arrangement with uniform structured Cartesian mesh. The fourth-order centered finite-difference scheme is used to discretize the first and second spatial derivatives of these equations. An explicit fourth-order Runge-Kutta method is used to advance on time the solution. Below are detailed the steps to update the velocity on time using a fourth-order Runge-Kutta method.

The Runge-Kutta methods are a kind of predictor-corrector scheme. These methods introduce points between t^n and t^{n+1} and evaluate the right hand side (RHS) of the Eq. 2 at these intermediate points (Moin, 2010). Note that the RHS of Eq. 2 gathers all spatial derivatives. The fourth-order Runge-Kutta method employed in the present code uses four predictor steps and one corrector formula. Let k_1 be the first predictor step:

$$k_1 = \frac{\partial u}{\partial t}^n \tag{6}$$

To calculate the second predictor step, $k_2 = \frac{\partial u}{\partial t}^{n+\frac{1}{2}}$, need to estimate the value of $u^{n+\frac{1}{2}}$. For this purpose, use the value of k_1 in the following expression:

$$u^{n+\frac{1}{2}} = u^n + \frac{dt}{2}\, k_1 \tag{7}$$

To calculate the third predictor step, $k_3 = \frac{\partial u}{\partial t}^{n+\frac{1}{2}}$, the same previous procedure is applied. The $u^{n+\frac{1}{2}}$ is estimated using the value of k_2 in the following expression:

$$u^{n+\frac{1}{2}} = u^n + \frac{dt}{2}\, k_2 \tag{8}$$

Similar to the previous steps, to calculate the fourth predictor steps, $k_4 = \frac{\partial u}{\partial t}^{n+1}$, the u^{n+1} is given by the following expression:

$$u^{n+1} = u^n + dt\; k_3 \tag{9}$$

Finally, the solution at time step t^{n+1} is given by the corrector formula:

$$u^{n+1} = u^n + dt\; \frac{1}{6}\left(k_1 + 2k_2 + 2k_3 + k_4\right) \tag{10}$$

where dt is the time step.

Due to the explicit formulation of the time-marching method, the time step is set as the minimum between the convective limit, i.e., the Courant-Friedrichs-Lewy (CFL) condition

$$dt_{conv} < min\left(\frac{\Delta x_i}{|u_i|_{max}}\right) \tag{11}$$

and the diffusive limit,

$$dt_{dif} < min\left[\frac{1}{2\mu}\left(\frac{1}{(\Delta x_i)^2}\right)^{-1}\right] \tag{12}$$

multiplied by a safety factor. In this work, the safety factor value used is 0.5.

The Poisson equation for pressure is an elliptic equation. For this reason, the Gauss-Seidel method is adopted to solve it iteratively. In this work, the stopping criterion of the Gauss-Seidel method is defined as the absolute difference value of the pressure ϵ between two successive time steps,

$$\epsilon = \left|p^{k+1} - p^k\right| \tag{13}$$

where k is the last Gauss-Seidel iteration and $k+1$ is the present Gauss-Seidel iteration. When the maximum value of the ϵ is lower than 10^{-8}, the stopping criterion is satisfied.

Regarding the stopping criterion of the numerical simulation, the $\mathcal{L}_2$ norm of the temporal residuals of the momentum equations is adopted in which the results are considered converged and the steady state reached when the $\mathcal{L}_2$ norm is lower than 10^{-4}.

3.1 Algorithm description

The initial condition adopted is a solenoidal velocity field in which the fluxes A_j are computed following Eq. 3. As mentioned, for incompressible flows, the Poisson equation for pressure is typically elliptic, and then implies to solve an implicit system. After this step, the velocity field is explicitly updated to advance the solution on time by the fourth-order Runge-Kutta method (Eq. 10). With the new velocity fields, the fluxes A_j are again computed with Eq. 3. This algorithm is repeated until the stopping criterion of the simulation is satisfied. Figure 2 presents the flowchart of the algorithm in which can be seen all steps that the code goes through to obtain the steady-state solution of the conservation equations.

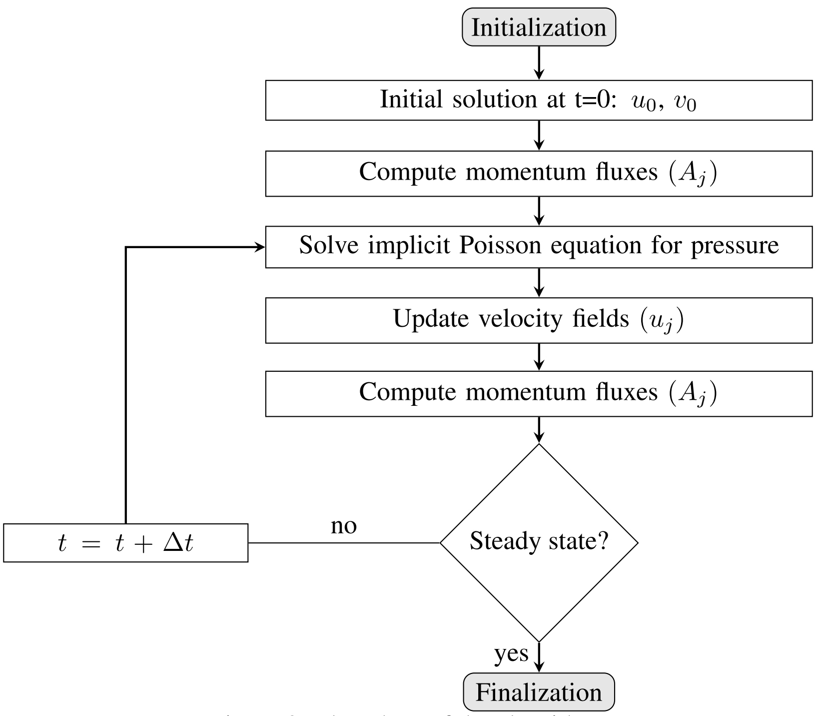
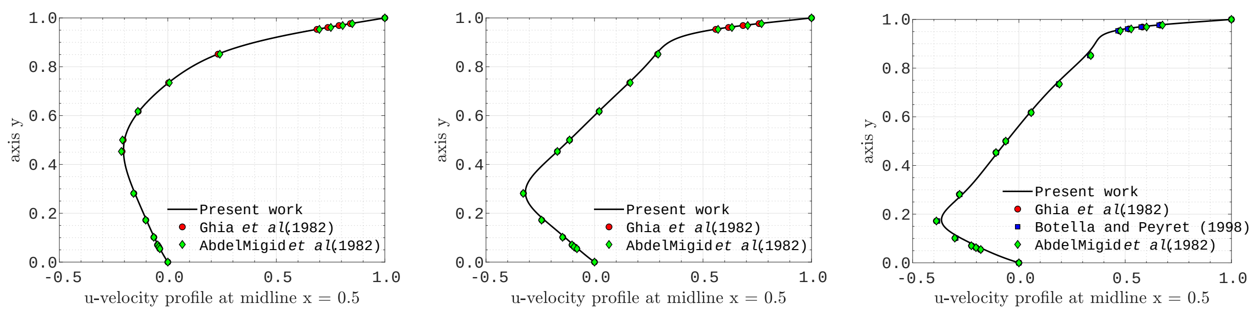


Figure 2: Flowchart of the algorithm.

4. RESULTS

In this section, the results of the velocity component profiles of the lid-driven cavity obtained through the in-house code are presented. Additionally, a comparative analysis between in-house code results with numerical works is performed.

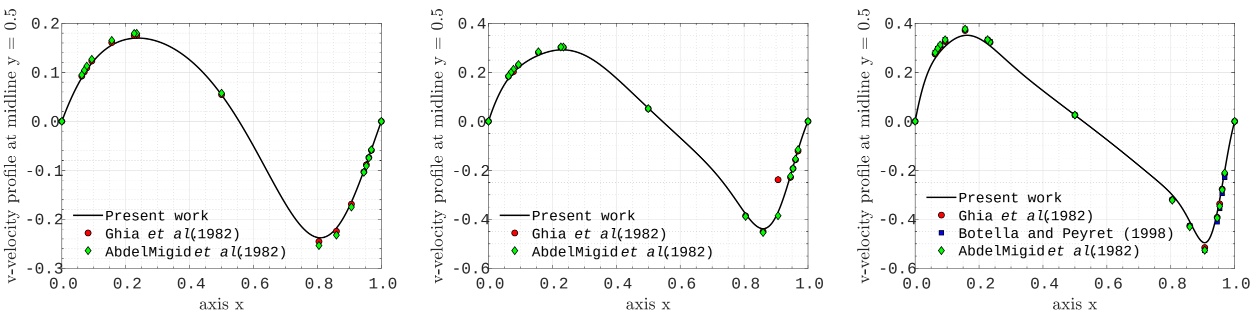
Figure 3 presents the u-velocity profiles at the vertical line passing through the geometric center of the square cavity for three different Reynolds numbers: 100, 400, and 1000. In Fig. 3(a), the results for Reynolds number 100 are showed and compared with numerical data of Guia *et al.* (1982) and AbdelMigid *et al.* (2017). For this case, a mesh of 129×129 nodes is used. Figures 3(b) and 3(c) present the results for Reynolds number 400 and 1000, respectively. In both cases, a mesh of 321×321 nodes is used. For Reynolds number 1000 (Fig. 3(c)), the results are also compared with the spectral solution of Botella and Peyret (1998).



(a) $Re = 100$ (b) $Re = 400$ (c) $Re = 1000$

Figure 3: Horizontal velocity along a vertical centerline passing through the geometric center of the square cavity.

Figure 4 shows the v-velocity profiles at the horizontal line passing through the geometric center of the square cavity for Reynolds numbers 100, 400, and 1000. The numerical works presented previously are used here in the comparative analysis of the v-velocity profiles. Analyzing Figs. 3 and 4, it can be seen that the present results compare generally well with the reference solution for the three studied Reynolds numbers.



(a) $Re = 100$ (b) $Re = 400$ (c) $Re = 1000$

Figure 4: Vertical velocity along a horizontal centerline passing through the geometric center of the square cavity.

5. CONCLUSIONS

In this work, a high-order two-dimensional Fortran code capable of determining the unsteady velocity and pressure fields of incompressible flows is proposed. Additionally, previous numerical works of the lid-driven cavity problem are used to asses and to verify the code. The in-house code results are compared with numerical works for Reynolds number 100, 400, and 1000. A good agreement is observed.

In future works, the parallelization of this solver based on a domain decomposition using the open library PETSc will be presented. Besides, the DNS (Direct Numerical Simulations) results of incompressible turbulent flows are reserved for a future paper.

6. ACKNOWLEDGEMENTS

The authors would like to thank the scholarship supported by the Coordenação de Aperfeiçoamento de Pessoal de Nível Superior - Brasil (CAPES) - Finance Code 001.

7. REFERENCES

AbdelMigid, T.A., Saqr, K.M., Kotb, M.A. and Aboelfarag, A.A., 2017. "Revisiting the lid-driven cavity flow problem: Review and new steady state benchmarking results using gpu accelerated code". *Alexandria Engineering Journal*, Vol. 56, No. 1, pp. 123–135.

Botella, O. and Peyret, R., 1998. "Benchmark spectral results on the lid-driven cavity flow". *Computers & Fluids*, Vol. 27, No. 4, pp. 421–433.

Burggraf, O.R., 1966. "Analytical and numerical studies of the structure of steady separated flows". *Journal of Fluid Mechanics*, Vol. 24, No. 1, pp. 113–151.

Fortuna, A.O., 2000. *Técnicas computacionais para dinâmica dos fluidos: conceitos básicos e aplicações*. Editora da Universidade de São Paulo, São Paulo, 2nd edition.

Guia, U.K.N.G., Guia, K.N. and Shin, C.T., 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.

Hirsch, C., 2007. *Numerical computation of internal and external flows: The fundamentals of computational fluid dynamics*. Elsevier, Oxford, 2nd edition.

Lomax, H., Pulliam, T.H. and Zingg, D.W., 2001. *Fundamentals of computational fluid dynamics*. Springer-Verlag Berlin Heidelberg, New York, 1st edition.

Moin, P., 2010. *Fundamentals of engineering numerical analysis*. Cambridge University Press, New York, 2nd edition.

Sagaut, P., 2006. *Large eddy simulation for incompressible flows: an introduction*. Springer-Verlag Berlin Heidelberg, New York, 3rd edition.

Shankar, P.N. and Deshpande, M.D., 2000. "Fluid mechanics in the driven cavity". *Annual review of fluid mechanics*, Vol. 32, No. 1, pp. 93–136.

8. RESPONSIBILITY NOTICE

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