Analysis of Fluid Flow Between Two Parallel Plates Using Different Numerical Methods

Petar Miljković[1[0000-0003-0450-3599]], Branka Radovanović[1[0000-0002-5706-8757]], Veljko Begović[1[0000-0001-8578-8287]] and Miloš Jovanović[1]

[1] University of Niš, Faculty of Mechanical Engineering, Serbia
petar.miljkovic@masfak.ni.ac.rs

Abstract. In this paper, the fluid flow between two parallel plates is considered using the SIMPLE method in the MATLAB software package. Velocity changes as a function of time is examined. The results obtained numerically coincided with the results given by the SIMPLE method. The ANSYS software package, module CFX was used for numerical proof. The acronym SIMPLE stands for Semi-Implicit Method for Pressure Linked Equations. The algorithm was originally put forward by Patankar and Spalding and is essentially a guess and-correct procedure for the calculation of pressure on the staggered grid. The method is illustrated by considering the two-dimensional laminar steady flow equations in Cartesian coordinates.

Keywords: SIMPLE method, parallel plates, fluid flow, momentum transfer, CFX.

1 Introduction

This paper deals with the case of viscous, incompressible fluid flow in a channel, between two parallel plates located at a distance h, where the length of the plates is L. We examine fluid flow in a two-dimensional plane along the x and y axes, assuming no change in physical size along the third axis. In other words, the flow occurs per unit width of the plate in the z-axis direction. Our example is fluid flow when the bottom plate transitions from a state of rest to a state of motion with an initial velocity v_0. Special attention is paid to the influence of the movement of the lower plate on the formation of the velocity field. When a viscous fluid flows in a parallel plate channel, a velocity boundary layer develops along the inner surfaces of the channel.

In the first part of this study, the mentioned phenomenon was analyzed using numerical simulations, specifically the SIMPLE method. In the second part of the study, CFD (Computational Fluid Dynamics) analysis was conducted to validate the obtained results. The commercial software package ANSYS CFX was used for the CFD analysis. Unlike the system codes, the CFD codes estimate the pressure drop from the velocity profile which is obtained by solving momentum conservation equations, and the resulting friction factor can be a representative parameter for a constant cross section channel flow [3].

2 Numerical analysis

To solve the problem, the SIMPLE method was used through the MATLAB program. The SIMPLE method is suitable for use because with minimal changes in the initial equations we can obtain different parameters. The method is iterative and is very suitable for solving Navier-Stokes equations numerically.

Two-dimensional laminar steady flow is described by the following equations (1-3): Continuity equation

$$\frac{\partial}{\partial x}\left(\rho v_x\right)+\frac{\partial}{\partial y}\left(\rho v_y\right)=0 \tag{1}$$

x momentum equation:

$$\frac{\partial \rho v_x}{\partial t}+\frac{\partial}{\partial x}\left(\rho v_x v_x\right)+\frac{\partial}{\partial y}\left(\rho v_y v_x\right)=\frac{\partial}{\partial x}\left(\mu \frac{\partial v_x}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu \frac{\partial v_x}{\partial y}\right)-\frac{\partial p}{\partial x}+S_{v_x} \tag{2}$$

y momentum equation:

$$\frac{\partial \rho v_y}{\partial t}+\frac{\partial}{\partial x}\left(\rho v_x v_y\right)+\frac{\partial}{\partial y}\left(\rho v_y v_y\right)=\frac{\partial}{\partial x}\left(\mu \frac{\partial v_y}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu \frac{\partial v_y}{\partial y}\right)-\frac{\partial p}{\partial y}+S_{v_y} \tag{3}$$

where:

- v_x – velocity component in the x direction
- v_y – velocity component in the y direction
- ρ – density of the fluid
- μ – dynamic viscosity coefficient of the fluid,
- p – pressure
- t – time
- S_{v_x}, S_{v_y} – source or sink.

In order to solve (1-3) using numerical techniques, CFD problems need to be discretized in space dimensions [1]. For the discretization of these equations, staggered grid is used, shown in Fig. 1.

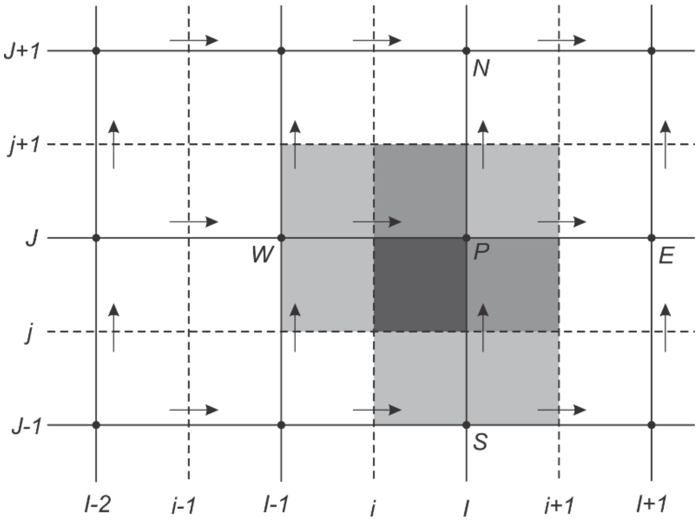

Fig. 1. **S**taggered grid (→ denote v_x velocities, ↑ denote v_y velocities)

Finally, the discretised moment equations and the continuity equation read:

$$a_{I,J}\, p'_{I,J} = a_{I+1,J}\, p'_{I+1,J} + a_{I-1,J}\, p'_{I-1,J} + a_{I,J+1}\, p'_{I,J+1} + a_{I,J-1}\, p'_{I,J+1} + b'_{I,J} \tag{4}$$

$$\frac{a_{i,J}}{\alpha_{v_x}} v_{xi,J} = \sum a_{nb} v_{xnb} + \left(p_{I-1,J} - p_{I,J}\right) A_{i,J} + b_{i,J} + \left[\left(1-\alpha_{v_x}\right)\frac{a_{i,J}}{\alpha_{v_x}}\right] v_{xi,J}^{(n-1)} \tag{5}$$

$$\frac{a_{I,j}}{\alpha_{v_y}} v_{yI,j} = \sum a_{nb} v_{ynb} + \left(p_{I,J-1} - p_{I,J}\right) A_{I,j} + b_{I,j} + \left[\left(1-\alpha_{v_y}\right)\frac{a_{I,j}}{\alpha_{v_y}}\right] v_{yI,j}^{(n-1)} \tag{6}$$

where:

- α_{v_x}, α_{v_y} – under-relaxation factors
- a – central coefficients of discretised velocity equations
- b – the source term

It is very important to choose an appropriate value of the under-relaxation factor. Choosing small values can result in extremely slow convergence, while a large value can lead to divergence [2].

2.1 Results SIMPLE method

In this part, we present the results obtained by numerical calculation of previously described equations and their physical interpretation. Fig. 2. illustrate values of the speed profile v_x for different time moments $t = 0.001s$; $0.005s$; $0.010s$; and further with the step $\Delta t = 0.01s$, where in our case, the total number of steps is $n = 16$. The lowest line represents the velocity distribution at time $t = 0.001s$ and it can be seen that the velocity of the fluid in contact with the bottom plate is transmitted through the momentum transfer due to the viscosity of the fluid vertically to the adjacent fluid layers. The fluid in contact with the lower plate has the velocity of the lower plate due to the sticking condition of the fluid to the lower plate of the channel, therefore the boundary condition is $v_x(y{=}0, t{>}0){=}v_0$, while the boundary condition on the upper plate, which is stationary all the time is $v_x(y{=}h){=}0$. At the initial time ($t = 0s$) both plates are at rest, and the same is true for all points of the fluid at that time. For calculation we are using normalized values of density, viscosity, speed and height channels, i.e. $\rho = 1.0$; $\nu = 1.0$; $v_0 = 1.0$; $h = 1.0$. So from Fig. 2. it can be seen that at the moment $t = 0.001s$ fluid velocity $v_x(y/h < 0.18) > 0$, i.e., that 18% of the height of the channel is affected by the momentum transfer of fluid from the lower plate which was brought to the state of movement from the state of rest by a bounce function.

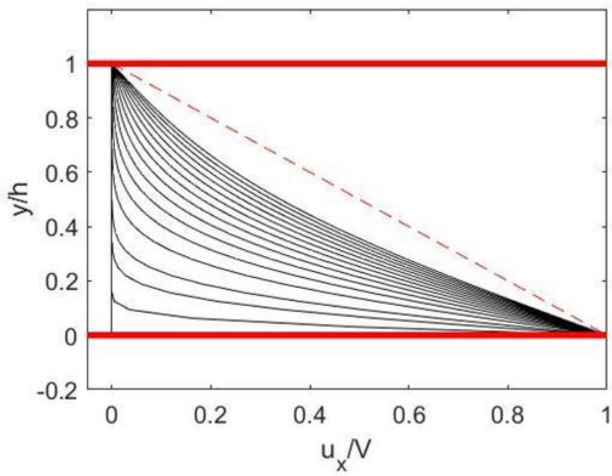

Fig. 2. Velocity distribution $v_x(y/h)$ of the viscous, incompressible fluid between the plates during sudden movement of the lower plate

Furthermore, we observe that the value of the velocity v_x at that moment is very small, $v_x(0.10{<}y/h{<}0.18){<}0.1v_0$, and that the higher values of this velocity are found in the layer closer to the plate, i.e. next to the plate itself, and that the change is quite sudden in that part, i.e that the first derivative of the velocity by coordinate y has a significant value, in other words $dv_x/d_y \gg 1$. Since the speed of momentum transfer in the vertical direction from the plate to the adjacent horizontal fluid layers is proportional to this first derivative, i.e. $\tau_{xy} = \mu dv_x/dy$ we can conclude that this transmission is the most intense at this moment in time, and that all the more if we are closer to the plate. At the following moment of time $t = 0.005s$, we observe that the area affected by the movement of the fluid is expanded to a height of $y/h \approx 0.3$ and that velocity values $v_x(y/h)$ in all layers of the fluid affected by the movement are higher than the velocity values for the previous moment of time, that is, that there was an acceleration of the fluid particles in the horizontal layers, which were affected by the momentum transfer and the fluid layers that were in a state of rest at the previous moment. The latter refers to the layer of fluid that was at a height $(0.18 < y/h < 0.30)$, and which in the meantime was affected by the momentum transfer upwards. The next line refers to the time instant $t = 0.01s$ and it is the third line from the bottom, it shows how time change has led to further expansion of the fluid area which is set in motion in the x-axis direction. Here it can be seen that now almost half of the channel is affected by movement parallel to the plates, where the velocity values for $y/h \approx 0.5$ are quite small $v_x(y/h \approx 0.4) < 0.01v_0$. The velocity gradients are now slightly smaller than they were in the previous two instants, which indicates that the momentum transfer via the shear stress $\tau_{xy} = dv_x/dy$ is now slightly smaller in all layers compared to the previous instants of time when it was significantly more intense. The Fig. 2. also shows the results for the remaining 13 time moments, and it is clearly seen that the final solution tends to a linear speed distribution between the plates, which is achieved when $t \to \infty$.

3 CFD analysis

ANSYS software is a suite of engineering simulation tools developed by ANSYS, Inc. Engineers and designers widely use it to simulate and analyze various physical phenomena in order to optimize designs, solve complex problems, and make informed engineering decisions. ANSYS offers a comprehensive range of simulation capabilities, covering structural mechanics, fluid dynamics, electromagnetics, Multiphysics, and more. The software enables engineers to simulate and evaluate the behavior of physical systems under different conditions without the need for costly and time-consuming physical prototypes [4].

Some key features of ANSYS software include:

- Finite Element Analysis (FEA), ANSYS allows engineers to perform FEA to analyze the structural integrity, strength, and deformation of components and systems.
- Computational Fluid Dynamics (CFD), ANSYS CFD tools simulate fluid flow, heat transfer, and related phenomena, helping engineers optimize designs for aerodynamics, HVAC systems, and more.
- Electromagnetics Simulation, Multiphysics Simulation, Optimization and Design Exploration, System-Level Simulation, and more.

ANSYS is widely used in industries such as aerospace, automotive, energy, electronics, civil engineering, and many others. It gives engineers powerful tools to analyze and optimize designs, reduce development costs, and enhance product performance and reliability.

CFD is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems involving fluid flow, heat transfer, and other related phenomena. The software uses the finite volume method to discretize the governing equations of fluid flow and solves them numerically on a computational grid. With ANSYS CFX, users can create detailed three-dimensional models of their systems, define boundary conditions, specify fluid properties, and set up simulation parameters. The software then performs simulations to predict and visualize fluid behavior, such as velocity distribution, pressure distribution, temperature distribution, and other related quantities. ANSYS CFX is a computational fluid dynamics (CFD) software developed by ANSYS. It is a powerful tool used for simulating and analyzing fluid flow and heat transfer problems.

3.1 Problem Set Up

The geometry of the model under consideration is simple, and the Design Modeler tool within the ANSYS CFX package was used for its generation. The next step is generating a mesh for the domain under consideration, and for this process, the Mesh tool within the ANSYS CFX package was used. The mesh used in this case is a hexahedral mesh, in order to discretize the domain into a large number of cells, or control volumes. The domain is divided into a large number of cells, based on the size of the minimum element. The optimal number of elements was chosen in order to save computational effort without compromising the accuracy of the results. The advantage of such a mesh is better convergence of results and better orthogonal quality of the mesh, which in this case was taken as the most important criterion. This kind of mesh is also known for its numerical stability.

The mesh used in the analyzes is shown in Fig. 3. It is important to note that ANSYS CFX does not have the capability to solve 2D problems, so it was necessary to define a 3D domain, which was achieved by specifying the channel width. Additionally, a Symmetry boundary condition was defined for the XY plane.

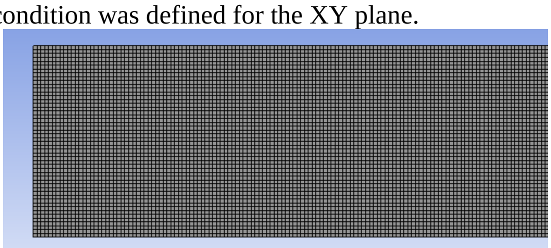

Fig. 3. Mesh used in CFD analysis

The basic requirements for CFD modeling were that the flow between the plates should be laminar. In this example, the chosen flow model is the laminar model. This model was chosen due to the characteristics of the problem and the goal of the analysis. The laminar flow model has several advantages. One of them is that it provides more accurate results in situations where the fluid flow is mostly laminar, i.e. when there are no significant turbulent effects. These models are particularly useful in situations with low flow velocity, low Reynolds number, and high fluid viscosity. Also, laminar flow models require fewer computing resources compared to turbulent models. This is particularly important when analyzing large and complex systems, as the simulation time can be significantly reduced. Fewer assumptions about boundary conditions are needed, ie. in some cases, they can be simplified a lot.

What is important to note in this case is that the simulation is non-stationary since it was necessary to analyze the change of the system over time. The simulation was performed step by step, wherein differential equations describing the change of the system during small-time intervals were solved. In the first case, the total duration was 1s, with a time step of 0.005s, and 20 iterations per time step were monitored. It was concluded that the system reaches equilibrium at a time delay of 0.35s. For this reason, the following case was approached, where the total duration of the simulation was set to 0.35s, using a time step of 0.001s with 20 iterations per each time step.

Solutions are started from a state of rest, i.e. it is important to note that at the initial moment, the plate was not moving.

The boundary conditions with indicated positions in Fig. 4 are provided in the list below:

1. Inlet - Opening
2. Outlet - Average Static Pressure 0 Pa
3. Top plate - No Slip Wall
4. Bottom plate - No Slip Wall, but with a speed in the direction of the x axis, U=1 m/s.

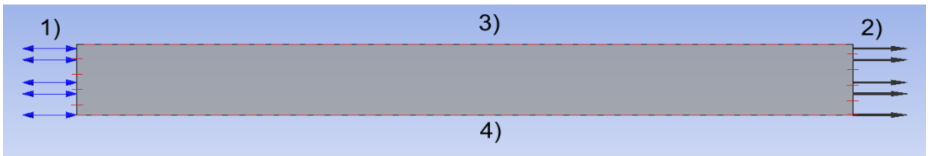

Fig. 4. Domain, with boundary conditions: 1) Inlet, 2) Outlet, 3)Top Plate, 4) Bottom plate

3.2 Results of CFD analysis

Below are the results for characteristic time instants. The results for each moment in time are represented by a diagram of velocity distribution depending on the coordinate y and a contour diagram where it can be clearly seen which fluid layers are moving.

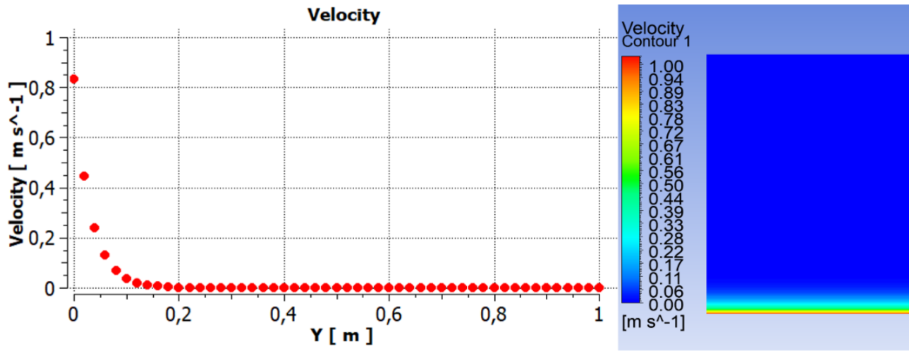

Fig. 5. Results for a moment in time 0.001 s

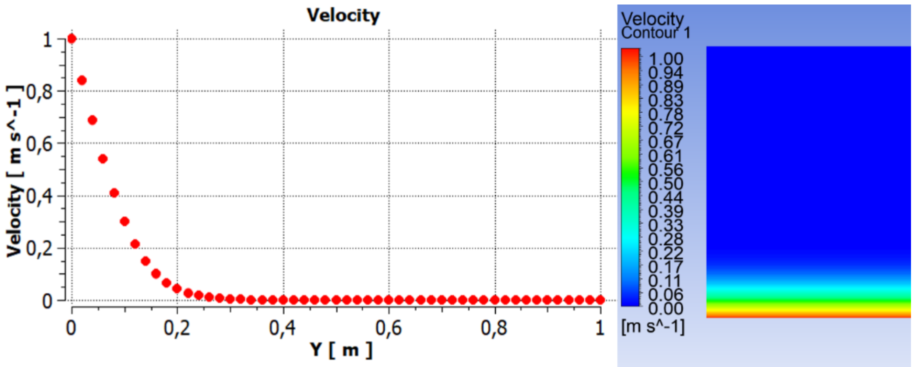

Fig. 6. Results for a moment in time 0.005 s

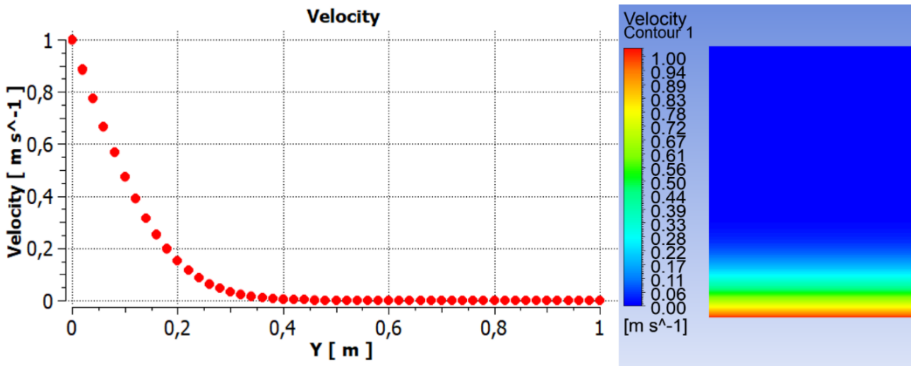

Fig. 7. Results for a moment in time 0.01 s

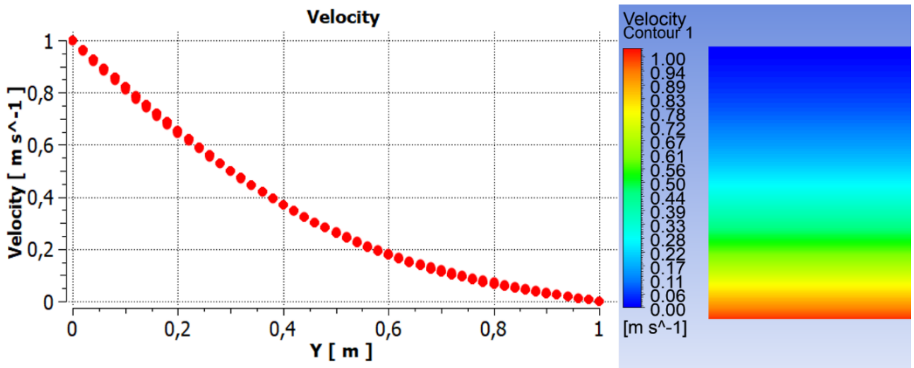

Fig. 8. Results for a moment in time 0.1 s

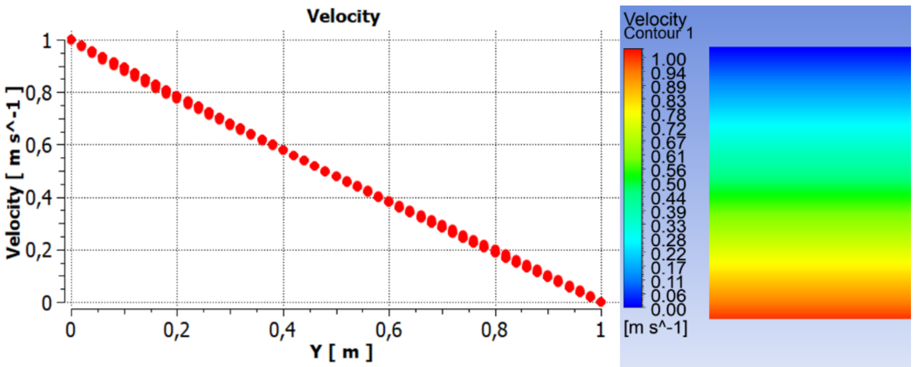

Fig. 9. Results for a moment in time 0,35 s

Characteristic moments in time are presented that coincide with the moments in time analyzed in the first part. Fig. 5 shows the moment in time immediately after the lower plate is started, it can be seen that only the layers non-parallel to the lower plate are moved. In the next two moments of time in Fig. 6 and Fig. 7. it can be observed that a significantly larger number of layers are moving. Fig. 5 represents the moment of time 0.1s from the beginning of the plates movement, and it can be seen that the speed changes are no longer as pronounced as in the initial moments, until the moment of time 0.35s (Fig. 9) we have a slight change of the velocity field until the state of equilibrium, as it was previously emphasized that is the moment when the system is balanced.

4 Conclusion

In that asymptotic case, we have that the shear stress field in all fluid layers has the same value because $dv_x/dy = const$. and we call such a field of shear stress homogeneous, which means that it is the momentum transfer from the lower plate through the fluid to the upper plate established an equilibrium state, and the same transfer rate in all layers of the viscous fluid. The numerical results obtained by MATLAB and ANSYS coincide with each other.

Acknowledgements

This research was financially supported by the Ministry of Science, Technological Development and Innovation of the Republic of Serbia (Contract No. 451-03-47/2023-01/ 200109)

References

1. Khawaja H., Moatamedi M.: Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) - solution in MATLAB. International Journal of Multiphysics (12), 313–325 (2018)
2. Versteeg H. i W.Malalasekera: Introduction to Computational Fluid Dynamics, The Finite Volume Method, 2nd Edition, Pearson, Harlow (2007)
3. Hyung Min Son , Soo Hyung Yang , Jong Hark Park: Computational Fluid Dynamic Pressure Drop Estimation of Flow between Parallel Plates. Transactions of the Korean Nuclear Society Autumn Meeting Pyeongchang, Korea, October 2014
4. Engin Gedik , Hüseyin Kurt , Ziyaddin Recebli , Corneliu Balan: Two-dimensional CFD simulation of magnetorheological fluid between two fixed parallel plates applied external magnetic field.
5. LNCS Homepage, http://www.springer.com/lncs, last accessed 2016/11/21.