DOI: 10.37943/19QEOK8161

Sultan Alpar
PhD, Associate Professor, Mathematical and Computer Modelling Department
s.alpar@iitu.edu.kz , orcid.org/0000-0003-0579-4180
International Information Technology University, Kazakhstan

Fatima Tokmukhamedova
MSc, Assistant-professor, Mathematical and Computer Modelling Department
f.tokmukhamedova@iitu.edu.kz, orcid.org/0000-0001-7980-8245
International Information Technology University, Kazakhstan

Bakhyt Alipova
PhD, Associate Professor, Mathematical and Computer Modelling Department
b.alipova@iitu.edu.kz, orcid.org/0000-0003-0915-2759
International Information Technology University, Kazakhstan
University of Kentucky, Department of Mechanical and Aerospace Engineering, USA

Yevgeniya Daineko
PhD, Associate Professor, Vice-Rector for Global Partnerships and Continuing Education
y.daineko@iitu.edu.kz, orcid.org/0000-0001-6581-2622
International Information Technology University, Kazakhstan

Nazerke Rysbek
BSc Student
rysbeknazerke2002@gmail.com , orcid.org/0009-0009-2015-7754
International Information Technology University, Kazakhstan

Diyar Abdrakhman
BSc, Master's student
slashhgm20032@gmail.com , orcid.org/0009-0009-6030-9847
International Information Technology University, Kazakhstan

MODELING OF BLOOD FLOW IN LAMINAR MODE

Abstract: This article presents a detailed analytical evaluation and comprehensive description of a mathematical model designed to simulate blood flow within the human cardiovascular system. The primary objective of this research is to develop a computational model capable of accurately simulating blood flow dynamics and to assess the variations in results using different numerical methods for solving the Navier-Stokes equations, which govern fluid motion. To achieve this, the study begins with an in-depth examination of the anatomy of the cardiovascular system, including various cardiovascular diseases such as stenosis and atherosclerosis, which significantly affect blood flow. The model incorporates important characteristics of blood, treating it as a viscous fluid under laminar flow conditions. Using the Navier-Stokes equations, it was developed a Python-based model to simulate these flow conditions and solve for different flow variables, such as velocity and pressure fields, under both normal and pathological conditions. The computational model was developed using two numerical methods: the Euler method and the Alternating Direction Implicit (ADI) method, which were compared in terms of their computational efficiency and accuracy. The simulations generated insights into how plaque buildup (stenosis) affects blood flow by altering wall shear stress and

Received: 27.06.2024 Accepted: 04.09.2024 Published: 30.09.2024

Scientific Journal of Astana IT University
ISSN (P): 2707-9031 ISSN (E): 2707-904X
VOLUME 19, SEPTEMBER 2024

velocity profiles. This model, while built on foundational fluid dynamics principles, serves as an essential step towards creating a virtual reality (VR) surgical simulator for cardiovascular procedures. This simulator aims to assist surgeons in visualizing and planning surgical interventions by providing an interactive and realistic environment for studying blood flow and related complications.

Keywords: blood flow modeling, alternating direction implicit method, blood hydrodynamics, modeling of cardiovascular diseases.

Introduction

The study of blood flow dynamics within the cardiovascular system has garnered significant attention due to its critical role in understanding and diagnosing cardiovascular diseases (CVDs). Blood flow characteristics, such as velocity, pressure, and wall shear stress, are essential factors in determining the progression of diseases like atherosclerosis and stenosis, which are leading causes of morbidity and mortality worldwide. Traditional medical imaging methods, such as computed tomography (CT) and magnetic resonance imaging (MRI), provide anatomical information but fall short in accurately predicting hemodynamic changes that precede clinical symptoms. Computational Fluid Dynamics (CFD), on the other hand, has emerged as a powerful tool for simulating blood flow and detecting early signs of cardiovascular disease.

Several studies have utilized CFD models based on the Navier-Stokes equations to simulate blood flow in arteries and predict hemodynamic factors under both normal and pathological conditions. For instance, [1] employed cylindrical coordinates to model blood flow in stenotic arteries and demonstrated the importance of shear stress distribution in disease progression. Similarly, [2] used a finite element approach to examine blood flow in bifurcating vessels but noted computational limitations when simulating complex geometries.

Despite these advances, many existing models rely on simplified assumptions or struggle with computational inefficiencies, particularly when solving the Navier-Stokes equations for complex cardiovascular geometries. To address these challenges, this study compares two numerical methods for solving the Navier-Stokes equations: the Euler method and the Alternating Direction Implicit (ADI) method. While both methods have been widely used in CFD, their relative strengths and weaknesses in modeling cardiovascular dynamics, particularly under pathological conditions such as plaque formation, have not been thoroughly explored.

A steady and pulsating flow at the site of carotid artery bifurcation with stenosis was studied using smoothed particle hydrodynamics (SPH) modeling. A steady flow directs blood to the internal carotid artery, while a pulsating flow causes significant fluctuations in velocity and pressure. The pulsating flow rates are 3.5 times higher and the pressure is 25% lower than with constant flow, which emphasizes the importance of understanding local flow stresses in the development of cardiovascular diseases [3]. The finite element model of the arteries of the human arm shows shear stress, mainly at the bifurcation sites, with the highest pressure in the radial artery during maximum velocity [4]. This model helps to noninvasively track pulse waveforms, constantly improving modeling and bidirectional fluid-structure communication. The shear stress of the wall is crucial for studying the effect of blood flow on endothelial cells [5]. The height of the stenosis affects the distribution of shear stress and the area of flow separation, while narrowing of the artery increases the peak shear stress of the wall without changing the structure of blood flow, which is important for analyzing the progression of cardiovascular diseases [6].

The primary objective of this study is to evaluate the computational efficiency and accuracy of the Euler and ADI methods for simulating blood flow in the cardiovascular system. Specifically, the aim is to: compare the computational time and accuracy of the Euler and ADI methods under physiological conditions; analyse the effects of plaque on blood flow dynamics, particularly in terms of shear stress and velocity profiles and assess the applicability of these methods in simulating various pathological conditions, including stenosis and pulsatile flow.

This study hypothesizes that while both methods can produce comparable results in terms of accuracy, the ADI method will prove more computationally efficient, particularly in scenarios involving complex geometries such as stenotic arteries. Additionally, the presence of plaque is expected to significantly alter shear stress distribution, a critical factor in understanding the progression of cardiovascular diseases.

Several computational studies have demonstrated the efficacy of CFD in predicting cardiovascular flow dynamics. [3] showed that cylindrical coordinate systems provide more accurate results for simulating arterial blood flow, though the computational cost remains high. In contrast, Cartesian coordinates, while less accurate in replicating the curved geometry of vessels, offer computational simplicity. The use of the Euler method in blood flow modeling has been well-documented, but its stability limitations often necessitate smaller time steps, which increase computational time. Meanwhile, the ADI method, known for its stability and efficiency in handling multidimensional data, has not been extensively applied to cardiovascular simulations, particularly under conditions involving plaques and stenosis.

While previous studies have primarily focused on either improving the accuracy of CFD models or optimizing computational time, few have directly compared the Euler and ADI methods in the context of cardiovascular disease modeling. This research fills that gap by providing a comprehensive comparison of these two methods, assessing their performance in simulating both normal and pathological blood flow conditions.

The mathematical and computational models developed in this work lay the foundation for simulating real-time blood flow dynamics in a virtual surgical setting. By accurately modeling how blood flow behaves in the presence of plaques, stenoses, and other pathologies, the simulator can offer valuable insights into potential complications and outcomes during surgery. This approach has significant clinical implications, as it could not only enhance diagnostic accuracy but also optimize surgical treatment strategies by allowing pre-operative simulations of different procedural approaches.

The remainder of this paper is organized as follows. The Materials and Methods section describes the numerical models, boundary conditions, and assumptions made in this study. The Results section presents a detailed comparison of the Euler and ADI methods. Finally, the Conclusion outlines the key outcomes of the research, the limitations of the current model, and directions for future work.

Methods and materials

Blood flow in laminar conditions can be modeled using the incompressible form of the Navier-Stokes equations, which describe the motion of viscous fluids with constant density. The equations are based on the concepts of conservation of mass and momentum and can be adjusted according to the problem's substance. According to the principle of conservation of mass, there is no mass flow differential between the system's input and outflow [7].

The motion of a viscous fluid is described by the Navier-Stokes equation:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial P}{\partial x} + \nu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{1.1}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial P}{\partial y} + \nu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{1.2}$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1.3}$$

Scientific Journal of Astana IT University
ISSN (P): 2707-9031 ISSN (E): 2707-904X
VOLUME 19, SEPTEMBER 2024

where t – time, [sec]; $u(x, y, t)$ and $v(x, y, t)$ – velocity by X and Y axis $[m/sec]$; ρ – fluid density, $[kg/m^3]$; $P(x, y, t)$ – pressure, [Pa]; v – viscosity, [Pa * sec].

Nondimensionalization simplifies the complex equations by reducing the number of physical parameters and highlighting core physical phenomena through the introduction of dimensionless numbers like the Reynolds number.

$$\bar{u} = \frac{u}{u_0}, \quad \bar{v} = \frac{v}{u_0}, \quad \bar{t} = \frac{t}{T},$$
$$\bar{x} = \frac{x}{L}, \quad \bar{y} = \frac{y}{L}, \quad \bar{p} = \frac{P}{P_0}.$$

For convenience the final result will be written without dash lines:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{\partial P}{\partial x} + \frac{1}{Re}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{1.4}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{\partial P}{\partial y} + \frac{1}{Re}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{1.5}$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1.6}$$

where $Re = \frac{u_0 L \rho}{v}$

A physiological scenario can only be accurately simulated when boundary conditions are chosen carefully when modeling blood flow with the Navier-Stokes equations.

$$\begin{array}{lll}
\text{Inlet}(x = 0,\ y): & u = u_{inlet}, & v = v_{inlet}, \\
\text{Outlet}(x = x_{max},\ y): & \dfrac{\partial u}{\partial x} = 0, & \dfrac{\partial v}{\partial x} = 0, \\
\text{No-Slip Boundary}(x,\ y = 0): & u = 0, & v = 0, \\
\text{No-Slip Boundary}(x,\ y = y_{max}): & u = 0, & v = 0,
\end{array}$$

All this boundary condition's illustrated in Figure 1. For pressure, apply only the Dirichlet condition [8]. It can be expressed as follows:

$$p(0, y) = 0, \quad p(x_{max}, y) = 0,$$
$$p(x, 0) = 0, \quad p(x, y_{max}) = 0,$$

While this study utilizes a cartesian grid for modeling blood flow and stenosis, it is acknowledged that this approach introduces limitations in terms of accurately reproducing the complex geometry of blood vessels, particularly in the case of curved structures. Blood vessels are more naturally represented using cylindrical coordinates, which better capture their circular cross-section and curvature. However, to simplify the numerical implementation and focus on the comparison of computational methods (Euler and ADI), a 2D Cartesian approximation was adopted.

The Cartesian grid allows for simpler boundary condition implementation and computational efficiency in the initial phase of modeling. Nevertheless, this assumption does result in the loss of certain geometric fidelity, and as a result, the model may not entirely replicate the true dynamics found in physiological blood vessels. Specifically, the assumption of an infinite vessel size in the direction orthogonal to the grid plane (Oz) simplifies the representation of flow, particularly in regions of curvature or bifurcation.

While accurate reproduction of real physiological conditions is not fully achievable using this grid structure, the chosen model still captures the core hemodynamic phenomena associated with stenosis, such as increased wall shear stress and altered velocity profiles. Future work will extend this model to cylindrical coordinates and incorporate the elasticity of vessel walls to enhance the physiological accuracy of the simulations.

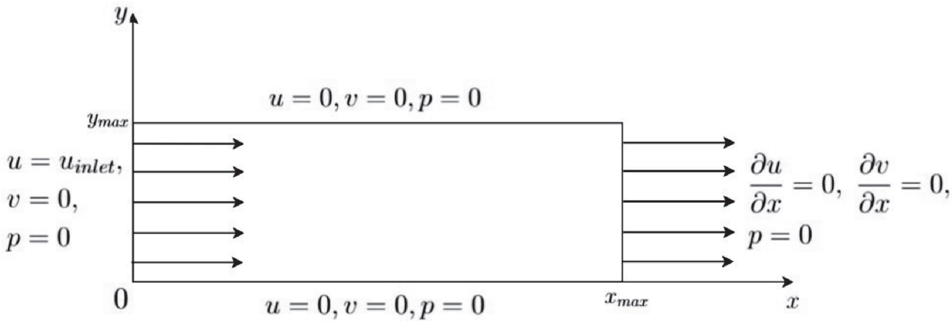

Figure 1. Overall working domain with schematic boundary conditions

For a two-dimensional (2D) Navier-Stokes simulation of blood flow, the initial condition typically involves specifying the initial velocity field, $u(x, y, t = 0)$, and possibly the initial pressure distribution, $\rho(x, y, t = 0)$ throughout the computational domain. Assuming a quiescent initial state, where the fluid is initially at rest, the initial velocity field can be expressed as:

$$u(x, y, t = 0) = \begin{bmatrix} u_0(x, y) \\ v_0(x, y) \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}$$

Here, $u_0(x, y)$ and $v_0(x, y)$ represent the initial velocity components in the x and y directions, respectively.

The initial pressure field, $\rho(x, y, t = 0)$ would then be expressed as:

$$p(x, y, t = 0) = 0$$

A popular numerical method for solving the pressure and velocity coupling in computational fluid dynamics (CFD) simulations, especially in incompressible flows, is the semi-implicit method for pressure-linked equations (SIMPLE).

The divergence of the velocity field, which is proportional to the pressure gradient, is how the velocity and pressure fields are related in the context of the incompressible Navier-Stokes equations. By adding a pressure-correction step, the semi-implicit method decouples the velocity and pressure fields, enabling a more stable and effective numerical solution [9].

The SIMPLE method for solving incompressible two-dimensional Navier-Stokes equations consists of three main stages:

1. $\int_{\Omega}^{1} \frac{u^* - u^n}{\tau} d\Omega = -\oint_{\Omega}^{1} (\nabla \cdot u^n u^* - \nu \Delta u^*) n_i d\Gamma,$

- the prediction stage, which includes the calculation of convective and diffusion terms in the momentum equations using the current velocity field.

2. $\oint_{d\Omega}^{1} (\Delta \mathrm{p}) d\Gamma = \oint_{\Omega}^{1} \frac{\nabla \cdot u^*}{\tau} d\Omega,$

- the pressure correction stage, which includes solving the pressure correction equation to determine the pressure field based on the velocity field obtained at the pulse prediction stage.

3. $\frac{u^{n+1} - u^*}{\tau} = -\nabla \cdot p,$

- the velocity correction stage, which includes the estimation of the velocity field using the pressure correction obtained in the previous stage.

Two popular methods for solving Navier – Stokes equations in computational fluid dynamics (CFD) are the Euler method for time integration and the Alternating Direction Implicit (ADI) method for handling multi-dimensional data efficiently [10].

The Euler method is used to solve the Navier-Stokes equations by advancing in time with small time steps. The method is explicit, meaning that the solution at the next time step depends directly on the solution at the current step. To maintain numerical stability, the time step Δt is kept small. The Alternating Direction Implicit (ADI) method is a more stable method for solving multi-dimensional data problems. The ADI method is implicit, meaning the equations

Scientific Journal of Astana IT University
ISSN (P): 2707-9031 ISSN (E): 2707-904X
VOLUME 19, SEPTEMBER 2024

for the next time step depend on a system of equations that need to be solved iteratively. ADI allows larger time steps while maintaining numerical stability, making it computationally more efficient for complex geometries. In a typical CFD simulation of blood flow through arteries, use Euler method to simulate flow over multiple time steps, and the ADI method to efficiently handle the complex geometry of the arterial network. This combination helps in predicting locations of high shear stress that could potentially lead to artery diseases. As described in step B in Fig. 2, the Jacobi iterative method was used here to calculate the pressure P.

After solving the first stage of calculating the variables u^* and v^*, the next task is to find the pressure for its further differentiation, since this is required to find the final value of u^{n+1} and v^{n+1} (the next iteration in time).

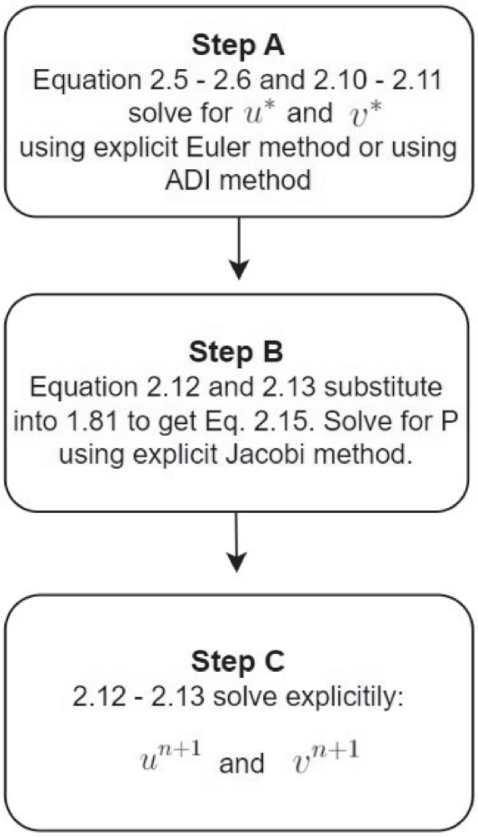

Figure 2. Structure of the SIMPLE method

Discussion

Case study

In the study of hemodynamics, particularly in modeling and simulation of blood flow through arteries, different types of boundary conditions are considered to represent the various physiological scenarios [11]. A commonly examined condition is pulsatile flow from Fig. 4, where blood flow varies in a rhythmic manner akin to the heartbeat [12]. However, another valuable approach for certain studies is to consider a parabolic constant inlet flow condition shown in Fig. 3. This simplification assumes that the blood enters a segment of the vascular system at a constant rate and with a parabolic velocity profile across the vessel cross-section.

The constant input flow parabolic model is useful for studying the impact of vessel geometry, stenosis, or the development of atherosclerotic plaques without the added complexity of pulsating flow [13]. However, this approach may not capture the full dynamics of blood flow under physiological conditions where the influence of the heartbeat is significant.

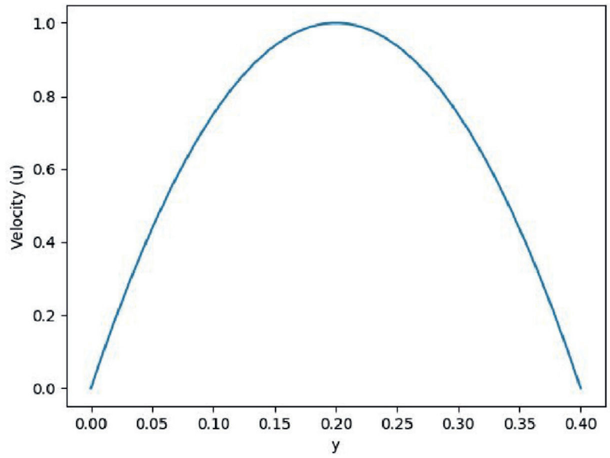

Figure 3. Input values of u along the y axis at $x-0$ at a parabolic constant flow

This pulsatile function of blood flow is vital for realistic simulations of circulatory system dynamics [14]. It affects everything from shear stress on arterial walls, which has implications for the development of diseases like atherosclerosis, to the efficient delivery of nutrients and oxygen to tissues [3].

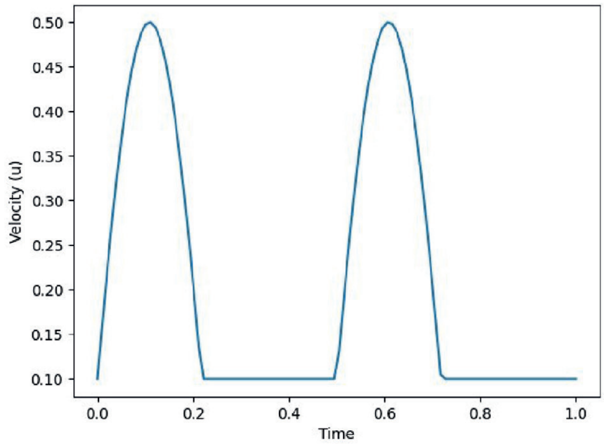

Figure 4. Pulsatile flow profile approximation used as a boundary condition

Results

Following individual differences, a comparative analysis of the methods is presented, assessing their computational efficiency, accuracy, and applicability in various fluid dynamics scenarios.

Blood circulates at a laminar pace. Blood moves more quickly in the blood vessel's center than it does in the vicinity of the vessel wall [15]. This means that the liquid that flows adjacent to the vessel wall is very sedentary, and the liquid that flows after it moves farther away. A fraction of the fluid at the middle of a blood channel travels a great distance, in contrast to all of this [16].

The code considered various conditions of blood vessels for simulation purposes. It provided options to simulate scenarios using Euler's method and the ADI method, with or without pulsation and with or without the presence of plaque. Each simulation scenario, irrespective of the method or conditions, was set to run for a total of 4000 iterations. This comprehensive setup allowed for a detailed exploration of blood flow dynamics under different physiological and methodological conditions [17].

Scientific Journal of Astana IT University
ISSN (P): 2707-9031 ISSN (E): 2707-904X
VOLUME 19, SEPTEMBER 2024

In Fig. 5-6, there is a constant blood flow at a rate of 1, as well as a slowdown in blood flow due to pulsation.

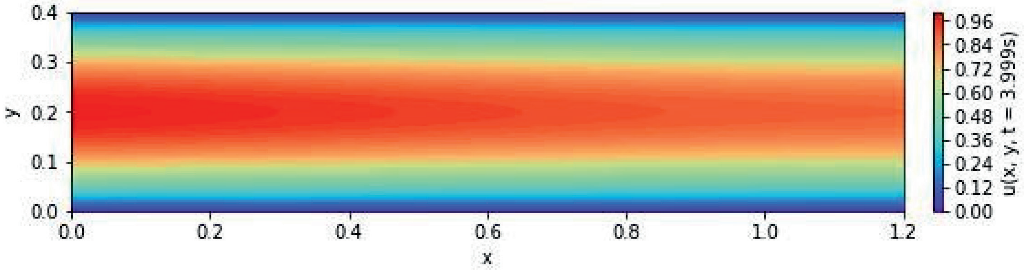

Figure 5. Euler method without pulsation without plaque at 4000 iterations

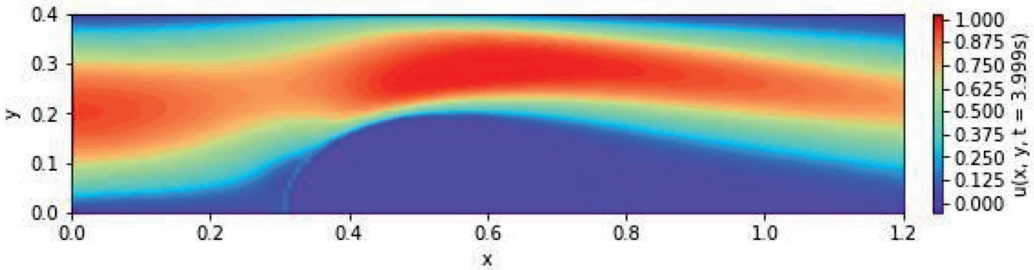

Figure 6. Euler method without pulsation with plaque at 4000 iterations

Figure 7 shows the graph resulting from calculations executed using the ADI method.

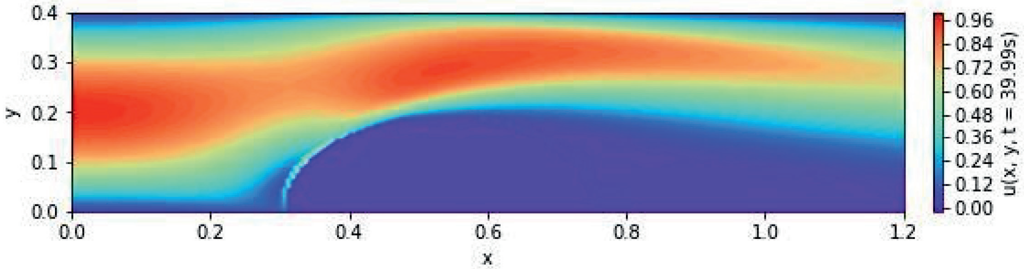

Figure 7. ADI without pulsation with plaque at 4000 iterations

On these graphs, the results from the ADI and Euler methods appear indistinguishable. However, a notable distinction exists in the time parameter, t, where the duration for ADI is significantly extended, being 10 times that of Euler's. This difference is attributed to the disparate time steps employed: Δt is set at 0.01 for ADI, while for Euler it is only 0.001, a setting enforced by Euler's stability condition. As a result of these settings, although the number of iterations is identical for both methods, to traverse the same amount of time, the Euler method requires ten times more iterations than ADI. This adjustment influences both the computational demands and the temporal resolution achievable by each method.

The graphs presented are designed to illustrate the variations in blood velocity, u, across different time intervals. the variations in blood velocity, u, across different time intervals. Four specific iterations have been selected to effectively depict the dynamics of blood flow velocity and its behavior in a segment affected by stenosis, where there is a significant constriction. It is observable that both the ADI and Euler methods indicate a heightened flow pressure at the point of constriction. Beyond this point, the flow continues forward but does not completely fill the vessel space [18]. This dynamic leads to a range of effects, including the generation of turbulence and vortices, as the stenosis impedes the smooth progression of the flow [19]. Such conditions can escalate to critical health issues, potentially resulting in life-threatening situations.

The Fig. 8 displayed below illustrates the same fluid flows, enhanced with the inclusion of vectors that denote the direction of the flow. This addition makes it easy to identify the locations where vortices form, as these areas are distinctly marked by changes in the flow direction indicated by the vectors. This visualization aids in a clearer understanding of the flow dynamics, especially in observing how the fluid movement deviates and swirls, creating these turbulent regions [20].

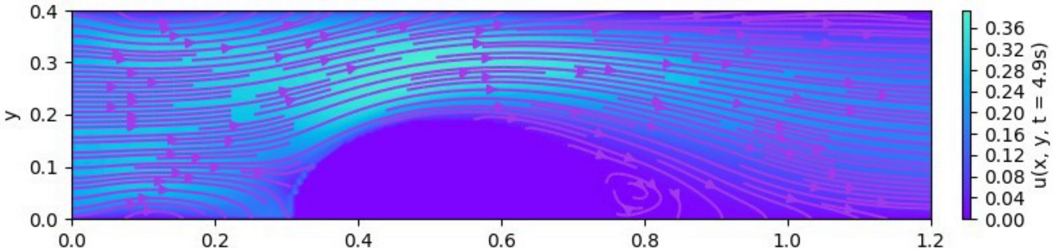

Figure 8. ADI method with pulsation with plaque and with vector arrows

Verification of calculations

After numerical solutions using Navier-Stokes, they should be compared with some verification, and one of these is the Poiseuille flow.

$$u_{poiseuille}(y) = \frac{1}{2\mu}\left(\frac{\partial P}{\partial x}\right)(y \cdot y_{max} - y^2)$$

where μ is the dynamic viscosity of the fluid, y is the distance from the no-slip boundary at one plate (where $y = 0$) to the point in question, up to $y = y_{max}$, which is the height to the opposite plate.

Comparing the Poiseuille flow profile with the results of numerical modeling in Fig. 9 helps to identify discrepancies, which allows us to assess the reliability and accuracy of the numerical method. Such comparative analyses are pivotal in refining computational tools, ensuring they provide reliable and accurate results in various engineering and scientific applications.

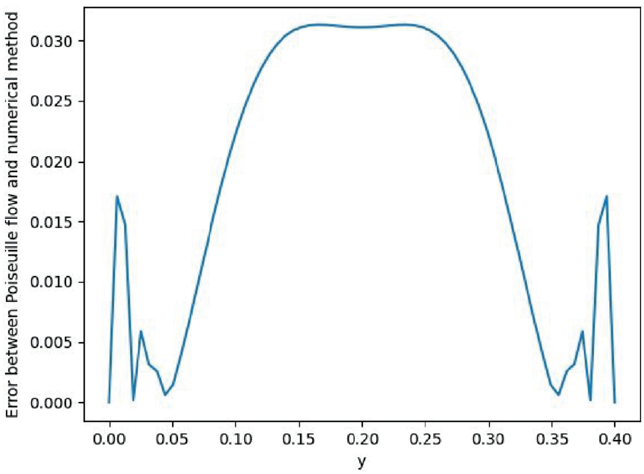

Figure 9. Error distribution between validated and numerical data.

In the above illustration the numerical solution exhibits a close resemblance to Poiseuille's flow for fluid flow between parallel plates with a maximum observed deviation of approximately 0.03. However, subtle variations are noticeable along the edges. The occurrence of these small angular discrepancies is attributed to the use of a large ε value during the pressure calculations employing the Jacobi method. In the whole, the numerical solution maintains a parabolic shape, closely mirroring the theoretical Poiseuille flow, demonstrating a high degree of accuracy. The Alternating Direction Implicit (ADI) method will be significantly more computationally efficient than the Euler method in simulating blood flow using the Navier-Stokes equations, particularly in scenarios involving complex geometries such as vessel constrictions caused by plaque. The ADI method will reduce computational time by up to 90% compared to the Euler method, due to its ability to use larger time steps while maintaining numerical stability.

Conclusion

This work highlights the growing interest in employing mathematical and computer models to examine blood flow dynamics within the cardiovascular system. These models allow for a vast array of numerical experiments to be conducted safely, without posing any risks to human subjects' well-being. Research has extensively explored how blood moves through arteries and vessels under various conditions.

This article compares two methods of solving the Navier-Stokes equation: the Euler method and the Alternating Direction Implicit (ADI) method. The main focus of the comparison is on their efficiency, accuracy and computational requirements in order to determine the most reliable and practical solution for modeling cardiovascular dynamics. Mathematical and computer models serve as potent tools for comprehending and dissecting complex physiological systems like the cardiovascular system. As technology evolves, these models will allow for more accurate modeling. This work simulates blood flow in a variety of conditions, including vessels with and without stenosis, demonstrating their flexibility and value in analyzing various pathophysiological conditions and contributing to cardiovascular research and treatment strategies.

This study is part of a broader scientific project aimed at developing a virtual reality (VR) surgical simulator for cardiovascular procedures. The primary objective of this study is to create a computational framework that can accurately simulate blood flow dynamics within the VR environment. Given that the simulator is intended for training and pre-operative planning purposes, the focus is on providing computationally efficient and educationally realistic representation of blood flow rather than on precise physiological accuracy that would require experimental validation. The VR simulator will enable surgeons to visualize how blood flow interacts with vascular conditions such as stenosis or bifurcation, offering a platform to practice decision-making and procedural techniques. While experimental data could enhance the clinical applicability of a fully validated simulation model, the goal of the VR simulator is to create a practical, interactive tool for surgeons rather than a high-fidelity research model. The simplifications made in this study—such as using Newtonian fluid assumptions and rigid vessel walls—are sufficient to meet the needs of a real-time interactive training system where computational speed and ease of use are prioritized over exact physiological detail.

By focusing on the computational performance and practicality for VR simulations, we ensure that the model meets the objectives of the project without the need for detailed experimental validation at this stage.

Acknowledgement

These studies are supported by the Science Committee of the Ministry of Science and Higher Education of the Republic of Kazakhstan (grant number AP14871641).

References

[1] Gupta, S. (2023). *3d bifurcating artery*. Retrieved from Skill-Lync online website: https://skill-lync.com/student-projects/3d-bifurcating-artery.

[2] Carvalho, V., Pinho, D., Lima, R. A., Teixeira, J. C., & Teixeira, S. (2021). *Blood flow modeling in coronary arteries: A review*. Fluids, 6(2), 53.

[3] Sinnott, M., Cleary, P. W., & Prakash, M. (2006). *An investigation of pulsatile blood flow in a bifurcation artery using a grid-free method*. In Proceedings of the Fifth International Conference on CFD in the Process Industries (pp. 1–6).

[4] Quanyu, W., Xiaojie, L., Lingjiao, P., Weige, T., & Chunqi, Q. (2017). *Simulation analysis of blood flow in arteries of the human arm*. Biomedical Engineering: Applications, Basis and Communications, 29(04), 1750031.

[5] Liu, G.T., Wang, X. J., Ai, B. Q., & Liu, L.G. (2004). *Numerical study of pulsating flow through a tapered artery with stenosis*. Chinese Journal of Physics, 42(4), 401–409.

[6] Shaikh, F., Shaikh, A. A., Hincal, E., & Qureshi, S. (2023). *Comparative analysis of numerical simulations of blood flow through the segment of an artery in the presence of stenosis*. Journal of Applied Mathematics and Computational Mechanics, 22(2).

[7] Anderson J,D. (1995). *Computational fluid dynamics*. McGraw-Hill.

[8] Gresho, P. M., & Sani, R. L. (1987). *On pressure boundary conditions for the incompressible navier-stokes equations*. International Journal for Numerical Methods in Fluids, 7(10), 1111–1145.

[9] Latimer, B. R., & Pollard, A. (1985). *Comparison of pressure-velocity coupling solution algorithms*. Numerical Heat Transfer, 8(6), 635–652.

[10] Coirier, W. J. (1994). *An adaptively-refined, Cartesian, cell-based scheme for the Euler and Navier-Stokes equations (Doctoral dissertation)*. University of Michigan.

[11] Alipova, B., Tokmukhamedova, F., Bulatov, Z., Daineko, Y., Ipalakova, M., & Tsoy, D. (2023). *Angioplasty virtual reality simulator: Modeling of blood flow in an artery*. *In* Proceedings of the 12th IEEE International Conference on Intelligent Data Acquisition and Advanced Computing Systems: Technology and Applications (IDAACS) (pp. 650-654). IEEE. https://doi.org/10.1109/IDAACS58523.2023.10348793.

[12] Kabir, M.A., Alam, M.F., & Uddin, M.A. (2021). *Numerical simulation of pulsatile blood flow: a study with normal artery, and arteries with single and multiple stenosis*. Journal of Engineering and Applied Science, 68, 1–15.

[13] Piskin, S., & Celebi, M.S. (2013). *Analysis of the effects of different pulsatile inlet profiles on the hemodynamical properties of blood flow in patient specific carotid artery with stenosis*. Computers in Biology and Medicine, 43(6), 717–728.

[14] Kamshilin, A.A., Teplov, V., Nippolainen, E., Miridonov, S., & Giniatullin, R. (2013). *Variability of microcirculation detected by blood pulsation imaging*. PLoS ONE, 8(2), e57117.

[15] Alipova, B., Daineko, Y., Rysbek, N., Tokmukhamedova, F., & Bulatov, Z. (2024). *Computer simulation of the human cardiovascular system in the Power BI software environment*. In CEUR Workshop Proceedings, 3680.

[16] Zakaria, M.S., et al. (2018). *Computational fluid dynamics study of blood flow in aorta using OpenFOAM*. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, 43(1), 81-89.

[17] Senol, P., & Serdar, C.M. (2013). *Analysis of the effects of different pulsatile inlet profiles on the thermodynamic properties of blood flow in patient specific carotid artery with stenosis*. Computers in Biology and Medicine, 43(6), 717-728.

[18] Jeong, Y., & Kang, I.S. (1995). *Pulsatile flows in stenosed blood vessels*. Korean Journal of Chemical Engineering, 12(4), 540-550.

[19] Alipova, B., Daineko, Y., Tokmukhamedova, F., Ipalakova, M., Bolotov, Zh., & Tsoy, D. (2023). *Blood flow description using math model and its visualization in Unity 3D engine as a part of medical educational simulator*. In EDULEARN23 Proceedings (pp. 5971-5975). IATED.

[20] Alpar, S., Faizulin, R., Tokmukhamedova, F., & Daineko, Y. (2024). *Applications of symmetry-enhanced physics-informed neural networks in high-pressure gas flow simulations in pipelines*. Symmetry, 16, 538. https://doi.org/10.3390/sym16050538.