talenta PUBLISHER

Journal of Research in Mathematics Trends and Technology
Journal homepage: https://talenta.usu.ac.id/jormtt

Computation Analysis of Flow in a Round Pipe with Navier-Stokes Equations

T.J. Marpaung*[1], Asima Manurung[2]

[1]*Programs of Statistics, Faculty of Vocational, Universitas Sumatera Utara, Medan, Indonesia*
[2]*Programs of Information Technology, Faculty of Vocational, Universitas Sumatera Utara, Medan, Indonesia*
*Corresponding Author: tj.marpaung@usu.ac.id

ARTICLE INFO

Article history:
Received: 01 January 2022
Revised: 02 February 2022
Accepted: 29 March 2022
Available online: 30 March 2022

E-ISSN: 2656-1514
P-ISSN: -

How to cite:
Marpaung, T.J and Manurung Asima, "Computation Analysis of Flow in a Round Pipe with Navier-Stokes Equations," International Journal of Engineering Research and, vol. V5, no. 11, Nov. 2016, doi: 10.32734/jormtt.v4i1. 15827

http://doi.org/10.32734/jormtt.v4i1.15827

ABSTRACT

This research analyzes the flow in a round pipe using the Navier-Stokes equations with the aim of understanding the characteristics of flow and friction within the system. The Navier-Stokes equations are employed to describe the movement of fluid within the round pipe, taking into account the effects of viscosity and pressure on the fluid flow. Additionally, friction within the round pipe is analyzed as a consequence of the fluid flow, with the consideration of friction coefficients to depict the magnitude of frictional forces exerted on the pipe walls. Furthermore, the researchers demonstrate that friction coefficients increase with higher flow velocities and fluid viscosities. Simulation results indicate that laminar flow is the dominant condition within the round pipe under investigation. In laminar flow conditions, the boundary layers exhibit greater organization and the fluid flow is more stable. However, at higher flow velocities, a transition from laminar to turbulent flow can occur. The computational analysis presented in this study utilizes the Computational Fluid Dynamics (CFD) software COMSOL Multiphysics. This software employs robust numerical algorithms to efficiently solve the continuity, momentum, and Navier-Stokes equations. The program provides information regarding velocity profiles, pressure distributions, and other flow parameters along the round pipe. The simulation results are obtained for varying water velocities of 0.001 m/s, 0.01 m/s, 0.1 m/s, and 1 m/s. By integrating discretization methods, the continuity equation, momentum equation, Navier-Stokes equations in component form, and energy loss calculations, this study offers profound insights into flow within round pipes and its characteristics. This research provides valuable insights into flow in round pipes, the effects of friction, and the challenges in achieving convergence solutions at high water velocities.

Keyword: Navier-Stokes, Round Pipe, COMSOL Multiphysics, Laminar Flow, Friction

1. Introduction

The analysis of flow in a round pipe is of great interest due to its relevance in various practical scenarios. To develop a more comprehensive comprehension of the intricate dynamics involved in fluid movement, researchers and engineers frequently utilize computational analysis. Therefore, there is a need for specific research that focuses on understanding the factors influencing fluid flow in pipes [1]. The focus of this study is to examine the properties of fluid flow that is incompressible and occurs within an extended, horizontal segment of a round pipe. The analysis will be based on the widely used Navier-Stokes equations, which provide a comprehensive mathematical framework for describing fluid flow [2]. By using computational techniques, our aim is to model and understand the intricate flow phenomena occurring within the pipe.

One crucial aspect of studying fluid flow is determining the velocity profile and its variation along the pipe. While the fluid moves within the pipe, the interaction between the fluid and the inner wall generates frictional forces, causing shear stress on the inner surface. As the fluid progresses, the velocity profile undergoes changes, ultimately reaching a state known as fully developed flow, wherein the velocity remains consistent throughout the length of the pipe [3]. Our objective is to explore the connection between shear stress and different parameters that govern the dynamics of fluid flow.

Additionally, we consider the influence of important factors such as fluid density and viscosity on flow behavior. These parameters play a significant role in determining the resistance encountered by the fluid as it passes through the pipe, requiring a pressure gradient to overcome frictional forces [4]. By developing nondimensional relationships between shear stress and other key parameters, valuable insights can be obtained regarding the fundamental characteristics of flow in a round pipe. Through computational analysis using the COMSOL Multiphysics program, this research aims to contribute to a broader understanding of fluid dynamics in pipe flow and provide useful insights for engineering applications [5]. By employing advanced numerical techniques and the Navier-Stokes equations, we can simulate and comprehensively analyze flow behavior, enabling us to optimize pipe designs, improve system efficiency, and enhance the overall performance of fluid-based systems.

2. Materials and Methods

2.1. Navier-Stokes Equations for Round Pipe

The method used in this study involves the use of the continuity equation, momentum equation, and Navier-Stokes equations in component form to describe the changes in velocity and fluid pressure along the pipe. The continuity equation is used to ensure that the inflow and outflow of fluid from the pipe remain balanced. The momentum equation and Navier-Stokes equations are used to depict the balance of forces in the fluid flow and account for viscosity [6]. To analyze fluid flow in a round pipe, the law of conservation of mass is employed, which states that the mass of fluid must remain constant within a closed system. In the context of fluid flow in a round pipe, the continuity equation is utilized to understand how the mass of fluid changes along the pipe. This continuity equation will be used to model and analyze fluid flow in a round pipe.

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho v) = 0, \tag{1}$$

where ρ is the density of the fluid and v is the velocity vector of the flow [6]. Next, the momentum equation will be explained, which states that the change in momentum within a fluid flow must be proportional to the forces acting on the fluid. The momentum equation allows us to analyze and predict the velocity, direction, and distribution of fluid momentum within a pipe. The momentum equation is given by:

$$\rho\left(\frac{\partial v}{\partial t} + v\,.\nabla v\right) = -\nabla p + \mu \nabla^2 v + f, \tag{2}$$

This equation describes the law of conservation of momentum in fluid flow, where p is the fluid pressure, μ is the dynamic viscosity of the fluid, and f is the external force acting on the fluid [6]. The continuity equation states that the mass flow rate in the pipe must remain constant. In component form, the continuity equation can be written as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0, \tag{3}$$

In the given equations, ρ represents the density of the fluid, t denotes time, u, v, and w symbolize the velocity components in the x, y, and z directions correspondingly, while x, y, and z represent the spatial coordinates. The momentum equations (2) in component form state that the change in fluid velocity along the pipe is influenced by the pressure gradient, inertia forces, and viscosity forces. These equations can be written as:

$$\begin{aligned}
&\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = -\frac{\partial p}{\partial x} + \rho f x + \partial/\partial x\left(\mu\left(\frac{\partial u}{\partial x} + \frac{\partial u}{\partial x} + \frac{\partial w}{\partial z}\right)\right)\\
&\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} + \frac{\partial(\rho vw)}{\partial z} = -\frac{\partial p}{\partial y} + \rho f y + \partial/\partial y\left(\mu\left(\frac{\partial v}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right)\right)\\
&\frac{\partial(\rho w)}{\partial t} + \frac{\partial(\rho uw)}{\partial x} + \frac{\partial(\rho vw)}{\partial y} + \frac{\partial(\rho w^2)}{\partial z} = -\frac{\partial p}{\partial z} + \rho f z + \partial/\partial z\left(\mu\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right)\right),
\end{aligned} \tag{4}$$

where p is the pressure, μ is the dynamic viscosity, and fx, fy, and fz are the external forces in the directions of x, y, and z correspondingly[7]. The Navier-Stokes equations in component form are a combination of the continuity equation and the momentum equations. These equations comprehensively describe the changes in fluid velocity and pressure along the pipe. Furthermore, researchers utilized appropriate boundary conditions for the flow problem in a round pipe. They applied initial velocity and pressure at the pipe inlet and implemented a zero-pressure gradient at the pipe outlet. By employing these boundary conditions, researchers were able to model realistic flow in the pipe [8]. The researchers also took into account the energy loss in the fluid flow inside the pipe. Energy loss can occur due to fluid friction with the pipe walls, changes in flow direction, or other resistances. By considering the energy loss equation, they can describe how pressure changes along the round pipe. This allows the researchers to understand radial pressure variations and study the factors that influence them, such as flow velocity, fluid viscosity, and pipe geometry. This equation describes the law of energy conservation of flow in a fluid in a round pipe.

$$\rho\left(\frac{\partial e}{\partial t}+v\left(\frac{\partial e}{\partial r}\right)\right)=-\left(\frac{\partial}{\partial r}\right)(pv)+\left(\frac{1}{r}\right)\left(\frac{\partial}{\partial r}\right)\left(r\left(\frac{\partial q}{\partial r}\right)\right)+\left(\frac{1}{r}\right)\left(\frac{\partial q}{\partial r}\right), \tag{5}$$

where e is the specific energy, q is the heat transfer rate, and v is the radial velocity of the flow. Furthermore, we can describe the law of conservation of energy in fluid flow [9].

$$\frac{dP}{dt}=-\rho\,\nabla\cdot v+\nabla\cdot(\mu\,\nabla v)+f\cdot v, \tag{6}$$

where dp/dt is the change in pressure over time, $\nabla \cdot v$ is the divergence of flow velocity, $\nabla \cdot (\mu\,\nabla\, v)$ is the divergence of the viscosity gradient, and $f \cdot v$ is the dot product of the external force and the flow velocity vector.

2.2. Friction in a Round Pipe

Friction within a pipe pertains to the hindrance experienced by a fluid while traversing the pipe, resulting from the interaction between the fluid and the inner surface of the pipe. This frictional force is responsible for the dissipation of energy and the conversion of mechanical energy into heat. Understanding and quantifying friction in a pipe is essential for various engineering applications involving fluid flow, such as designing pipelines, optimizing system efficiency, and calculating pressure drops [10]. Frictional forces occur as a result of shear stress at the interface between the fluid and the solid, causing the fluid particles in contact with the pipe wall to encounter a resistive drag force. This shear stress, denoted as τ_w, acts tangentially to the surface of the pipe. It is directly related to the fluid's viscosity, which signifies the internal resistance of the fluid to flow, as well as the velocity gradient across the pipe's cross-section. The flow eventually becomes hydrodynamically fully developed, resulting in a constant velocity profile along the pipe. Due to friction with the pipe wall, a constant shear stress τw is present on the inside wall. The pressure is the only parameter that varies linearly along the pipe to overcome friction the fluid through. The objective is to establish a dimensionless correlation between shear stress τw and other parameters involved in the problem.

The analysis focuses on deriving a nondimensional relationship for friction in a pipe by employing the iterative technique of repeating variables in a systematic manner. Assumptions include hydrodynamically fully developed flow, incompressible fluid, and no other significant parameters in the problem. Six variables and constants are considered: shear stress (τ_w), average velocity (V), fluid viscosity (μ), fluid density (ρ), pipe diameter (D), and average roughness height (ϵ) [11]. The fundamental dimensions of each parameter are provided, with shear stress sharing the same dimensions as pressure. By setting j equal to 3 (depicting the core dimensions of mass, length, and time), the expected number of nondimensional parameters ($\Pi's$) is determined as 3. Three repeating parameters are chosen: V, D, and ρ. The dependent Π (Π_1) is derived as $\Pi_1 = \tau_w/(\rho V^2)$, which is modified to represent the Darcy friction factor. Two independents $\Pi's$, related to fluid viscosity and roughness ratio, are also generated. The dependent Π is generated

$$\Pi_1=\tau_w V^{a_1}D^{b_1}\rho^{c_1}\;\rightarrow\;\{\Pi_1\}=\{(m^1L^{-1}t^{-2})(L^1t^{-1})^{a_1}(L^1)^{b_1}(m^1L^{-3})^{c_1}\} \tag{7}$$

from which $a_1=-2, b_1=0$, and $c_1=-1$, and thus the dependent Π is

$$\Pi_1 = \frac{\tau_w}{\rho V^2} \tag{8}$$

The nondimensional parameter that closely resembles this Π_1 is the Darcy friction factor, which is defined with a numerator factor of 8 (Figure 1) [12].

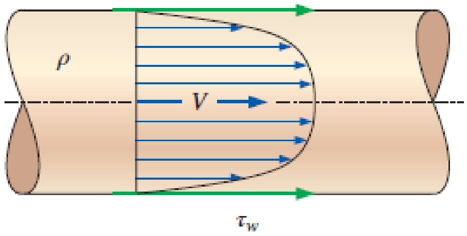


Figure 1. The Darcy friction factor is widely used in pipe flows

Modified Π_1

$$\Pi_{1,modified} = \frac{8\tau_w}{\rho V^2} = Darcy\ friction\ factor\ = f \tag{9}$$

Likewise, two independents Π's are derived, and the specific details of these derivations are left for you to explore independently [12]

$$\begin{aligned} \Pi_2 &= \mu V^{a_2} D^{b_2} \rho^{c_2} \quad \longrightarrow \quad \Pi_2 = \frac{\rho V D}{\mu} = \ Reynolds\ number\ =\ Re \\ \Pi_3 &= \epsilon V^{a_3} D^{b_3} \rho^{3} \quad \longrightarrow \quad \Pi_3 = \frac{\epsilon}{D} = \ Roughness\ ratio \end{aligned} \tag{10}$$

The final functional relationship is written as,

$$f = \frac{8\tau_w}{\rho V^2} = f\left(Re, \frac{\epsilon}{D}\right), \tag{11}$$

where Re represents the Reynolds number and ϵ/D is the roughness ratio. This correlation is applicable to both laminar and turbulent fully developed pipe flow conditions, with the roughness ratio being more significant in turbulent flow. Matching the roughness ratio ϵ/D ensures geometric similarity between pipes [13]. The analysis highlights the connection between geometric similarity and dimensional analysis, emphasizing the importance of matching ϵ/D in maintaining similarity between different pipe systems.

2.3. Materials and Programs

To illustrate the first step of defining the problem and establishing objectives in studying frictional forces in water flow inside a pipe. As an initial step in building a pipe flow model, a database of round pipe geometries has been designed. Numerical simulations of the water flow inside the pipe will be conducted using the COMSOL Multiphysics program. The water velocity is determined in the COMSOL Multiphysics program, while the friction and flow inside the pipe come from the results of the COMSOL Multiphysics program to analyze accurate interactions with water flowing in a round pipe [14]. The fully developed flow of a Newtonian fluid in a round pipe, also known as Poiseuille flow [2]. The flow is steady, laminar, and incompressible. The fluid flows through an infinitely long round pipe of diameter D or radius $R = D/2$. The pressure gradient $\partial P/\partial x$ is applied in the x-direction, causing the fluid to move from a higher-pressure region to a lower-pressure region. The following data can be used in the modeling of water flow in round pipe at COMSOL Multiphysics:

1. Geometry:
   - Dimensions: Height = 4 centimeters, Width = 40 centimeters.
2. Material properties:

- Dynamic viscosity of water (μ) = $\eta_{liquid_1}\left(T\left[\frac{1}{K}\right]\right)[Pa * s]$
- Density (ρ) = $\rho_{liquid_2}\left(T\left[\frac{1}{K}\right]\right)\left[\frac{kg}{m^3}\right]$
- Water flow velocity (u_0) = 0,001 m/s; 0,01 m/s; 0,1 m/s; and 1 m/s

3. Numerical parameters:
    - Initial damping factor = 0.01
    - Maximum number of iterations = 1000
    - Tolerance factor = 0.01

This data will be used in COMSOL Multiphysics to build a mathematical model, inputting these parameters and conditions, and perform numerical simulations that depict the flow of water inside a round pipe. As a result, the created model will resemble the one shown in this image in COMSOL Multiphysics.

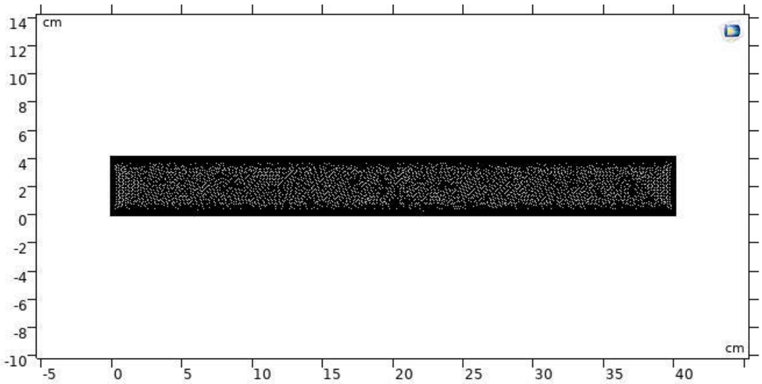


Figure 2. Round Pipe Structure

The "Build All Mesh" results in COMSOL Multiphysics provide information about the generated mesh. Number of vertex elements: There are 4 vertex elements in the mesh. Vertex elements represent the nodes or vertices in the mesh where the solution is computed. Number of boundary elements: The mesh consists of 616 boundary elements. Boundary elements define the surface or boundary of the geometry where the simulation takes place. Number of elements: The total number of elements in the mesh is 11,388. This includes both volume elements and boundary elements. Volume elements represent the elements within the domain or volume of the geometry. Minimum element quality: The minimum element quality in the mesh is 0.2406. The element quality indicates how well the mesh elements conform to geometric requirements and modeling accuracy. A higher value indicates better quality elements. These results provide insights into the complexity and quality of the generated mesh. The number of elements and boundary elements gives an indication of the overall mesh resolution. The minimum element quality serves as a measure of how well the mesh meets the geometric requirements. It is important to have a sufficient number of elements and high-quality elements to ensure accurate simulation results, although it may also increase computational time.

3. Results and Discussion

3.1. Analysis Laminar Pipe Flow

The result of water flowing in a round pipe, known as a Poiseuille flow. The fluid motion remains constant, smooth, and unaffected by compression. The fluid flows through a circular pipe with an infinite length, having a diameter D or a radius R equal to half of the diameter ($R = D/2$). A pressure difference $\partial P/\partial x$ is applied along the x-axis, causing the fluid to move from an area of high pressure to an area of low pressure [15].

$$Applied\ pressure\ gradient:\ \frac{\partial P}{\partial x} = \frac{P_2 - P_1}{x_2 - x_1} = constant \quad (12)$$

To achieve this, several assumptions and boundary conditions are established.

Assumptions [13]:

1. The pipe extends infinitely along the x-axis.
2. The flow remains constant, indicating that all partial derivatives with respect to time are zero.
3. The flow is fully developed and lacks any radial velocity component ($ur = 0$).
4. The fluid is incompressible, follows Newtonian behavior, and exhibits laminar flow characteristics.
5. An unchanging pressure gradient is applied along the x-axis, as described in Equation (12).
6. The velocity profile exhibits symmetry around the axis, with no swirling motion $u\theta = 0$, and all partial derivatives with respect to θ are zero.
7. The influence of gravity is disregarded.

Boundary Conditions [13]:

1. At the boundary of the pipe ($r = R$), the velocity is zero, indicating a no-slip condition: $\vec{V} = 0$.
2. At the centerline of the pipe (r = 0), the radial derivative of velocity is zero: $\partial u/\partial r = 0$.

The examination of fully developed flow in a circular pipe, commonly referred to as Poiseuille flow, entails the following procedure. We initiate the analysis by employing the incompressible continuity equation in cylindrical coordinates, which is a modified form of Equation (3).

$$\frac{\partial u}{\partial x} = 0, \tag{13}$$

First from (12), which states that u is not a function of x, implies that the velocity does not vary with the axial position inside the pipe. This observation is independent of the choice of origin and is a consequence of the fully developed nature of the flow. In other words, the flow has reached a stable and constant velocity profile along the pipe's length, regardless of its position. This behavior can be understood by considering the assumptions made in the problem. Assumption 1 states that the pipe is infinitely long, meaning there is nothing special about any specific position along the x-axis. As a result, the flow has had enough distance to fully develop and attain a constant velocity profile. Additionally, assumptions 2 and 6 state that the flow is steady and lacks any time or angular dependence. Since the flow is not changing with time or rotating around the axis (no swirl), it further supports the conclusion that the velocity is only a function of the radial distance r from the pipe's center.

$$Result\ of\ continuity\text{:}\ \ u = u(r)\ \ only \tag{14}$$

Subsequently, our objective is to simplify the axial momentum equation, which is a modified version of Equation (4), to the greatest extent possible.

$$\frac{1}{r}\frac{d}{dr}\left(r\frac{du}{dr}\right) = \frac{1}{\mu}\frac{\partial P}{\partial x} \tag{15}$$

Next the incompressible continuity equation and the axial momentum equation are stated and simplified. The continuity equation reveals that the axial velocity u does not depend on x, indicating that the velocity remains constant along the axial position inside the pipe due to the infinitely long length. Consequently, this indicates that the velocity profile has reached a fully developed state. Similarly, in the r-momentum equation, Equation (4), all terms become zero except for the pressure gradient term, which is also forced to be zero.

$$r - momentum\text{:}\ \frac{\partial P}{\partial r} = 0 \tag{16}$$

To put it differently, the variable P does not depend on r. Considering that P remains unaffected by time (assumption 2) and θ (assumption 6), it can be deduced that P is, at most, a function of x. In such cases, the differential equations are solved. By solving the continuity equation and the radial momentum equation, expressions for the axial velocity u as a function of the radius r and the pressure P as a function of x are obtained. It is important to highlight that the θ component of the Navier-Stokes equation is not applicable in this particular scenario.

$$Result\ of\ r - momentum\colon\ P = P(x)\ \ only \tag{17}$$

Consequently, in Equation (14), we replace the partial derivative operator representing the pressure gradient with the total derivative operator, given that P solely varies with x. Consequently, all terms in the θ-component of the Navier-Stokes equation (Equation (4)) become zero. These conditions facilitate the determination of the integration constants in the equation for axial velocity. By multiplying both sides by r, we perform one integration to acquire

$$r\frac{du}{dr} = \frac{r^2}{2\mu}\frac{dP}{dx} + C_1 \tag{18}$$

Here, C_1 represents a constant of integration. It is important to observe that the pressure gradient, dP/dx, remains constant in this context. By dividing both sides of Equation 17 by r, we perform a second integration and obtain the following result.

$$u = \frac{r^2}{4\mu}\frac{dP}{dx} + C_1 \ln r + C_2 \tag{19}$$

where C_2 is a second constant of integration.

To conclude, the solution is summarized as follows. By incorporating all the integration constants, we obtain the equation for axial velocity u as a function of pipe radius r and pressure gradient dP/dx. This equation enables the calculation of the axial velocity at any point within the circular pipe and provides an estimation of the viscous friction force per unit area exerted on the pipe walls, based on the relationship between the pressure gradient and fluid viscosity. Another interpretation of this boundary condition implies that the velocity, u, must have a finite value at the centerline of the pipe. This condition can only be satisfied if the constant C_1 is equal to 0 since the natural logarithm of 0 is undefined in Equation (19). We can now proceed to apply boundary condition 1 to the equation.

$$u = \frac{R^2}{4\mu}\frac{dP}{dx} + 0 + C_2 = 0, \quad \longrightarrow C_2 = \frac{R^2}{4\mu}\frac{dP}{dx} \tag{20}$$

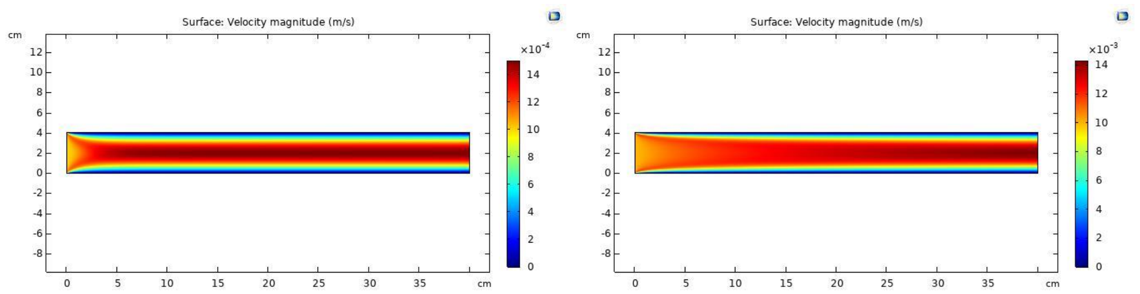
Finally, Eq. 18 becomes

$$u \ = \left(\frac{1}{4\mu}\right)\frac{dP}{dx}\ (r^2\ -\ R^2) \tag{21}$$

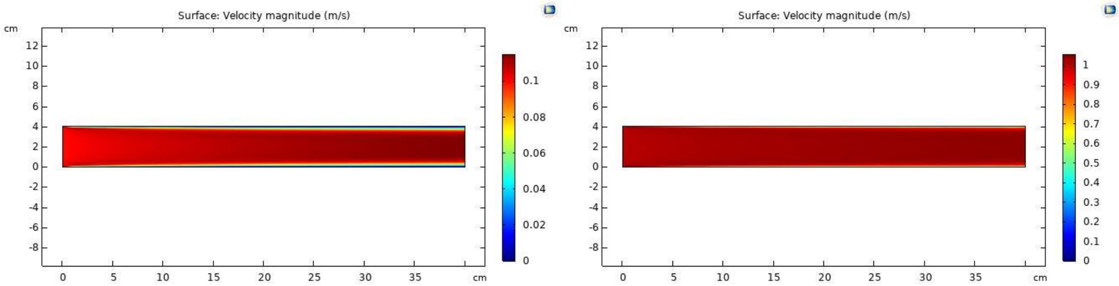
Using this equation, the axial velocity can be calculated at any point within the round pipe. Additionally, the viscous friction force per unit area acting on the pipe walls can be estimated using the relationship between the pressure gradient and fluid viscosity.

3.2. Simulation Fluid Flow of Round Pipe

Here are the results simulation from COMSOL Multiphysics depicting the flow of water inside the pipe with varying water velocities. By observing the generated simulation, we can understand how the flow changes with varying velocities and comprehend how changes in water velocity affect other variables such as pressure and frictional forces in the pipe.



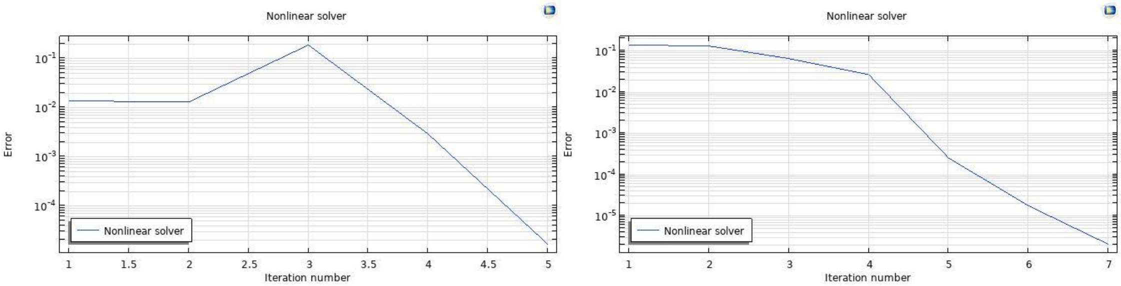
(a) (b)



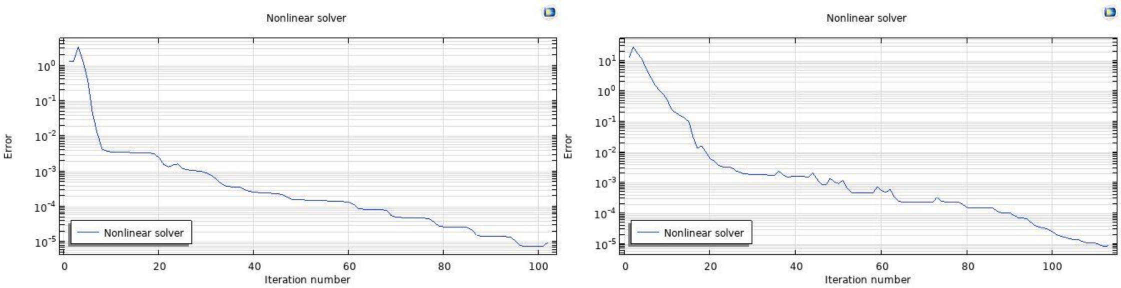
(c) (d)

Figure 3. Plot the movement of water flow in a round pipe with each speed (a) $u_0 = 0{,}001$ m/s, (b) $u_0 = 0{,}01$ m/s, (c) $u_0 = 0{,}1$ m/s, (d) $u_0 = 1$ m/s,

Furthermore, observing the convergence results at each water velocity is important to ensure that the obtained numerical solution is stable and non-divergent. A good convergence indicates that the solution iterations have reached a convergent and reliable state. By comparing the convergence results at different water velocities, it can be determined whether the model has provided consistent convergence results unaffected by changes in water velocity. This demonstrates the reliability of the model in depicting the flow of water inside the pipe, allowing for the analysis of the influence of water velocity on the observed flow phenomena. Below are the convergence results for each water velocity in COMSOL Multiphysics [16].



(a) (b)



(c) (d)

Figure 4. Plot of the convergence of the velocity of water flow in a round pipe with each velocity (a) $u_0 = 0{,}001$ m/s, (b) $u_0 = 0{,}01$ m/s, (c) $u_0 = 0{,}1$ m/s, (d) $u_0 = 1$ m/s,

3.3. Discussions from Result

In the first simulation with a velocity of 0.001 meters per second, it can be observed that a convergent solution was obtained within 5 iterations. The iterations started with an initial solution estimate of 0.014 and an initial residue of 0.032. In each iteration, the solution was updated with a specified damping factor, and the step size used for convergence also changed in each iteration. By the final iteration, the residue value had reduced to approximately 25. Next, in the second simulation with a water velocity of 0.01 meters per second, it was observed that a convergent solution was obtained within 7 iterations. The iterations started with an initial solution estimate of 0.14 and an initial residue of 0.32. By the final iteration, the residue value had reduced to approximately 3.4. Moving on to the third simulation with a water velocity of 0.1 meters per second, it was observed that a convergent solution was obtained within 102 iterations. The iterations started with an initial solution estimate of 1.4 and an initial residue of 3.2. By the final iteration, the residue value had reduced to approximately 1.3 x 10^2. Finally, in the last simulation with a water velocity of 1 meter per second, it was observed that a convergent solution was obtained within 113 iterations. The iterations started with an initial solution estimate of 14 and an initial residue of 32. By the final iteration, the residue value had reduced to approximately 7.5 x 10^2. As before, the iterations started with different initial solution estimates and initial residues. The differences in iterations occur due to variations in the water flow velocity. Higher flow velocities can impact the flow conditions and pressure distribution inside the pipe, which, in turn, affects the convergence of the solution. As the flow velocity increases, the flow becomes more complex, and the potential for more challenging convergence arises. Therefore, more iterations are required to achieve a convergent solution at higher water velocities. The disparities in water velocity can influence the convergence rate, the number of iterations needed, and the stability of the solution.

4. Conclusion

Based on the analysis of the Navier-Stokes equation for water flow in a round pipe, the axial velocity, u, is influenced by pressure factors and radius differences. By conducting simulations using the COMSOL Multiphysics program, this study obtained clear results regarding the computational analysis of water flow in a round pipe. Following the previous analysis, an examination of the velocity differences in water from equation (21) was carried out. Higher flow velocities can affect the flow conditions and pressure distribution inside the pipe, which in turn affects the convergence of the solution. The higher the flow velocity, the more complex the flow becomes, and the potential for more challenging convergence. Therefore, more iterations are required to achieve a convergent solution at higher water velocities.

5. Acknowledgement

We would like to express our gratitude to Universitas Sumatera Utara for the research funding provided through the TALENTA program under the Penelitian Dosen Muda scheme in 2020.

References

[1] Barik, R. N., Dash, G. C., & Rath, P. K. Steady laminar MHD flow of visco-elastic fluid through a porous pipe embedded in a porous medium. *Alexandria Engineering Journal*. doi: 10.1016/j.aej.2017.01.025. (2017)

[2] Blanco, P. J., Mansilla Alvarez, L. A., & Feijóo, R. A. Hybrid element-based approximation for the Navier–Stokes equations in pipe-like domains. *Computer Methods in Applied Mechanics and Engineering*, 283, 971–993. doi: 10.1016/j.cma.2014.10.036. (2015)

[3] Wu, H., Bagherzadeh, S. A., D'Orazio, A., Habibollahi, N., Karimipour, A., Goodarzi, M., & Bach, Q.-V. Present a new multi objective optimization statistical Pareto frontier method composed of artificial neural network and multi objective genetic algorithm to improve the pipe flow hydrodynamic and thermal properties such as pressure drop and heat transfer coefficient for non-Newtonian binary fluids. *Physica A: Statistical Mechanics and Its Applications*, 535, 122409. doi: 10.1016/j.physa.2019.122409. (2019)

[4] Letelier, M. F., Siginer, D. A., Almendra, D. L., & Stockle, J. S. Resonance in laminar pipe flow of non-linear viscoelastic fluids. *International Journal of Non-Linear Mechanics*. doi: 10.1016/j.ijnonlinmec.2019.03.015. (2019)

[5] Tulus, T J Marpaung, and J L Marpaung. Computational Analysis for Dam Stability Against Water Flow Pressure. *Journal of Physics: Conference Series*. 2421 012013. (2023)

[6] Coutand, D., & Shkoller. On the splash singularity for the free-surface of a Navier–Stokes fluid. *Annales de l'Institut Henri Poincaré C, Analyse Non Linéaire*. doi: 10.1016/j.anihpc.2018.06.004. (2018)

[7] Usha, R., & Sahu, K. C. Interfacial instability in pressure-driven core-annular pipe flow of a Newtonian and a Herschel-Bulkley fluid. *Journal of Non-Newtonian Fluid Mechanics*, 104144. doi: 10.1016/j.jnnfm.2019.104144. (2019)

[8] Malik, M., & Skote, M. A linear system for pipe flow stability analysis allowing for boundary condition modifications. *Computers & Fluids*, 104267. doi: 10.1016/j.compfluid.2019.104. (2019)

[9] Masna, Tulus and Sawal. Computational analysis dynamics fluid of flow in dam using COMSOL Multiphysics. Journal of Physics: Conference Series. 2421 012047. (2023)

[10] Avci, A., & Karagoz, I. A new explicit friction factor formula for laminar, transition and turbulent flows in smooth and rough pipes. *European Journal of Mechanics - B/Fluids*, 78, 182–187. doi: 10.1016/j.euromechflu. (2019)

[11] Abushammala, O., Hreiz, R., Lemaître, C., & Favre, É. Laminar flow friction factor in highly curved helical pipes: Numerical investigation, predictive correlation and experimental validation using a 3D-printed model. *Chemical Engineering Science*, 207, 1030–1039. doi: 10.1016/j.ces.2019.07.018. (2019)

[12] Marušić-Paloka, E., & Pažanin, I. Effects of boundary roughness and inertia on the fluid flow through a corrugated pipe and the formula for the Darcy–Weisbach friction coefficient. *International Journal of Engineering Science*, 152, 103293. doi: 10.1016/j.ijengsci.2020.103293. (2020)

[13] Y. A. Çengel and J. M. Cimbala. *Fluid Mechanics: Fundamentals and Applications*. McGraw-Hill Education: New York. (2018)

[14] Tulus, Suriati and Marpaung T J. Sedimentation Optimization on River Dam Flow by Using COMSOL Multiphysics. *IOP Conf. Ser. Mater. Sci. Eng*. 300. (2018)

[15] Modesti, D., & Pirozzoli, S. Direct numerical simulation of supersonic pipe flow at moderate Reynolds number. *International Journal of Heat and Fluid Flow*, 76, 100–112. doi: 10.1016/j.ijheatfluidflow. (2019)

[16] J. Tu, G. H. Yeoh, and C. Liu. *Computational Fluid Dynamics: A Practical Approach*. Burlington, MA: Elsevier, (2008).