Analysis of stenosis effect on blood pressure using moving particle semi-implicit method

H Purnama¹, Suprijadi² and N Kurniasih³

¹ Knolelge and Software Engineering Research Group, Institut Teknologi Bandung, 40132, Bandung, Indonesia
Email: purnama@informatika.org

² High Energy Theoretical Physics and Instrumentation Research Group, Institut Teknologi Bandung, 40132, Bandung, Indonesia
Email: supri@fi.itb.ac.id

³ Nuclear and Biophysics Research Group, Institut Teknologi Bandung, 40132, Bandung, Indonesia
Email: neny@fi.itb.ac.id

Abstract. Computational fluid dynamics are developed by scientists in the last few decades. Computing is used to explain the physical problem which is using a numerical method. In this paper, the phenomenon of stenosis inhibited blood flow is simulated and observed using the Moving Particle Semi-Implicit Method. The simulation was built using the Navier-Stokes equation. This simulation does not use data with quantitative units because it only aims to observe the phenomenon of blockage of stenosis in the bloodstream, so the value was obtained without units. The velocity profile of the blood particles which are located between the rigid stenosis will be fastest if the percentage of stenosis is higher. The velocity profile shows a fluctuation in the direction of the radial position = 0 when the horizontal position is 1.5 ≤ x ≤ 2.5 with the maximum velocity when the stenosis is 60% i.e 9.2 units. This shows that there are obstacles that cause the velocity profile to be higher than the blood flow without stenosis, and the greater the percentage of stenosis, the greater the fluctuations occurring in the area so that it has a higher velocity.

1. Introduction

Fluid dynamics Research for blood vessel has an important role in the development of research methods to cure diseases in a blood vessel [1]. Various diseases begin with problems in the bloodstream. One is the narrowing of the arteries caused by the development of atherosclerotic stenosis that is sticking out to the lumen and causing arterial stenosis. This narrowing is caused by the accumulation of fat such as cholesterol in an artery and will increase flow resistance and reduced blood flow to certain places due to changes in flow velocity and blood pressure. Changes in velocity and pressure in the bloodstream have implications on the functioning of the human body system because they are causing reduced blood flow to certain organs in the body. Carotid artery stenosis can cause an ischemic attack. Reduction of supply to the brain will cause cerebral strokes. Coronary arteries can cause myocardial infarction resulting in heart failure. Therefore, research on the effect of arterial stenosis on the velocity profile and blood pressure is important.
Computational fluid dynamics are used to examine how the impact of stenosis in the bloodstream. The scientists developed the method using the Navier-Stokes (NS) equation [2], a classical Newton equation that describes the physical of a fluid that does not store the quantity of fluid in the grid form, but into particles. This method is known as the particle method. In particle methods, all the quantities of physics such as mass, fluid density, pressure, and viscosity are deposited into particles. The particle method is used to construct a blood flow simulation is the Moving Particle Semi-Implicit (MPS) method. MPS method is a computational method for simulating non-compressed fluid flow. This method is a gridless method that uses particle interpolation. The fluid is considered a set of particles. MPS has been applied in the field of Nuclear Engineering [3], Environmental Hydraulics [4], and Coastal Engineering [5].

In this paper, we propose a computational study on blood vessels and the effect of variations in the ratio of the height of stationary stenosis to the height of the cross-section of blood vessels to the velocity profile and pressure of the blood between stenosis.

2. Research Methods

In the theory of water waves, fluids with homogeneous and incompressible densities affect the conservative period. Formulation:

$$\nabla . U = 0$$  (1)

The velocity vector with the wave is irrational and written as the potential velocity of $U$ regarding Equation:

$$\nabla^2 U = 0$$  (2)

Assuming it is not condensed, the equation becomes:

$$\frac{du}{dt} = -\frac{1}{\rho} \nabla P + g + \sigma \kappa \delta n$$  (3)

The essence of the MPS method is defined and divided into two parts:

$$\left( \frac{du}{dt} \right)^{\text{explicit}} = g + \sigma \kappa \delta n$$  (4)

$$\left( \frac{du}{dt} \right)^{\text{implicit}} = -\frac{1}{\rho} \nabla P$$  (5)

Explicit means to show changes in particles caused by an external force, like gravity and surface tension. Implicit means that the change of velocity is affected by the gradient of pressure.

The semi-implicit method should be transformed if it will be entered into a computer program. There are three quantities of fluid that must be transformed, density, laplacian operators to obtain velocity, and gradient operators to calculate the pressure.

The weighting function is defined in the first MPS paper as follows [6]:

$$w(r_{ij}) = \begin{cases} \frac{r_e}{r} - 1 & \text{if } r < r_e \\ 0 & \text{if } r \geq r_e \end{cases}$$  (6)

This weighting function as depicted in figure 1 is the property to be used to interact between particles. The weighting function above, between particles $i$ to particles $j$ which has a distance $r_{ij}$

The cut-off distance is the quantity of the particles to strike the neighboring particles [6].
Fluid Density is proportional to the amount of particle density or known as particle number density. The formulation is written as follows:

\[ n_i = \sum_{j \neq 1}^N w(|r_j - r_i|) \]  

(7)

In this paper, the case used is the ideal fluid, so it has incompressible properties. There are constants used as a consequence of the assumption, i.e. the constant \( n_0 \) defined as the maximum value of the particle density:

\[ n_0 = \max n_i \]  

(8)

The Laplace operator of the value \( u \) for particle \( i \) can be written like [3]:

\[ \langle \nabla^2 U \rangle_i = \frac{2d}{An_0} \sum_{j \neq i}^N ((U_j - U_i)w(|r_j - r_i|)) \]  

(9)

If we use a two-dimensional space then the value \( d \) is 2.

\[ \lambda = \frac{\sum_{i}^N |r_j - r_i|^2 w(|r_j - r_i|)}{\sum_{j \neq i}^N w(|r_j - r_i|)} \]  

(10)

The gradient operator’s equation can be written as in Equation below. \( P \) shows the minimum pressure between the particles in the cut-off distance [6].

\[ \langle \nabla P \rangle_i = \frac{d}{n_o} \sum_{j \neq i}^N \frac{P_j - P_i}{|r_j - r_i|^d} \]  

(11)

The particle density \( n \) should be constant and proportional to \( n_0 \) but has a correction of \( n_c \). This is because there is a temporal movement. Formulation [6]:

\[ n_0 = n_c + n_e \]  

(12)

From the law of conservation of mass, we can conclude that:

\[ \frac{1}{\rho} \frac{d\rho}{dt} + \nabla \cdot U = 0 \]  

(13)

\[ -\nabla \cdot U^* = \frac{n_c}{n_0 \Delta t} \]  

(14)

\[ U^* = -\frac{\lambda}{\rho} \nabla p^{k+1} \]  

(15)

\[ d\rho = n_c \] and \( \rho = n_0 \). Substituting above can be derived Poisson equations as follows:

\[ \langle \nabla^2 p^{k+1} \rangle = -\frac{\rho}{\Delta t^2} \frac{(n_c) - n_0}{n_0} \]  

(16)

Figure 2 shows the MPS algorithm flowchart. We calculate every parameter like as particle parameter \( (r, u, \rho, \mu) \) simulation parameter \( (\sigma, \beta, \alpha, t, \epsilon) \), configuration system \( (l, L_0) \), and plot parameter \( (File Name, Directory) \). Step two, we calculate the particle number density and boundary condition. And then we do the explicit step, implicit step and particle plot until a certain time.
The blood flow model is built with the boundary condition of a rigid wall and mirror using the MPS method as depicted in figure 3. The boundary conditions that are used in the initialization phase are rigid walls. The rigid wall interprets the upper and lower cross-section of the blood vessel. The blood vessel is represented by the rigid walls and dummy walls.

The system has been constructed in such a way that it resembles the Cason fluid model [1] which has the characteristics of two symmetric stenoses up and down the vessel wall and is placed in the center of the vein domain under review. There are two symmetric stenoses in the midline of the
bloodstream. This simulation is modeling without using dimension units, so the quantity that is used is not a precise and accurate physical representation, but the amount obtained in the simulation can be used for physical interpretation.

Figure 4 shows a simulation model of blood flow with silent stenosis. Part A is the channel that lies between the stenosis whereas part B is the channel before the stenosis. The flow of blood flows from the direction of \( x = 0 \) in the direction of \( x > 1 \).

3. Result and discussion

The velocity profile of the simulation is made by plotting the graph between the velocity to the radial position. The radial position 0 is defined as the middle position of the radial direction of the flow simulation. The value is positive when the direction of y-axis is positive and vice versa. The velocity profile is obtained by plotting the particle velocity at the point \( x = 2.5 \) against its radial position.

The graph of the average velocity of the radial position equal to zero can be seen as shown in Figure 5. The velocity of each time is multiplied by the time interval and divided by the total time interval. The data was obtained by calculating the average velocity from \( t = 0.3 \) time units to \( t = 0.4 \) time units. As a result, the velocity begins to fluctuate around \( x = 1.5 \) to \( x = 3.5 \), and fluctuations in velocity correspond to an escalation of the stenosis. The greater it is in the flow, the greater the fluctuation. The greater fluctuations occur in stenosis 70% and 80%. That is because the blood cross-sectional area to flow in the pipe is very small. That is causing a greater pressure and velocity than the blood model with another stenosis. In addition, the particles that pass through between the stenoses are less than the conditions when less than 70%. It is because the cross-sectional area to flow is smaller. The number of blood particles in between only about 2-4 particles is not statistically representative.

Figure 4 Blood flow model with stenosis

Figure 5 Velocity profile on whole stenosis in the \( x \) direction
The velocity profile of the whole model can be seen as shown in Figure 5. It appears that the trend profile pattern at 10%, 30%, 50% stenosis is following with the reference rate profile model. The graph has the same pattern, near the wall there is a peak of velocity and then oscillating. It is also seen that the maximum velocity that was achieved from each model shows that the greater the stenosis in the flow, the greater the velocity in the area of the stenosis.

Figure. 6 Velocity profile on whole stenosis

If there are obstacles that interfere with the flow of blood, the blood pressure will rise along with how big the size of the obstacles that interfere with the flow (Jonuarti, 2011). Figure 6. showed a similar phenomenon, that the value of the maximum pressure achieved from each percent of the stenosis was different, increasing in proportion to the increase in the percentage of plaque value compared to the height of the blood vessel. Four consecutive maximum pressures obtained during stenosis with 80% percentage of 3506 units; stenosis with a 70% percentage of 1910 units; stenosis with a percentage of 60% of 1623 units; while stenosis with 50% percentage has a pressure of 1470 units.

4. Conclusions
This simulation is a model without using dimensional units so that the amount used in it is not definite and accurate physical representation. Although uncertain, the quantities obtained in this simulation can be used for physical interpretation. From several simulation results that are obtained based on the model built in this research activity, a model of blood flow simulation with silent stenosis has been developed using the MPS method. If the percentage of stenosis is greater, the velocity profile of the blood particles located between the silent stenosis will be greater. The velocity profile fluctuates between $1.5 \leq x \leq 2.5$ with the maximum velocity when the stenosis is 60% i.e. 9.2 units.

The velocity profile shows fluctuations in the direction of the radial position zero when the horizontal position is $1.5 \leq x \leq 2.5$. This shows that in conditions $x$ there are obstacles that cause the velocity profile to be higher than the blood flow without stenosis, and the greater the percentage of stenosis, the greater the fluctuations occurring in the area so that it has a higher velocity.

Conclusions can be drawn also between the height of the stenosis and the pressure generated between the stenosis. The highest is the stenosis with the percentage of 80% produce a pressure of 3506.31 units.

References
[1] Haryanto F and Jonuarti R, editor. Analisis model fluida casson untuk aliran darah dalam stenosis arteri. Prosiding Simposium Nasional Inovasi Pembelajaran dan Sains 2011 (SNIPS 2011), August 2011.
[2] Wendt F. John. *Computational of Fluid Dynamics: An Introduction*. Springer, 1996.
[3] Nomura K, Koshizuka S, Oka Y and Obata H. (2001): *Numerical analysis of droplet breakup behavior using particle method*. Journal of Nuclear Science and Technology, Vol. 38 No. 12, 1057-1064.
[4] Shakibaenia A. and Yee-Chung J. (2009): *A weakly compressible MPS method for modeling of open-boundary free-surface flow*. International Journal for Numerical Methods In Fluids 63(10): 1208-1232. DOI: 10.1002/fld.2132.
[5] Khayyer A, Gotoh H : *Modified Moving Particle Semi-implicit method for the prediction of 2D wave impact pressure*. Coastal Engineering 56: 419-440, 2009.
[6] Koshizuka S., Oka Y.: *Moving particle semi implicit method for fragmentation of incompressible fluid*. Nuclear Science and Engineering, 123(3):421-434, 1996.
[7] C. F. Naa. *Stochastic Moving Particle Semi-Implicit For Linear Water Wave Simulation And Its Analysis Using Surface Detection Algorithm*. Master’s thesis, Institut Teknologi Bandung, 2011.