Proceedings of the 4th ITB Graduate School Conference
Innovation and Discovery for Sustainability
July 6, 2023

2D Fluid Motion Simulation Using the Smooth Particle Hydrodynamic Method

Kaisa Nooreza[1,*], Triati Dewi Kencana Wungu[2] & Fiki Taufik Akbar Sobar[2]

[1]Physics Teaching Study Program, Faculty of Mathematics and Natural Sciences, Institut Teknologi Bandung, Jalan Ganesa 10, Bandung 40132, Indonesia
[2]Department of Pyhsics, Institut Teknologi Bandung, Jalan Ganesa 10, Bandung 40132, Indonesia
[*]Email: kaisanoreza6@gmail.com

Abstract: 2D virtual fluid simulation can help in simulating fluids directly in various problems, such as fluid simulations for education, geological simulations for landslides, and so on. This study aims to simulate the fluid behavior of water in an unhindered medium and observe the fluid particle interactions that occur. This research is simulated using the Python programming language to simulate fluids and using the Wondershare Filmora X application to combine images. This research method uses the Smooth Particle Hydrodynamics method to simulate fluids in a 2-dimensional container with a total of 10,000 particles. The Smooth Particle Hydrodynamics method is used to help model fluids in particle form by deriving the direct force density field from the Navier-Stokes equation. The Navier-Stokes equation is used to find the acceleration and velocity of fluid particles by including external forces, internal forces, and gravity through the Smooth Particle Hydrodynamics method. Acceleration and velocity will be validated due to wall collisions which cause changes in particle position and particle velocity. At the time of visualization, the fluid velocity slows down over time due to damping caused by interactions between fellow particles and particles with walls. And here you can see the fluid behavior of water flowing from a high place to a lower place and the water fluid particles are shaped to follow the container.

Keywords: *fluid; smooth; Navier-Stokes; smooth particle hydrodynamics.*

1 Introduction

Fluids play an important role in everyday human life. Such as the phenomenon of landslides, water flowing in pipes, sea waves, and so on. So because of that, fluids are something that must be studied in natural science, especially in physics. Fluid material in physics is one of the materials that students find difficult to understand. So that makes it difficult for teachers to give a better picture of fluid. To overcome this problem the teacher can demonstrate fluid events in the form of simulations. The simulation can show fluid behavior. Of course, this simulation can help teachers save expenses in looking for various kinds of fluid substances, containers, and other materials that can be used to demonstrate fluid material. So that in this study fluid simulation modeling will be carried out to overcome teacher problems.

One of the approaches used to simulate fluids is a numerical method. Numerical methods will assist in solving the problem of particles as complex fluids moving with the physical parameters of the fluid, such as viscosity, density and surface tension. With this perspective, the problem of the movement of fluid particles can be solved by the Navier-Stokes equation. Solving the Navier Stokes equation with numerical methods that can be used include Finite Volume Methods (FVM), Smoothed Particle Hydrodynamics (SPH), Finite Difference Methods (FDM), and Lattice Boltzmann Methods (LBM). however in this study using SPH because it does not require a regular grid, easy to implement for fluid flow simulation,and has a higher accuracy in modeling fluids[1] .

Smoothed Particle Hydrodynamics (SPH) is a particle-based computational method that can simulate the dynamics of a continuous medium such as fluid flow [2]. The Smoothed Particle Hydrodynamics (SPH) method has been used in several studies, including "Implementation of Particle Systems Using the Smoothed Particle Hydrodynamics (SPH) Method for Lava Flow Simulation" which was studied by Hamdi [2],"2-D Tsunami Simulation with using the Smoothed Particle Hydrodynamics (SPH) method" which was studied by Neny[3], and "Particle-Based Fluid Simulation for Interactive Applications" which was studied by Muller[4]. Many studies use the SPH method because this method can simulate phenomena with complex geometries that cannot be modeled by other methods. Therefore, in this research, fluid simulation will be carried out using the Smoothed Particle Hydrodynamics (SPH) method to solve the Navier-Stokes equation.

2 Theoretical background and method

2.1 Fluid dynamics

Fluid in general is a collection of molecules arranged randomly and held together by weak cohesive forces and by forces exerted from the outside as well as from the walls of the container. If a force is applied from outside, the fluid will deform continuously due to shear stress[5]. This continuous change is influenced by the size of the force exerted from the internal and external forces of the fluid.

These internal and external forces will be a necessary part of the formulation for a mathematical basis for adjusting fluid motion simulations according to actual conditions. Fluid motion simulation in this study uses the case of isothermal fluids, the quantity that applies to the fluid velocity field (v), density field (ρ), and pressure field (P). The physical quantity will change over time (continuous). Muller, et al. writes that the change is governed by two equations, the first used is:

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

The above equation guarantees the conservation of mass. Since the number of particles is constant and each particle has a constant mass, equation (1) is fulfilled, so only the second fluid dynamics equation, namely the Navier-Stokes equation, is used for the conservation of momentum,

$$\rho \frac{\partial v}{\partial t} = -\nabla p + \rho g + \mu \nabla^2 v \tag{2}$$

If all parts of equation (2) are multiplied by V (volume) then the equation becomes,

$$m \frac{\partial v}{\partial t} = -V\nabla p + mg + V\mu\nabla^2 v \tag{3}$$

where μ is the viscosity of the fluid, m (mass), V (volume), and P (pressure). Equation (3) is on the left side "$m\frac{\partial v}{\partial t}$ = F or F = ma", This corresponds to Newton's 2nd law of force. Where as on the right side of equation (3), $-V\nabla p$ is the force by pressure, mg is the external force which comes from the gravitational field and $V\mu\nabla^2 v$ is force by viscosity. The force by pressure and force by viscosity are included in the internal force. Equation (3) if simplified becomes,

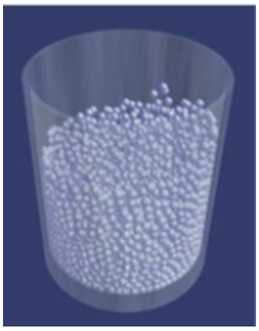
. $$F_i = F_{i\ pressure} + F_{i\ viskosity} + F_{i\ eksternal} \tag{4}$$

These external and internal forces will produce acceleration and velocity in the fluid.

$$a_i = \frac{dv_i}{dt} = \frac{F_i}{m_i}\ or\ a_i = \frac{-\nabla p}{\rho} + \mathrm{g} + \frac{\mu\nabla^2 v}{\rho} \tag{5}$$

2.2 Smooth Particle Hydrodynamics Method

The SPH method is used to solve problems related to fluids in three-dimensional and two-dimensional space. Currently the use of the SPH method has been widely used for various field modeling because of the ability of SPH to combine various complex physical aspects in the SPH formulation [6]. In SPH, fluids are modeled as small particles that behave fluidly. Each particle describes fluid properties such as mass, viscosity, position, velocity, acceleration and so on.



(a) (b)*

Figure 1 Fluid is described as a collection of particles in 2D (figure a) and 3D (figure b, *)Muller[4].

In accordance with 'Smoothed Particle Hydrodynamics', the naming of the word 'smoothed' means that some particle properties are determined by taking the average value of neighboring particles. Neighboring particles are particles located in a circle or sphere area in the two-dimensional and three-dimensional cases with a radius h. The variable h is usually called the smoothing length. Meanwhile, particles outside the radius of h are called particles that may become neighboring particles. As in the case of a particle with radius h with particles inside the radius in blue and outside the radius in black as shown in the image below,

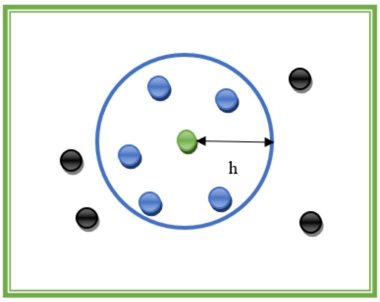


Figure 2 The contribution of neighboring particles in the area h

In the picture above, it can be seen in the two pictures how the contribution of the nearest neighbors is like on the inside of the circle, where in the circle the green particles have five closest neighbors, namely the blue particles which are in a circle with radius h (smoothing length). Meanwhile, black particles are particles that are likely to be the closest neighbours. Each particle contained in the container has its nearest neighbor, so that the parameters calculated using particle contributions will differ from one particle to another.

2.2.1 SPH Basic Equation

The basic formulation in numerical SPH is used to solve scalar quantities (A_s), of a particle such as mass (m), density (ρ), volume (V), surface area (A), and so on. These scalar quantities can be calculated using the general SPH equation in equation (6) below[6].

$$A_s(r_i) = \sum_j m_j \frac{A_j}{\rho_j} W(r_i - r_j, h) \tag{6}$$

Where r is the position vector of the particle, m is the mass of the particle, ρ is the density, h is the smoothing length, and W is the smoothing kernel. While j is the neighboring particle of particle i under review. Particles can be said to be neighbors if both are still within the scope of h. Therefore, the SPH equation above illustrates the magnitude of the quantity value of a particle determined by the contribution of its neighbors. while the derivative function for equation (6) will only be applied in the smoothing kernel, because other variables are considered constants. So the reduction is

$$\nabla A_s(r_i) = \sum_j m_j \frac{A_j}{\rho_j} \nabla W(r_i - r_j, h) \tag{7}$$

and the Laplacian,

$$\nabla^2 A_s(r_i) = \sum_j m_j \frac{A_j}{\rho_j} \nabla^2 W(r_i - r_j, h) \tag{8}$$

2.2.2 SPH for the Navier-Stokes equation

The basic SPH equation in equation (6) will be used to solve the Navier-Stokes equation. in the Navier-Stokes equation on the right-hand side has three parts, namely pressure (-∇p), viscosity ($\mu\nabla^2 v$), and external force (m.g). Pressure and viscosity will be derived using the basic SPH equation. Applying the basic SPH formula for pressure (-∇p) can be written

$$-\nabla p(r_i) = -\sum_j m_j \frac{p_j}{\rho_j} \nabla W\left(r_i - r_j, h\right) \tag{9}$$

However, the above equation is not symmetrical because only two particles interact, the gradient of the kernel at the center is zero, the pressure on the two particles is generally not the same and causes asymmetrical pressure. Then the symmetric version for equation (9) becomes[6].

$$-\nabla p(r_i) = \sum_j m_j \frac{p_i + p_j}{2\rho_j} \nabla W_{spiky}\left(r_i - r_j, h\right) \tag{10}$$

With is W_{spiky} a kernel function; and p is pressure. The pressure p in equation (10) can be calculated using a simple ideal gas

$$p_i = \kappa(\rho_i - \rho_0) \tag{11}$$

Where κ is the gas constant, ρ_i is the density of particle i, and ρ_0 is the initial density. To calculate the density of a particle i, what is necessary is to add up the mass contributions of all neighboring particles of a particle I using the summation density. So, the basic equation can be written

$$\rho_i = \sum_{\mathrm{j}} m_j\, W(r_i - r_j, h) \tag{12}$$

As for the viscosity, the same SPH basic equation will be used, so the equation becomes

$$\mu \nabla^2 v(r_i) = \mu \sum_j m_j \frac{v_j}{\rho_j} \nabla^2 W_{viskos}\left(r_i - r_j, h\right) \tag{13}$$

With the symmetrical version[5]

$$\mu \nabla^2 v(r_i) = \mu \sum_{\mathrm{j}} m_j \frac{v_j - v_i}{\rho_j} \nabla^2 W_{viskos}(r_i - r_j, h) \tag{14}$$

Where v is the velocity and μ is the viscosity of the fluid.

2.2.3 Smoothing Kernel

Smoothing Kernel or known as the kernel function is used to transform particles from a point mass into a form that spreads in space. Because this study uses a cylindrical container, the suggested kernel function is a polynomial function. According to Muller, et al, there are three suggested polynomial function-based kernel functions, namely poly 6, viscosity, and spiky. As for the functionality of the poly 6 kernel is[4],

$$W_{poly\,6}(r,h) = \begin{cases} \frac{315}{64\pi h^9}(h^2 - |r|^2)^3 & ,0 \le |r| \le h \\ 0 & ,|r| > h \end{cases} \tag{15}$$

The above kernel function is used for all calculations except those relating to viscosity and pressure. In calculations related to viscosity, the viscosity kernel will be used,

$$W_{viskosity}(r,h) = \begin{cases} r\frac{15}{2\pi h^3}\left(-\frac{|r|^3}{2h^3}+\frac{|r|^2}{h^2}+\frac{h}{2|r|}-1\right) & ,0 \le |r| \le h \\ 0 & ,|r| > h \end{cases} \tag{16}$$

And for calculations related to pressure using spiky kernel,

$$W_{spiky}(r,h) = \begin{cases} \frac{15}{\pi h^6}(h-|r|)^3 & ,0 \le |r| \le h \\ 0 & ,|r| > h \end{cases} \tag{17}$$

With $r = r_i - r_j$.

2.3 Damping

Damping is a natural event where the process of absorbing energy in a moving object occurs. As a result of absorbing energy in this moving object, it will cause the object to be obstructed and will eventually stop, as in the case of this visualization, the particle is obstructed by walls and other particles. That's why almost all objects on earth have a damping coefficient whose magnitude depends on certain factors[7].

Damping can be divided into structural damping, Viscous Damping, Coulomb damping and Negative damping[7]. Where structural damping is damping caused by internal friction of the molecules in a particular material, this friction occurs between structural members and connecting devices. Whereas Viscous Damping is damping caused by friction between objects and liquids as happened in this simulation, the magnitude of the damping coefficient depends on the type of material, but in this study the damping coefficient will use the default coefficient of the structure with 90% damping. Meanwhile, Coulomb damping is the damping caused by friction between dry solid objects. And finally, negative dumping is reverse damping which does not dampen movement but vice versa.

2.4 Visualization of Unhindered Fluid Particle Interaction

The visualization process of the unhindered fluid begins by entering the parameters into the simulation, namely the value of mass, density, smoothing length, viscosity, gas constant, acceleration due to gravity, number of particles, volume damping coefficient and time step. After entering the input parameter values, to enter into the Navier-Stokes equation to find the external and internal force values. Because the fluid is likened to a particle, it is required to find the nearest neighbor with Tree Search in radius h. The average density and pressure can be found after obtaining the nearest neighbors with the conditions j ≠ i and |r| ≥ h. After obtaining the average density value, it will be entered into equation (12) and also included is the poly 6 kernel function so that equation (12) becomes

$$\rho_i = \frac{315 m_j}{64\pi h^9} \sum_j \left(h^2 - \left|r_i - r_j\right|^2\right)^3 \tag{18}$$

When the ρ_i value is obtained, the ρ_i value will be entered into the ideal gas equation (11) to obtain the p_i value. The p_i value will be used to find the Pressure value (-∇p). after entering the p_i value and the spiky kernel function, equation (10) becomes

$$\nabla p = \left(-\left(\frac{-45\, m}{\pi h^6}\right)\right) - \frac{r_j - r_i}{(r_i - r_j)} \frac{(p_j + p_i)}{(2\rho_j)} (h - (r_i - r_j))^2 \tag{19}$$

after getting the Pressure value then proceed with finding the Viscosity value through the equation (14). By entering the viscous kernel function, the equation becomes

$$\mu \nabla^2 v = \frac{(45\mu m)}{\pi h^6} \sum_j \frac{v_j - v_i}{\rho_j} (r_i - r_j, h) \tag{20}$$

After the pressure and viscosity calculations are obtained, they will be entered into equation (5) to obtain the "$\frac{\partial v}{\partial t}$" value. The "$\frac{\partial v}{\partial t}$" value will be integrated to obtain the particle position and particle velocity values. The particle position and particle velocity will be updated every time with the Leap-frog technique. The particle position and particle velocity will be updated again due to damping. In order to understand the process of running the programming in this research, the following is an overview of the SPH flowchart of this research

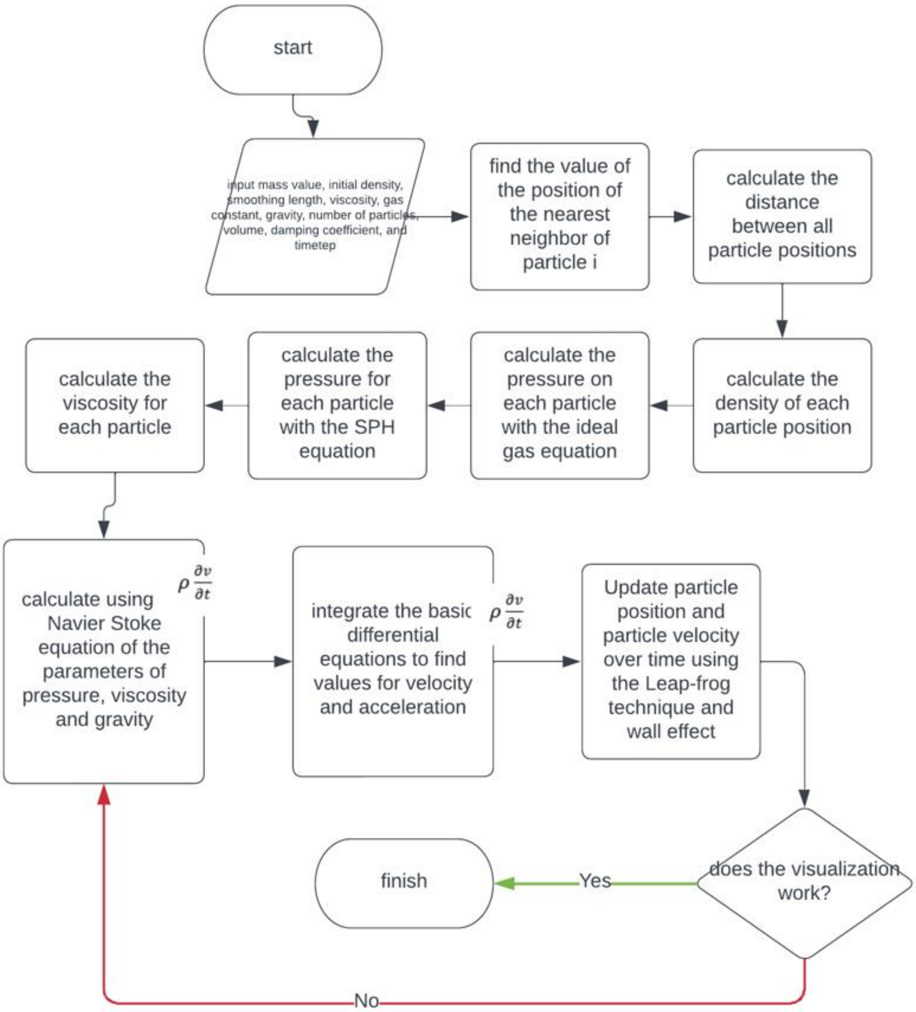


Figure 3 Flowchart SPH

2.4.1 Design Simulation

The simulation program in this study was created using the Python language using Jupyter Notebook as the coding environment. Visualization. The visualization results are in the form of images that are combined using the Wodershare Filmora X application to make an mp4 video. The computer used in this study uses a Lenovo Ideapad AMD Ryzen 5 with 8GB of RAM. This specification is able to run the simulation well with a total of 10,000 particles. The following is a fluid simulation design,

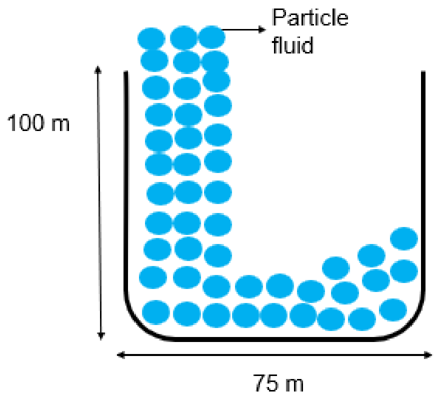


Figure 4 Design Simulation

The simulation above uses 10,000 fluid particles that are dropped as if it were rain falling with a height of y is 100 m and a width of x is 75 m with 50 particles being dropped every second.

3 Result and Discussion

3.1 Research Parameters

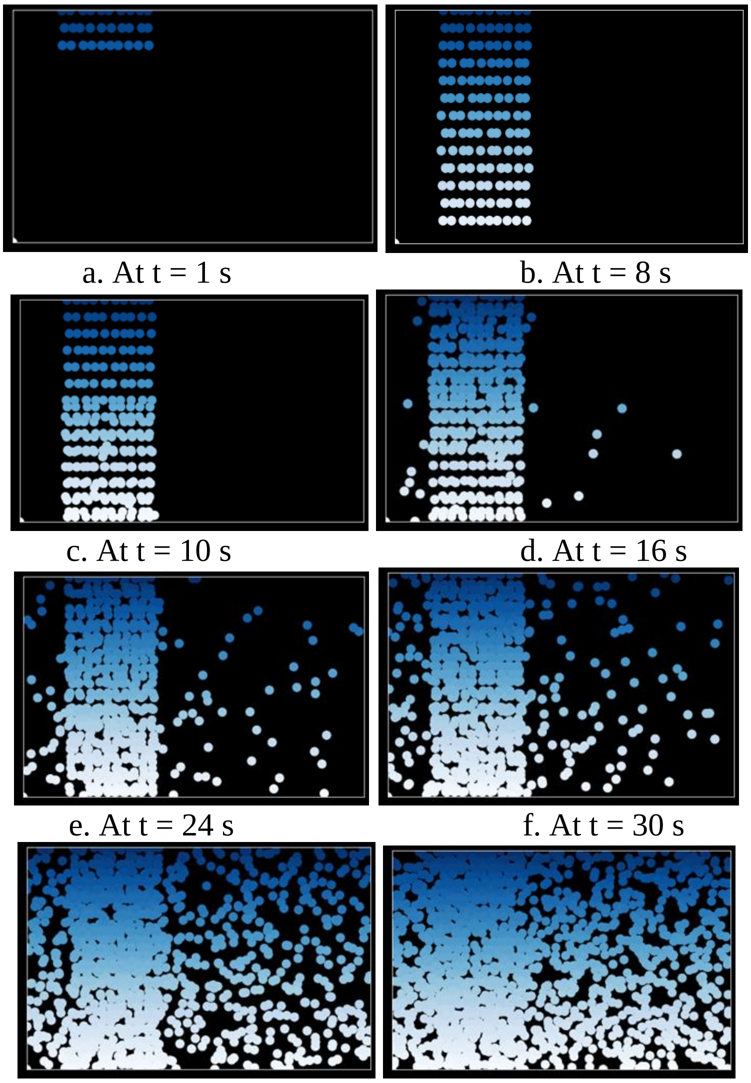
The visualization in this thesis uses parameters to simulate the fluid to be input in the fluid coding are as follows:

Table 1 physical parameters.

No	Variable	Value	Unit
1	Fluid Particle Mass (m)	0.012	Kg
2	Initial density (ρ_0)	998.29	Kg/m^3
3	Volume	m/ρ_0	m^3
4	*Smoothing length(h)*	0.021	m
5	Viscosity(μ)	3.5	Pa.s
6	Gas constant (k)	3.0	J
7	Gravity	9.81	m/s^2
8	*Timestep*	0.01	S
9	The number of particles	10.000	Partikel
10	Damping coefficient	0.9	N

3.2 Fluid Simulation Process

In this section will be shown in general the course of programming



a. At t = 1 s
b. At t = 8 s
c. At t = 10 s
d. At t = 16 s
e. At t = 24 s
f. At t = 30 s
e. At t = 60 s
f. At t = 120 s

Figure 5 SPH program visualization in general

The picture above shows the flow of fluid that is dropped from above by the force of gravity of the earth into a container to limit fluid movement. In the initial image, it can be seen that particles are dropped every second by 50 particles. after which collisions begin to occur between the fluid particles and the wall. The conditions at t = 24 s to t = 120 s can be seen that there are many collisions between the fluid particles and the wall so that the fluid velocity accelerates and the forces on the particles occur. especially on the bottom visible particles. Here it can be seen that the white particles have a greater force than the blue particles above. This continues to occur until the fluid is fully filled with 10,000 particles by adding 50 particles every second.

The results of this study show that when the particles are dropped into the container. Particles flow from a high place to a lower place to fill this empty space according to the behavior of the fluid which has the behavior of flowing from a high place to a lower place. Then when filling the empty space over time the fluid takes the shape of a container, this is the same as fluid behavior which does not refuse to change shape and follow the shape of the container. And if the dialysis is further away when the particles are dropped the particles have a faster speed due to the earth's gravitational field until they reach the walls of the container on the surface. after arriving at the bottom of the container wall, the particles will be reflected, causing their speed to slow down. Over time the speed of the particles that are below will be slower due to reflection between the particles and the wall, causing the energy of the particles to be damped continuously.

4 Conclusion

The conclusion in this study is that water particles behave like flowing from a high place to a lower place and the fluid particles are shaped like containers. As well as the speed of the fluid slows down over time due to attenuation caused by interactions between fellow particles and particles with walls. This has very well described the fluid behavior of water in a barrier-free medium and the interaction of fluid particles in actual conditions. So that the SPH method is very effective for fluid simulation. However, the SPH method will be more effective if it uses better computer speculation because the greater the number of particles in the simulation, the more effective the simulation will be, causing the computer to work more than usual.

Based on this initial research which is quite good, the researcher plans to add more features to the simulation such as Boundary, different containers, different fluids, and other cases that are useful in developing the technology. So that it can simplify solving fluid problems without using more expensive costs in actual simulations. This will be enough to help with costs and lighten the work of researchers in the actual industry.

References

[1] Liu, G.R., Liu, M.B., *Smoothed Particle Hydrodynamics – A Meshfree Particle Method*, Singapura: World Scientific Publishing, 2003.

[2] Hamdi, K., Implementasi sistem partikel menggunakan metode Smoothed Particle Hydrodynamics (SPH) untuk simulasi aliran lava., Institut Teknologi Bandung, Bandung, 2005.

[3] Minarno, N. L., Prasetyowati, S. S., & Tarwidi, D. *Simulasi Tsunami 2-d Dengan Menggunakan Metode Smoothed Particle Hydrodynamics (SPH).* Proceedings of Engineering, 2(3). 2015.

[4] Muller, M., Solenthaler, B., Keiser, R., Gross, M.,*Particle-Based Fluid-Fluid Interaction*. Zürich: Eurographics/ACM SIGGRAPH Symposium on Computer Animation. 2005.

[5] Giancoli, D., C. Fisika Edisi Ke 7-Jilid 1. Jakarta: Erlangga.2019.

[6] Muller, M., Charypar, D., dan Gross, M. Particle-Based Fluid for Interactive Application, Proceeding of 2003 ACM SIGGRAPH Symposium on Computer Animation, 154-159. 2003.

[7] Chopra, A.K., *Dynamics of structures*. New Jersey: Englewood Cliffs.1975.