Development of the smoothed particle hydrodynamics (SPH) method for modeling movement of fluid layer interaction between impermeable soil layers with boundary conditions sloping surface

Y Wicaksana*, E Bahsan¹, and R R D Rika¹

¹Civil Engineering Department, Faculty of Engineering, Universitas Indonesia, Depok, Indonesia 16424

E-mail: yudha.wicaksana@ui.ac.id

Abstract. Liquefaction usually occurs on a saturated sand deposit when the pore water pressure increases during the earthquake, and the water level will rise that could even burst out above the liquefied layer. If there is a relatively impermeable soil layer above the liquefied one, then the water could be trapped between the liquefied soil and other layers on the top. This research tries to develop a numerical model to simulate the phenomenon using the Smoothed Particle Hydrodynamics (SPH) method. SPH is a numerical method based on particle interactions that initially developed to solve the problems in astrophysics. Two sets of particles are assigned to behave like water and impermeable soil, where the thin layer of water is placed between two layers of soil. The model is built using the Fortran programming language developed in the previous research to simulate the impermeable layer's behavior by controlling the stiffness coefficient and coefficient of damping. The soil and water layers are aligned at specific angles to simulate the field's sloping surface condition. The simulation results are presented in graphical animation using Gnuplot and visually checked that the particle's movement could represent the field's actual movements.

1. Introduction

Liquefaction is a symptom of loose sand leaching that is fused with water because of the earthquake shake. In retaining the shake, the triggering force exceeds the force that is owned by the local lithology. This condition caused some of the events such as quick settlement, tilting or differential settlement, and desiccated well water replaced by non-cohesive materials. Based on the spatial planning bill, department of housing and regional infrastructure [2007], a land allows for housing when it has flat to rather steep topography with a slope number is 0-25%. Here are the criteria of the slope that consist of flat (0-8%), sloping (8-15%), rather steep (15-25%), steep (25-45%), very steep (>45%). Modeling that used numeric method is a simulation that can break any problem using a simulation process that can differentiate the water layer and soil realistically and differentiate the slope that occurred on the surface level.

A simulation-based on the Finite Element Method (FEM) hasn’t shown the impact of water flow to the surface level (Stefanova et al., 2012). The principle of pressure water that affect small pressure can be analyzed using FEM. However, the movement and the process of erosion can’t be portrayed by the FEM analysis. Meshfree Methods such as Smoothed Particle Hydrodynamics (SPH) can analyze the behavior and interaction between particles and different materials along with its movement. Besides that, it able to portray any type of different land. This method development began from Monaghan in 1992 up until now (Lydiana [2012], Syaiful Wulandari [2018]). One of the advantages of this method is the ability to observe the result of each particle both from the property values and its vision to get
representative results. Therefore, the objective of this research about the effect of a sloping surface as the numeric analysis from water-soil interaction using the Smoothed Particle Hydrodynamics (SPH) numeric approach. This modeling was done as a form of numeric analysis validation to the occurring phenomenon.

2. Research Methodology

2.1. Theoritical Review

The Smoothed Particle Hydrodynamics (SPH) method was originally developed to simulate nano symmetric in astrophysics field inside the three-dimensional open space, a modeling movement of particle-based fluid method (Lucy, 1977; Gingold and Monaghan 1977). One of the meshfree methods, Smoothed Particle Hydrodynamics (SPH) is one of the famous and effective in simulating free flow (Monaghan J., 2005), applied largely in the technical problems (Gomez-Gesteira & Dalrymple, 2004; Chen & Qiu, 2008) and soil-water dynamics problem (Bui, Fukugawa, Sako, & Ohno, 2008; Maeda & Sakai, 2010). SPH method is a meshfree method, langrangian, particle method which it characteristic has an advantage in adaptive rate of property value estimation to the randomly distributed particle (Liu & Liu, 2003). SPH method can formulate the necessary variables based on the definition in the Navier-Stokes equation. Based on the analysis of SPH method, it can be known the amount of the velocity from the simulated particles. The velocity can be gotten from the result of dividing the internal and external forces that can be defined with the Navier-Stokes equation to the mass density. The equation of velocity can be formulated into:

\[ \dot{a}_i = \frac{\sum F}{\rho} \]  

The value of velocity can be processed to know the velocity and the position from the reviewed particle. SPH method basically is the interpolation integral method that estimates the value and the derivatives of a quantity using particle as the discrete value (Vijayakumar, 2012). The interpolation method is using kernels function that very close with delta function (Kelager, 2006). The integral interpolation function in the \( A_i \) value in the \( \Omega \) can be stated below:

\[ A(r) = \int_{\Omega} A(r_j) W(r - r_j, h) \, dr_j \]  

Where \( r \in \Omega \) and \( W(r - r_j, h) \) is smoothing kernels function with the width, and core radius is \( h \) (smoothing radius). In the numeric form, the function can be written into,

\[ A(r) = \Sigma_j A_j V_j W(r - r_j, h) \]  

Where \( V_j \) is area volume from \( j \) particle, and the amount of \( A_j \) value is constant in \( V_j \) and integral from volume for each particle close to zero. The form of \( V_j \) is transformed into another equation form with,

\[ V_j = \frac{m_j}{\rho_j} \]  

With equation substitution \( V_j \), SPH interpolation integral equation can be rewritten into,

\[ A(r) = \Sigma_j A_j \frac{m_j}{\rho_j} W(r - r_j, h) \]  

Where \( m_j \) is the mass of \( j \) particle and \( \rho_j \) is the mass density of the particle

On the kernel equation, \( h \) is kernel’s radius that is plotted to get some particle that make up the volume of the modeling (Kelaher, 2003). The equation from the size of kernel’s radius is determined by the amount of particle with the equation below,

\[ h = \sqrt[3]{\frac{3V_x}{4\pi n}} \]
Where kernel’s radius can be changed based on the amount of the particle that gotten from the kernel’s radius in each time (Pirovano, 2011).

SPH method can estimates the quantity value and its derivative from the continued field based on the discrete points that are called smoothed particles. Each of the particles in SPH has mass property, position vector, velocity, and particle acceleration, and its quantity such as mass density and pressure. The value quantity is estimated from the effect of the particle in its surrounding inside the kernel’s radius.

Navier-Stokes equation in the formulation of Langrangian as the fluid for the incompressible and isothermal is shown by Kelager (2003) in this equation,

\[
\begin{align*}
\rho \left( \frac{du}{dt} + \nabla \cdot u \right) &= -\nabla p + \mu \nabla^2 u + \mathbf{f}_{\text{external}} \\
\end{align*}
\]  

With the water flow that assumed by incompressible flow (constant mass density), so the equation is

\[
\nabla \cdot u = 0
\]  

Therefore the equation that can be used in the SPH method is below,

\[
\rho \frac{du}{dt} = -\nabla p + \mu \nabla^2 u + \mathbf{f}_{\text{external}}
\]  

SPH’s method notice about the velocity that is happening in each particle based on the basic form of the velocity of Langrangian’s, particle is received by measuring the density, internal and external forces (each particle). Internal forces are a force that occurred because of pressure and viscosity, while external forces are forces that affect particle movement because of the gravity, buoyancy, and surface pressure. (Muller et al, 2003)

Therefore it can be known if it is the basic Navier-Stokes equation because of internal and external force is below,

\[
\rho \frac{du}{dt} = \mathbf{f}_{\text{internal}} + \mathbf{f}_{\text{external}}
\]  

So, the velocity equation is

\[
\frac{du}{dt} = \frac{(\mathbf{f}_{\text{internal}} + \mathbf{f}_{\text{external}})}{\rho} a_i = \frac{(\mathbf{f}_{\text{internal}} + \mathbf{f}_{\text{external}})}{\rho_i}
\]  

The relation between fluid interaction and solid affect each of the contacts that happened because of the interaction power that can differentiate the value and the direction motion. Cundall and Hart (1992) differentiate between hard-contact, an interpenetration of the body considered as non-physics, and soft-contact that would be possible to become interpenetration. In the simulation, the condition of hard-contact can give a result in the surface’s deformation. Besides that, the condition of soft-contact is assumed as equal because the contact power is related to the movement and the amount of particle. Based on the concept of discrete method, an element is possible to make interaction accurately and stable. It can be simulated without boundaries with the amount of its particle, the amount of particle is
limited in accordance with the computing power. When the maximum penetration is reached, the power of the particle can counterbalanced the force that occurred because of its collision.

2.2. Research Method
In this research, there will be a modeling simulation scenario of the fluid-solid interaction using Smoothed Particle Hydrodynamics method. The modeling variation was done using a different slope and will develop it with the big amount of particle and interaction to the soil layer. The modeling variation is done by analyzing the response of fluid and solid interaction step by step.

Figure 1. Model Scenario in 2D

Figure 2. Model Scenario in Oblique's look

Here are the steps that are going to be used,

Figure 3. Flow of the Modeling Plan
In its development, the modeling of interaction between fluid-solid particle based on Müller et al. (2005), the equation that changed from the basic Navier-Stokes is in the measurement of the force due to the viscosity. By modifying the force equation and adding the type of particles such as sand and silt surface it will be connected into the fluid. Based on figure 3.2, the flow of the program that will be developed shows the relation of interaction between fluid-solid with a limit of an impermeable layer in the form of silt in the upper layer and the upper and below layer or between that fluid and solid. Here is the plan for interpolation and the modification of the development,

**Figure 4. Development plan and modification**

The first step is initialization of variable, parameter, and initial condition. In this step, there will be an input process in each variable that defined in the first research such as space geometry, initial position and velocity in each particle for \( t=0 \), entity in each particle, initial condition of the SPH formula, and the amount of particle used during the simulation experiment.

Based on that condition and variable, the measurement is done against distance between particle and mass density for the whole particle that is already defined in the beginning. Then, there will be measurement computation to the forces that happened in the internal forces either pressure force and viscosity, and external forces in the form of gravity, buoyancy force, surface tension force, boundary forces. After that, there will be measurement of the occurring forces, the result is summed and it will become the value of total force that happened in each particle. The total force values are going to be
divided with the density under the Navier-Stokes equation to get acceleration during the experiment in each particle.

The particle who experienced collision will have acceleration change in each particle with the new value of acceleration. By that, we can get the new value of the position and velocity with the new acceleration differentiation. The value of the position and acceleration will be saved and the output will be processed with the time that has been determined to reach maximum time step under the experiment simulation.

3. Result and Discussion

3.1. Result

At the beginning of the modeling, the authors modeled the initial fluid and solid particle as below,

![Image](image_url)

**Figure 5.** Modeling 1 from the Side View

The slope to x-axis is 14 degrees and the model of one solid layer under one fluid layer on the z-axis. This modeling is based on the space geometry in the experiment of “water film in liquefied sand and its effect on lateral spread” (Kokusho, 1999).

In the initial modeling, the response got from the fluid particle that can recognize the solid particle when there is collision. It can make an interaction because of the impact of boundary force. Here is picture of the modeling based on the time change,
Figure 6. Result of the initial modeling on the time step (a) 0.001 s; (b) 0.015 s; (c) 0.025 s; (d) 0.040 s; (e) 0.055 s; (f) 0.070 s.

In the interaction between fluid and solid particles, the response from the collision creates reflection based on the line direction of the normal force in each of the particles that is perpendicular to the contact surface of the solid particle. The particle response is based on the collision that happened because it inside the boundary force in 0.040 s.

Table 1. Modeling 3 Property

| X in SupportRadius | Radius Kernel (meter) |
|--------------------|-----------------------|
| 10                 | 0.00452               |
| 30                 | 0.00642               |
| 50                 | 0.00762               |
| 100                | 0.00959               |
| 150                | 0.01098               |

Here is the graphic of the relationship between radius kernel and x in support radius,
With x variation in the support, the radius is based on table 3 and figure 7, the relationship between radius kernel and x in the support radius.

Based on the result of the first modeling program, we can see the behavior of the fluid and solid visually, and the analysis of variables can be reviewed when there is response from the collision between fluid and solid particle in 0.040 s. Here are the result of the forces:

**Table 3. Forces based on the variation of the Radius Kernel in Modeling 1**

| X in Support Radius | Pressure Force | Viscous Force | Surface Tension Force | Gravity Force | Boundary Force | Total Force |
|---------------------|----------------|---------------|-----------------------|---------------|----------------|-------------|
| 10                  | 1.05           | 8.61          | 7214.71               | 6776.10       | 6930.58        | 5702.54     |
| 30                  | 3.60           | 7.25          | 1547.68               | 6076.95       | 5921.77        | 1041.60     |
| 50                  | 1.74           | 2.24          | 480.49                | 5116.27       | 5088.38        | 423.55      |
| 100                 | 1.33           | 1.23          | 185.94                | 4051.73       | 4158.94        | 234.83      |
| 150                 | 0              | 0             | 0                     | 287.05        | 0              | 0           |

[Figure 7. The relationship between Radius Kernel and X in Support Radius]

[Table 2. The Modeling Property Detail 1]

|                         | Fluid         | Solid        |
|-------------------------|---------------|--------------|
| Spacing between Particle| 0.00333       | 0.00204      |
| Volume                  | 0.0000231     | 0.0000212    |
| Amount of Particle      | 625 Particle  | 2500 Particle|
| SpacingRatio            | 0.613         |              |
| Stiffness Coefficient   | 35000         |              |
| Damping Coefficient     | 276.4         |              |
| Coefficient of Friction | 0.1           |              |
| Iteration               | 100           |              |
| dt                      | 0.001         |              |
3.2. Discussion

In the next modeling (modeling 2), the authors modeled the initial position of fluid, and solid particle with the solid particle layer is in between the fluid particle layer as seen below,

![Figure 8. Modeling 2 from the Side View](image)

The slope to x-axis is under 14 degrees, and the model of one solid layer under one fluid layer on the z-axis. In the initial modeling, the response got from the fluid particle can be recognized by the solid particle when there is collision. It can make an interaction because of the impact of boundary force. Here is the modeling based on the time change as seen below,

![Figure 9. The result of the continuation modeling on the time step of](image)

In the interaction between the fluid and solid particle, the response from the collision create reflection based on the line direction of the normal force in each of the particles that is perpendicular to the contact surface of the solid particle. The particle response is based on the collision that happened because it inside the boundary force in 0.029 s.
With the variation of stiffness coefficient and damping coefficient that adjust the stable and ideal condition based on the research model purpose.

Based on the programming result, the second modeling said that the attitude of fluid and solid could be seen visually, and the analysis of variables can be reviewed when the response of fluid and a solid particle when it blows in the 0.029 s. In this modeling there is testing to the variation of the stiffness coefficient and the damping coefficient. Here are the results of the testing that is very close to stability based on the expected conditions using some of the results of the forces below.

Table 5. Forces based on the Variation of Radis Kernel in Modeling 2

| Ks     | Kd  | Pressure Force | Viscous Force | Surface Tension Force | Gravity Force | Boundary Force | Total Force |
|--------|-----|----------------|---------------|-----------------------|---------------|----------------|-------------|
| 20000  | 53.3| 0.962          | 6.07          | 7593.60               | 7495.89       | 5493.57        | 7055.28     |
| 20000  | 276.4| 0.858         | 4.34          | 7612.35               | 7516.570      | 6901.83        | 6987.57     |
| 25000  | 276.4| 0.898         | 4.59          | 7668.96               | 7519.58       | 7560.80        | 6900.54     |
| 35000  | 276.4| 0.945         | 4.82          | 7724.05               | 7502.28       | 8388.48        | 6783.16     |

Based on the table, surface tension force, gravity force, and boundary force are dominating forces. Here is the advanced analysis. In some of the experiment done by the authors, the value of Ks is close to the ideal condition where the range is 20000 to 35000, to modeling the ideal condition, the value of Ks must be suitable, therefore the relation between the value of Ks in it boundary condition takes effect to the contact respond happening between the particles, here are the value of the boundary force and the variation of Ks in modeling 2

Table 6. The relationship between Ks Value and Boundary Force

| Ks     | Kd  | Boundary Force |
|--------|-----|---------------|
| 20000  | 53.3| 5493.57       |
| 20000  | 276.4| 6901.83       |
| 25000  | 276.4| 7560.80       |
| 35000  | 276.4| 8388.48       |
Based on the data gotten, the value of Ks or Stiffness Coefficient has a value that directly proportional to the value of Boundary Force; the bigger the value of boundary force the bigger the value of stiffness coefficient when there is a collision. The value of Ks or Stiffness Coefficient has an important effect in the no-penetration force, the value of Ks will show the magnitude of solid property, so there will be no penetration by the fluid particle.

It is very important to know the value of Ks to fulfill the stable or ideal condition to visualize the attitude of fluid and solid in accordance with the expected condition. The value of Ks also affects the fluid particle to not recognize the solid particle when the value of KS in the experiment is more than equal to 50000.

4. Conclusion
In modeling and analyzing, the effect of land surface slope on the water layer movement to the silt layer can use numeric approach method, which is the Smoothed Particle Hydrodynamics (SPH). Modeling with the effect of the particle’s slope as the initial position can affect the fluid particle movement to move randomly. Modeling can be done in three steps. In the first step, using modeling 1, most of the fluid particle can recognize the encountered particles, the value of x in the support radius affect the movement of fluid-particle with the contact responds to solid particle. The responses of stable collisions can happen when the solid particle able to be recognized by fluid particle, a collide particle will reflect following the velocity and line direction that is created, and fluid will not surpass the impermeable land layer. The determination of collision boundary from the particle is affecting the particle movement with the initial position of the particle in certain slope condition, there will be different value and boundary direction that has been determined. Therefore with the different slope, it will be difficult to determine the movement boundaries of the fluid particle. The value of stiffness coefficient and damping coefficient affect the result of the collision response that happened. The stable modeling is affected by some of the parameters as seen in table 7.

Table 7. Result of the Modeling

|                     | Modeling 1 | Modeling 2 (Slope 25%) | Modeling 3 (Slope 12%) | Modeling 3 (Slope 16%) |
|---------------------|------------|------------------------|------------------------|------------------------|
| X in Support Radius | 10         | 10                     | 10                     | 10                     |
| Radius Kernel (m)   | 0.0045     | 0.0045                 | 0.0045                 | 0.0045                 |
| Stiffness Coefficient (N/m) | 35000     | 35000                  | 25000                  | 20000                  |
| Damping Coefficient (Ns/m) | 276.4      | 276.4                  | 276.4                  | 276.4                  |
In that modeling, the response of the collision that happened was affected by the forces of the particles, such as surface tension force, gravity force, and the one which affects the collision is boundary force. Boundary force affects the no-penetration fluid particle and damps the oscillations. The parameter that affects the modeling scenario includes the amount of solid particle, the more particles included it will eventually make the space between solid particle space so the collision will be fulfilled. In Kernel Radius, the more the scope is included in the measurement the more the particle will be processed, so the effective radius is needed in each different condition. But Stiffness Coefficient has the function to prevent the penetration when there is collision between fluid and boundary, the more stiffness coefficient is included it will enlarge the boundary force. Damping coefficient will prevent oscillations after there is collision in the small changing time and has the function to know the different particle collision force with not congruent shape. In this modeling, it hasn’t portrayed the soil movement under the liquefaction phenomenon because it needs more new algorithms to visualize the soil movement that followed the liquefaction phenomenon. This modeling portrayed a modeling of water layer in between the impermeable soil and used numeric approach of Smoothed Particle Hydrodynamics (SPH) as a method.

References

[1] Andersson S, Söderberg A, and Björklund S 2007 Friction models for sliding dry, boundary and mixed lubricated contacts Tribology International. 40 4 pp 550-587.
[2] Brendel L and Dippel S 1998 Lasting Contacts in Molecular Dynamics Simulations (Physics of Dry Granular Media) pp 313-318.
[3] Bui H, Fukugawa R, Sako K, and Ohno S 2008 Langrangian meshfree particles method (SPH) for large deformation and failure flows on geomaterial using elastic-plastic soil constitutive model International Journal for Numerical and Analytical Methods in Geomechanics. 32 pp 1537-1570.
[4] Chen W and Qiu T 2012 Numerical simulations for large deformation of granular materials using smoothed particle hydrodynamics method International Journal of Geomechanics. 12 2 pp 157-164.
[5] Colagrossi A and Landrini M 2003 Numerical Simulation of interfacial flows by smoothed particle hydrodynamics Journal of Computational Physics. 191 1 pp 448-475. doi:10.1016/S0021-9991(03)00324-3.
[6] Gomez-Gesteira M and Dalrymple R A 2004 Using a three-dimensional smoothed particle hydrodynamics method for wave impact on a tall structure American Society of Civil Engineering 130 2.
[7] Kelager M 2006 Langrangian Fluid Dynamics Using Smoothed Particle Hydrodynamics (Denmark: Departement of Computer Science, University of Copenhagen).
[8] Kokusho T 1999 Water Film in Liquefied Sand and Its Effect on Lateral Spread (Tokyo: Japanese Geotechnical Society).
[9] Liu G R and Liu M B 2003 Smoothed Particle Hydrodynamics: a meshfree particle method (Singapore: World Scientific Publishing Co. Pte. Ltd).
[10] Lydiana B C 2017 Pengujian Keandalan Metode Smoothed Particle Hydrodynamics (Depok: Universitas Indonesia).
[11] Maeda K and Sakai H 2010 Seepage failure and erosion of ground with air bubble dynamics. In GeoCivil Engineering and Geotechnics; Progress in Modeling and Applications pp 261-266.
[12] Menson B 2019 Geotechnical reconnaissance: the 28 september 2018 M 7.5 Palu-Donggala, Indonesia Earthquake (USA: Oregon State University).
[13] Monaghan J J 2005 Smoothed particle hydrodynamics Reports on Progress in Physics. 68 8 pp 1704-1759 doi:10.1088/0034-4885/68/8/R01.
[14] Monaghan J J, Kos A, and Issa N 2003 Fluid motion generated by impact Journal of Waterway Port Coastal and Ocean Engineering 129 6 pp 250-259.
[15] Muller M, Charypar D, and Gross M 2003 Particle-based Fluid Simulation for Interactive Applications. Eurographics/SIGGRAPH Symposium on Computer Animation (Switzerland: Department of Computer Science; Federal Institute of Technology Zurich) 154-159.

[16] Popov V L 2010 Contact Mechanics and Friction, Physical Principles and Applications (Berlin: Springer-Verlag Berlin Heidelberg).

[17] Potter M C, Wiggert D C, and Ramadan B H 2002 Mechanics of Fluids. Upper Saddle River (United States: CL Engineering).

[18] Pemerintah Indonesia 2007 Undang-Undang Republik Indonesia Nomor 26 Tahun 2007, Tentang Penataan Ruang (Jakarta: Pemerintah Indonesia).

[19] Vetsch D 2012 Numerical Simulation of Sediment Transport (Zürich: Zitiervorschlag für VAW-Mitteilungen; Versuchsanstalt für Wasserbau Hydrologie und Glaziologie der Eidgenössischen Technischen Hochschule Zürich).

[20] Vijaykumar A 2012 Smoothed Particle Hydrodynamics Simulation for Continuous Casting (Stockholm: Royal Institute of Technology; School of Engineering Science).

[21] Wulandari A 2018 Investigasi Nilai Stiffness Coefficient (Ks) Dan Damping Coefficient (Kd) Terhadap Parameter Numerik Dan Fisik Smoothed Particle Hydrodynamics (SPH) Pada Fenomena Interaksi Fluida-Solid (Depok: Universitas Indonesia).