Here, the traditional Extended Darcy's law to represent such high Reynolds number flow fields was suggested by Forchheimer. One can write,

$$\frac{\mu_f}{K}\overline{\boldsymbol{v}}_f + \frac{\rho_f C}{\sqrt{K}}\overline{\boldsymbol{v}}_f\left|\overline{\boldsymbol{v}}_f\right| = -\nabla P + \rho_f \boldsymbol{g} \tag{5}$$

The above equation, however, is not enough to simulate the collapse behavior of seawall and breakwater structure, and thus, it is necessary to unify the governing equations for both the free surface flow and seepage flow. The Akbari's equation is implemented in the current study as the unified governing equation, which can be described as the following,

$$\frac{C_r(\varepsilon)}{\varepsilon}\frac{D\overline{\boldsymbol{v}}_f}{Dt} = -\frac{1}{\bar{\rho}_f}\nabla P + \frac{\boldsymbol{g}}{\varepsilon} + \nu_E(\varepsilon)\nabla^2\overline{\boldsymbol{v}}_f - a(\varepsilon)\overline{\boldsymbol{v}}_f - b(\varepsilon)\overline{\boldsymbol{v}}_f\left|\overline{\boldsymbol{v}}_f\right| \tag{6}$$

Here, $\bar{\rho}_f$ means the local density of the fluid described by using porosity ε as $\bar{\rho}_f = \varepsilon\rho_f$. $\boldsymbol{v}_D$ is the Darcy flow velocity or averaged velocity, and it has a relationship with the regular velocity $\boldsymbol{v}_f$ as $\boldsymbol{v}_D = \varepsilon\boldsymbol{v}_f$. Various coefficients in the Akbari's equation are given as a function of porosity and their expressions are written as the following.

$$C_r(\varepsilon) = 1 + 0.34\frac{1-\varepsilon}{\varepsilon} \quad \text{: inertia coefficient} \tag{7}$$

$$\nu_E(\varepsilon) = \frac{\nu_w + \nu_t}{\varepsilon} \quad \text{: effective viscosity coefficient} \tag{8}$$

$$a(\varepsilon) = \alpha_c\frac{\nu_w(1-\varepsilon)^2}{\varepsilon^3 {D_{50}}^2} \quad \text{: linear coefficient} \tag{9}$$

$$b(\varepsilon) = \beta_c\frac{(1-\varepsilon)}{\varepsilon^3 D_{50}} \quad \text{: nonlinear coefficient} \tag{10}$$

Here, D_{50} is expressed as the average particle size of soils and the value of α_c and β_c are given as $\alpha_c = 200$ and $\beta_c = 1.1$, respectively, according to Akbari.

In Eq. (6), the 3rd and 4th term from the right side, which is not included in Navier-Stokes equation, can be considered as a resistance force acting on the fluid flows in the porous medium. This resistance force must act on the porous medium as well satisfying the action-reaction law. However, this unified governing equation is limited to only represent the fluid flows in the fixed porous medium. It is, therefore, necessary for us to consider the motion of a solid phase and reformulate the unified equation in order to analyze a multiphase flow. In some research of fluid-solid multiphase flow simulation such as Sakai et al. (2013), a drag force acting on the solid phase is considered as an interaction force which also acts on the fluid phase by action-reaction law. A drag force formulated by Ergun. (1952) and Wen and Yu (1966) are adopted and calculated from the relative velocity between the fluid and the solid phase. This formulation gives a modification to the unified governing equation described as,

$$\frac{C_r(\varepsilon)}{\varepsilon}\frac{D\overline{\boldsymbol{v}}_f}{Dt} = -\frac{1}{\bar{\rho}_f}\nabla P + \frac{\boldsymbol{g}}{\varepsilon} + \nu_E(\varepsilon)\nabla^2\overline{\boldsymbol{v}}_f - a(\varepsilon)\varepsilon\boldsymbol{v}_r - b(\varepsilon)\varepsilon^2\boldsymbol{v}_r|\boldsymbol{v}_r| \tag{11}$$

where $\boldsymbol{v}_r$ here is the relative velocity.

2.2. Solution of Fluid Equation by Stabikized ISPH Method

In this study, SPH method is adopted to solve the fluid governing equation. The basic concept in SPH method is that for any function φ attached to particle "i" located at r_i is written as a summation of contributions from its neighboring particles.

$$\varphi(r_i) \approx \langle\varphi_i\rangle = \sum_j \frac{m_j}{\rho_j}\varphi_j W(r_{ij}, h) \tag{12}$$

Note that, the triangle bracket $\langle\varphi_i\rangle$ means SPH approximation of a function φ. The divergence of vector function and gradient can be derived from the Eq. (13) and (14).

$$\nabla\cdot\varphi(r_i) \approx \langle\nabla\cdot\overline{\varphi}_i\rangle = \frac{1}{\rho_i}\sum_j m_j(\overline{\varphi_j} - \overline{\varphi}_i)\cdot\nabla W(r_{ij}, h) \tag{13}$$

$$\nabla\varphi(r_i) \approx \langle\nabla\overline{\varphi}_i\rangle = \rho_i \sum_j m_j\left(\frac{\varphi_j}{{\rho_j}^2} + \frac{\varphi_i}{{\rho_i}^2}\right)\nabla W(r_{ij}, h) \tag{14}$$

The Laplacian of the function is defined as

$$\langle\nabla^2\varphi_i\rangle = \sum_j m_j\left(\frac{\rho_i+\rho_j}{\rho_i\rho_j} + \frac{r_{ij}\cdot\nabla W(r_{ij},h)}{{r_{ij}}^2+\eta^2}\right)(\varphi_i - \varphi_j) \tag{15}$$

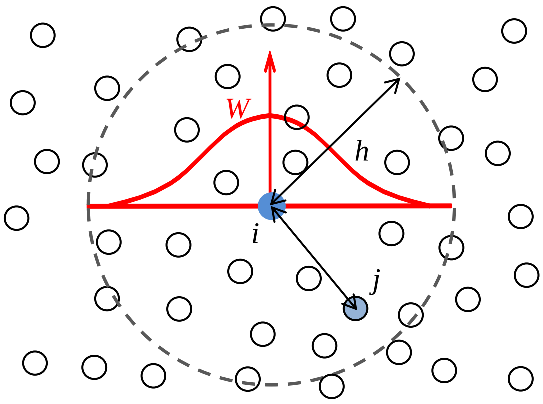

Figure 1. Particle placement and influence radius in the SPH method

The main concept of the stabilized ISPH method proposed by Asai (2012) for Navier-Stokes equation is to separate the governing equations of the incompressible fluid by using projection method. In this method, the pressure is calculated implicitly and the velocity fields are updated explicitly. In this study, the same idea of ISPH for the Navier-Stoke equation is applied to solve the unified equation as follows,

$$\frac{D\bar{\rho}_f}{Dt} + \bar{\rho}_f\nabla\cdot\left(\frac{\overline{\boldsymbol{v}}_f}{\varepsilon}\right) = 0 \tag{16}$$

$$\frac{C_r(\varepsilon)}{\varepsilon}\frac{D\overline{\boldsymbol{v}}_f}{Dt} = -\frac{1}{\bar{\rho}_f}\nabla P + \frac{\boldsymbol{g}}{\varepsilon} + \nu_E(\varepsilon)\nabla^2\overline{\boldsymbol{v}}_f - a(\varepsilon)\boldsymbol{v}_r - b(\varepsilon)\boldsymbol{v}_r|\boldsymbol{v}_r| \tag{17}$$

Here, the density can be assumed as a constant value as the fluid is assumed to be incompressible. Hence, Eq. (15) can change to the following relationship,

$$\nabla \cdot \bar{\boldsymbol{v}}_f = 0 \tag{18}$$

In the incompressible SPH method, the final pressure Poisson equation is given by

$$\langle \nabla^2 P_i^{n+1} \rangle = \frac{C_r(\varepsilon_i)}{\varepsilon_i} \frac{\rho_i^0}{\Delta t} \langle \nabla \cdot \bar{\boldsymbol{v}}_{fi}^* \rangle \tag{19}$$

During a numerical simulation, the 'particle' density may change slightly from the initial value because the particle density is strongly dependent on particle locations in the SPH method. If the particle distribution can be kept almost uniformly, the difference between 'physical' and 'particle' density may be vanishingly small, or in other words, an accurate SPH results in incompressible flow needs to keep the uniform particle distribution. For this purpose, the different source term in the pressure Poisson equation can be derived using the 'particle' density. In stabilized ISPH method, the pressure Poisson equation in Eq. (19) can be reformulated as:

$$\langle \nabla^2 P_i^{n+1} \rangle \approx \frac{C_r(\varepsilon_i)}{\varepsilon_i} \left(\frac{\rho_i^0}{\Delta t} \langle \nabla \cdot \bar{\boldsymbol{v}}_{fi}^* \rangle + \alpha \varepsilon_i \frac{\rho_i^0 - \langle \rho_i^n \rangle}{\Delta t^2} \right) \tag{20}$$

where α is the relaxation coefficient, $\boldsymbol{v}_{Di}^*$ is temporal velocity and triangle bracket $\langle \ \rangle$ means SPH approximation. Note that this relaxation coefficient is dependent on the time increment and the particle resolution. Then, the reasonable value can be estimated by the simple hydrostatic pressure test using the same settings on its time increment and the resolution. In this study, $\alpha = 0.01$.

2.3. Calculation of a Porosity and Updating a Physical Quantity

In this multiphase flow analysis model, porosity is an important factor to define each phase's motion. The porosity is calculated by the number of solid particles around a fluid particle. According to the SPH method, the equation for solving the porosity is defined as

$$\varepsilon_{f,i} = 1 - \sum_k V_{s,k} \widetilde{W}(r_{ik}, h) \tag{21}$$

Here, $\widetilde{W}$ is the modified weight function. The numerical error may occur in the area where there are not enough water particles such as around the wall and water surface. Then the weight function W can be modified as

$$\widetilde{W}(r_{ik}, h) = \frac{W(r_{ik}, h)}{\int_V W(r, h) dV} \tag{22}$$

where k indicates a solid particle while the porosity of a solid particle is calculated by using $\varepsilon_{f,j}$.

$$\varepsilon_{s,k} = \frac{\sum_j \frac{m_j}{\rho_j} \varepsilon_{f,j} W(r_{kj}, h)}{\int_V W(r,h)dV} \tag{23}$$

The representing volume of one fluid particle is different from its actual value when the particle flows in a porous medium, and thereby, it concludes the sparse of fluid particles in a porous medium. This variation of the volume is calculated simply by changing the density and given as

$$\bar{\rho}_f = \varepsilon \rho_f \tag{24}$$

$$V_f = \frac{m_f}{\bar{\rho}_f} \tag{25}$$

Here m_f is a constant value to satisfy the conservation law.

2.4. Solid Governing Equation in Fluid

In this study, the soil motion is analyzed by using DEM. In general, DEM particles make contact judgments at any time step with each other and they are moved by these contact forces. In addition to that, the fluid force also acts on soil particles, exerting pressure which can move the solid particles as well. While considering multiphase flow analysis, the non-overlap analysis model is conceivable at first. Here, the size of fluid particles has to be small enough compared to the solid particle size in order to calculate an accurate pressure in this model. However, nevertheless, the analysis cost and time will be increased by this assumption. To overcome this issue, an overlap model is adopted in this study, allowing the overlapping of fluid and solid particles without contacting each other. Instead of the interaction with pressure, the interaction force obtained by the porosity affecting each particle is considered. In this overlap model, the particle size of fluid can be as large as the solid particle and the analysis cost can be kept low, less than the non-overlap model.

There are several kinds of fluid forces acting on a solid particle, but their influences are different towards the solid particles motion. Then drag force and buoyancy force are adopted as the main fluid forces in this study. The buoyancy force is generated by the difference of each phase density. The momentum equation of one solid particle is shown as

$$m_s \frac{d\boldsymbol{v}_s}{dt} = (\rho_s - \rho_f) V_s \boldsymbol{g} + \boldsymbol{F}_d + \sum \boldsymbol{F}_c \tag{26}$$

where m_s, ρ_s, $\boldsymbol{v}_s$, V_s, $\boldsymbol{F}_d$, $\boldsymbol{F}_c$ stand for the solid mass, density, velocity, volume, drag force and contact force, respectively. The rotational motion is described by using angular velocity as

$$I \frac{d\boldsymbol{\omega}}{dt} = \sum \boldsymbol{T} \ , \tag{27}$$

where I, $\boldsymbol{\omega}$ and $\boldsymbol{T}$ are the moment of inertia, angular velocity and torque, respectively. For a sphere particle the moment of inertia is specific and determined as

$$I = \frac{m_s d_s^2}{10} \ , \tag{28}$$

where d_s stands for the diameter of a solid particle. The torque in this case can be calculated correspondingly from the contact force.

The contact force in DEM is generally calculated by a spring-dashpot model using the overlap displacement of a particle. In this model, the contact force is divided into two components of repulsive force in the normal direction and a friction force in the tangential direction as

$$\boldsymbol{F}_c = \boldsymbol{F}_c^n + \boldsymbol{F}_c^t \ , \tag{29}$$

where the superscript n and t represent normal and tangential direction. The normal contact force $\boldsymbol{F}_c^n$ is described as

$$\boldsymbol{F}_c^n = (-k\delta^n - \eta|\boldsymbol{v}_r^n|)\boldsymbol{n} \ , \tag{30}$$

where $k, \delta, \eta, \boldsymbol{v}_r^{\boldsymbol{n}}$ and $\boldsymbol{n}$ are the stiffness, the displacement, the damping coefficient, the relative velocity between solid-solid particles or solid-wall particles and normal unit vector. The tangential contact force $\boldsymbol{F}_c^t$ considering the sliding is described as

$$\boldsymbol{F}_c^t = \begin{cases} (-k\delta^n - \eta|\boldsymbol{v}_r^t|)\boldsymbol{t} & |\boldsymbol{F}_c^t| < \mu|\boldsymbol{F}_c^n| \\ -\mu|\boldsymbol{F}_c^n|\boldsymbol{t} & |\boldsymbol{F}_c^t| \geq \mu|\boldsymbol{F}_c^n|' \end{cases} \tag{31}$$

where μ and $\boldsymbol{t}$ are the friction coefficient and the tangential unit vector. Then, damping coefficient η is given by

$$\eta = -2\ln(e)\sqrt{\frac{k}{\ln^2(e)+\pi^2}\frac{2m_i m_j}{m_i+m_j}} \ , \tag{32}$$

where e is the coefficient of restitution. The torque is calculated from the tangential contact force.

$$\sum \boldsymbol{T} = \sum \boldsymbol{r} \times \boldsymbol{F}_c^t \ , \tag{33}$$

where $\boldsymbol{r}$ indicates the vector from center of the particle to the contact point.

The meaning of the drag force is the same to the interaction force between two phases. In some research of fluid-solid multiphase flow simulation, the drag force introduced by Ergun [3] and Wen and Yu [4] is used and given by

$$\boldsymbol{F}_d = \beta(\boldsymbol{v}_f - \boldsymbol{v}_s) \tag{34}$$

$$\beta = 150\mu_f \frac{(1-\varepsilon)^2}{\varepsilon d_s^2} + 1.75 \frac{(1-\varepsilon)\rho_f |\boldsymbol{v}_f - \boldsymbol{v}_s|}{d_s} \quad (\varepsilon \le 0.8) \tag{35}$$

$$\beta = 0.75 C_d \frac{(1-\varepsilon)\rho_f |\boldsymbol{v}_f - \boldsymbol{v}_s|}{d_s} \varepsilon^{-2.7} \quad (\varepsilon > 0.8) \tag{36}$$

In such research, the interaction between fluid and solid phase is done by adding this drag force to each fluid and solid momentum equation in opposite sign satisfying the action reaction law. However, notice that the interaction force acting on the fluid phase is already included in the fluid governing equation. According to this concept, the interaction force terms mentioned at section 2.2 is adopted while calculating the drag force exerted on the solid phase in this study. Moreover, the interaction force term in Eq.(11) also means a force per unit volume of fluid phase, so by using the volume and void ratio of a solid particle, the interaction force for one solid particle is shown as

$$\boldsymbol{F}_d = \bar{\rho}_f (a(\varepsilon) + b(\varepsilon)|\boldsymbol{v}_r|)\boldsymbol{v}_r \frac{V_s}{1-\varepsilon} \tag{37}$$

By expanding $a(\varepsilon)$ and (ε) , this equation will be

$$\boldsymbol{F}_d = (\alpha_c \frac{\nu_f \rho_f (1-\varepsilon)^2}{\varepsilon d_s^2} + \beta_c \frac{\rho_f (1-\varepsilon)}{d_s} |\boldsymbol{v}_r|)\boldsymbol{v}_r \frac{V_s}{1-\varepsilon} \tag{38}$$

In comparing Eq. (35) and Eq. (38), the similarities are seen, this indicates that we can regard the meaning of this interaction term as a drag force.

2.5. Validation Test

To validate this multiphase model by ISPH method, the experiment conducted by Sakai et al. is used in this study. In this experiment, water and glass beads are used to represent the fluid and solid bodies, and the dam-break test was simply done. The dimension of the tank is shown in Fig.2, and the water and glass beads are steady in the tank. The gate is pulled up toward a vertical direction with 0.68m/s and the water and glass beads start to move. According to the reference, the primary distance and density of water particle are 3cm and 1g/cm^3, and the diameter, density, restitution coefficient and friction coefficient of the solid particle are 0.27cm, 2.5 g/cm^3, 0.5 and 0.2 in this numerical analysis. Time increment in this analysis is 0.0001s. We compare the analysis result with the experimental result's snapshot and the comparison is shown in Fig.3.

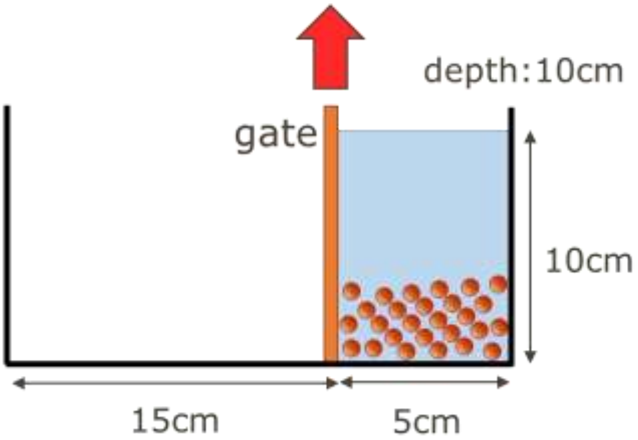

Figure 2. Dimension of the dam-break tank

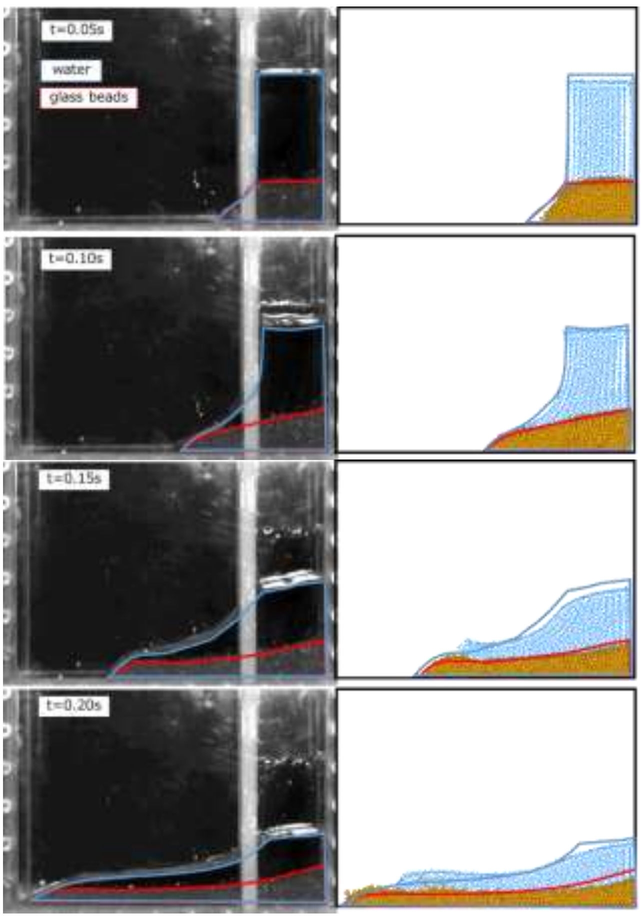

Figure 3. Comparison between experimental and analysis results

The result shows a good agreement with experiment, but at t=20s, the difference is still seen in the top of the water. There are some particles in there, but water particles are not distributed smoothly. This is due to the interaction force in the area where the porosity is high. The fluid governing equation proposed by Akbari is made under the condition of low porosity area. The interaction term is also suited to that area. The interaction term will need to be changed to satisfy the correct interaction force in that area.

3. CONCLUSIONS

In this study, an SPH-DEM coupled method for fluid-solid particle flow analysis has been developed based on the unified fluid governing equation between the free surface flow and the seepage flow which is proposed by Akbari. The additional resistance term from the original Navier-Stokes equation in the unified fluid governing equation is considered as an interaction force between the fluid and the solid particle, and the interaction force is added at each phase with opposite sign. Our numerical solutions show a good agreement with experimental result. However, it is also necessary for us to modify the interaction force in high porosity area. In the final future works, scouring phenomena behind the seawall will be analyzed by using this proposed method after improving the interaction force in the high porosity area.

REFERENCES

Akbari, H. (2014), “Modified moving particle method for modeling wave interaction with multi layered porous structures”, *Coast. Eng*., **89**, 1-19.

Xiaosong Sun, Mikio Sakai, Yoshinori Yamada. (2013), “Three-dimensional simulation of a solid-liquid flow by the DEM-SPH method, Journal of Computational Physics”, **248**, 147-176.

S. Ergun. (1952), “Fuluid flow through paked columns”, Chemical Engineering Progress **48**, 89-94.

C. Wen., Y. Yu. (1966), “Mechanics of fluidization”, Chemical Engineering Progress Symposium Series **62**, 100.

Asai, M., Aly, M.A., Sonoda, Y. and Sakai, Y. (2012), ”A Stabilized Incompressible SPH Method by Relaxing the Density Invariance Condition”, Journal of Applied Mathematics, **2012**, Article ID 139583.