BRIDGES WASH OUT SIMULATION DURING TSUNAMI BY A STABILIZED ISPH METHOD

YOSHIYA MIYAGAWA [1], MITSUTERU ASAI [2] AND NUR` AIN BINTI IDRIS [3]

[1] Department of Civil Engineering, Graduate School of Engineering, Kyushu University
Motoka744, Nishi-ku, Fukuoka 819-0395, JAPAN
e-mail: y_miyagawa@doc.kyushu-u.ac.jp,
web page: http://www.doc.kyushu-u.ac.jp/kouriki/index_jp.htm

[2] Dr. Eng., Associate Prof., Department of Civil Engineering, Graduate School of Engineering, Kyushu University
Motoka744, Nishi-ku, Fukuoka 819-0395, JAPAN
e-mail: asai@doc.kyushu-u.ac.jp,
web page: http://www.doc.kyushu-u.ac.jp/kouriki/index_jp.htm

[3] Department of Civil Engineering, Graduate School of Engineering, Kyushu University
Motoka744, Nishi-ku, Fukuoka 819-0395, JAPAN
Email:nurain@doc.kyushu-u.ac.jp

Key words: Tsunami, stabilized ISPH method, simulation, bridge wash out.

Abstract. In 2011, the huge tsunami caused by the great east Japan earthquake devastated many infrastructures in pacific coast of north eastern Japan. Particularly, collapse of bridges caused a traffic disorder and these collapse behaviors led to delay of recovery after the disaster. In this study, the bridge wash away accident is selected as a target issue, and it is represented in order to investigate the criteria and its mechanism by a numerical simulation. For this purpose, Incompressible Smoothed Particle Hydrodynamics (ISPH) Method, which is one of the pure mesh free methods, is utilized for the rigid body motion simulation. In this study, rigid body motion is introduced for the fluid-rigid interaction behavior during bridge wash away simulation. In the numerical analysis, the upper bridge structure is washed out by receiving an impact fluid force. The validation tests in two scales showed good agreement with experimental test and the real accident on the great east Japan earthquake tsunami.

1 INTRODUCTION

On March 11, 2011, the huge tsunami caused by the great east Japan earthquake devastated many infrastructures in pacific coast of north eastern Japan. Particularly, the damage of outflow of bridge girders caused a traffic disorder and these collapse behaviours led to delay of recovery after the disaster. After 2011 tsunami, disaster prevention and mitigation techniques are actively developing in coastal infrastructures and establishing prediction method for tsunami disaster is one of the severe issues toward the next millennium tsunami.

In this study, the bridge wash out accident is selected as a target issue, and we try to represent these accidents by using a numerical analysis. For this purpose, one of the mesh free methods;

Smoothed Particle Hydrodynamics (SPH) Method is utilized for Tsunami flow. The SPH technique was originally proposed by Lucy [1] and further developed by Gingold and Monaghan [2] for treating astrophysical problems. The main advantage of SPH and the other particle simulation is the absence of a computational grid or mesh since it is spatially discretized into Lagrangian moving particles. This allows the possibility of easily modeling flows with a complex geometry or flows where large deformations or the appearance of a free surface occurs. Recently, this method is widely used in field of fluid and solid dynamics. A stabilized ISPH [3], which is one of the modified versions of the SPH and can evaluate much smoother pressure distribution, has been developed by our research group. Then a fluid-sorid interaction algorithm including rigid body motion is developed in this study.

2 FLUID-RIGID INTERACTION FORMULATION

In this section, a stabilized ISPH, which includes a modified source term in the pressure Poisson equation, for incompressible flow is summarized firstly. Then the treatment of moving rigid body is introduced by reviewing the related work in distinct element method (DEM).

2.1 GOVERNING EQUATION

The governing equations are the continuum equation and the Navier-Stokes equation. These equations for the flow are represented as

$$\frac{D\rho}{Dt} + \rho\nabla\cdot\boldsymbol{u} = 0 \tag{1}$$

$$\frac{D\boldsymbol{u}}{Dt} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\boldsymbol{u} + \frac{1}{\rho}\nabla\cdot\boldsymbol{\tau} + \boldsymbol{F} = \boldsymbol{0} \tag{2}$$

here, ρ and ν are density and kinematic viscosity of fluid, $\boldsymbol{u}$ and p are the velocity and pressure vectors of fluid respectively. $\boldsymbol{F}$ is external force, and t indicates time. The turbulence stress τ is necessary to represent the effects of turbulence with coarse spatial grids. In the most general incompressible flow approach, the density is assumed by a constant value with its initial value.

2.2 MODIFICATION IN SOURCE TERM OF PRESSURE POISSON EQUATION

The main concept in an incompressible SPH method is to solve a discretized pressure Poisson equation at every time step to get the pressure value. In a sense of physical observation, physical density should keep its initial value for incompressible flow. However, during 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 keep almost uniformity, the difference between 'physical' and 'particle' density may be vanishingly small. In other words, accurate SPH results in incompressible flow need to keep the uniform particle distribution. For this purpose, the different source term in pressure Poisson equation can be derived using the 'particle' density. The SPH interpolations are introduced into the original mass conservation law before the perfect compressibility condition is applied.

$$< \nabla\cdot\boldsymbol{u}_i^{n+1} > \; = -\frac{1}{\rho^0}\frac{< \rho_i^{n+1} > - < \rho_i^{*} >}{\Delta t} \tag{3}$$

Then, the pressure Poisson equation reformulated as:

$$< \nabla^2 p_i^{n+1} > \ = \frac{\rho^0}{\Delta t} < \nabla \cdot \boldsymbol{u}_i^* > + \alpha \frac{\rho^0 - < \rho_i^* >}{\Delta t^2} \tag{4}$$

where, α is relaxation coefficient, u^* is temporal velocity and triangle bracket $< >$ means SPH approximation. Note that this relaxation coefficient is strongly 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.

2.3 TREATMENT OF MOVING RIGID BODY

In this study, the general momentum conservation law of the rigid body is solved numerically with the external forces including hydrodynamic as a fluid-rigid interaction formulation. Fig.1 show the rigid motion algorithm. The rigid body is discretized to particles at the beginning. Next, we postulate that the external force is calculated from the gravity $\boldsymbol{g}$, hydrodynamic force at the rigid surface $\boldsymbol{F}_f$ and contact force between rigid body and fixed boundary $\boldsymbol{F}_e$, and these value are calculated by following equations.

$$\boldsymbol{F}_f = \sum_{\boldsymbol{i}}^{on\,the\,surface} \boldsymbol{P}_i \Delta S_i \boldsymbol{n}_i \tag{5}$$

$$\boldsymbol{F}_e = k\delta^{3/2} \tag{6}$$

$$\delta = r - l \tag{7}$$

$$k = \frac{4\sqrt{r}}{3} \frac{E_i E_j}{\left(1-\nu_i^{\,2}\right)E_j + \left(1-\nu_j^{\,2}\right)E_i} \tag{8}$$

where, $\boldsymbol{P}_i$ is the pressure of surface rigid particle, and ΔS_i is the area of surface rigid particle. And the contact force is modeled by the elastic contact theory by Hertz. In the above equation, l is the distance between rigid particle and the fixed boundary. In above equation, ν_i, E_i, E_j is Poisson ratio, elastic modulus of the particle coming into contact, elastic modulus of the contacted particle. Once the hydrodynamic and external forces are obtained, the moment is easily calculated by using the distance between the target particle and center of gravity of rigid body.

$$\boldsymbol{M}_f = \sum_{\boldsymbol{i}}^{on\,the\,surface} (\boldsymbol{r}_i - \boldsymbol{r}_c) \times \boldsymbol{P}_i \Delta S_i \boldsymbol{n}_i \tag{9}$$

$$\boldsymbol{M}_e = \sum_{\boldsymbol{i}}^{on\,the\,surface} (\boldsymbol{r}_i - \boldsymbol{r}_c) \times \boldsymbol{F}_e \tag{10}$$

The translational velocity $\boldsymbol{T}$ and rotational velocity $\boldsymbol{\omega}$ of a solid object are calculated as:

$$\boldsymbol{T}^{n+1} = \boldsymbol{T}^n + \Delta t \left(\boldsymbol{g} + \frac{\boldsymbol{F}_f}{m} + \frac{\boldsymbol{F}_e}{m} \right) \tag{11}$$

$$\boldsymbol{\omega}^{n+1} = \boldsymbol{\omega}^n + \boldsymbol{I} \Delta t \left(\boldsymbol{M}_f + \boldsymbol{M}_e \right) \tag{12}$$

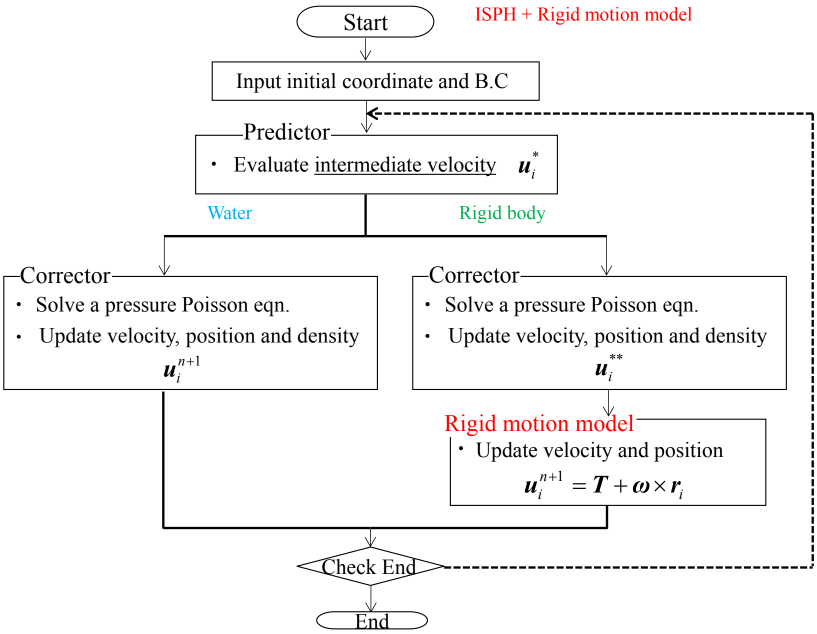

Figure 1. Rigid motion algorithm

Finally, the velocity of each particle in the solid body is updated by the following equation. The position of the rigid particle is updated based on the velocity.

$$\boldsymbol{u}_i^{n+1} = \boldsymbol{T}_i^{n+1} + \boldsymbol{\omega}_i^{n+1}\left(\boldsymbol{r}_i - \boldsymbol{r}_j\right) \tag{13}$$

Fig. 1 summarizes the whole flowchart for the fluid-rigid coupling by the stabilized ISPH method.

3 VALIDATION TEST

In the following section, two type of validation tests have been introduced. One is comparison between analysis and experimental test in a small scale and the other is real scale validation by comparing with disaster report.

3.1 SMALL SCALE VALIDATION TEST

The analysis model and the detail of the girder model are shown in Fig.2 and Fig.3. In this experiment and analysis, water which collected on the left-hand flow and collide with the girder model by opening the gate and girder wash out. The experimental test is carried out three times and the location of girder model is recorded by using motion capture system. In this system, the self-light is attached on the four corners of bridge girder model, its position is measured by using multi cameras. The bridge pier is fixed and the density of the bridge girder which is on the pier is 1161kg/m^3. The particle distance d_0 = 0.25cm, time increment Δt = 0.0005s and the total number of particles is about 8millions.

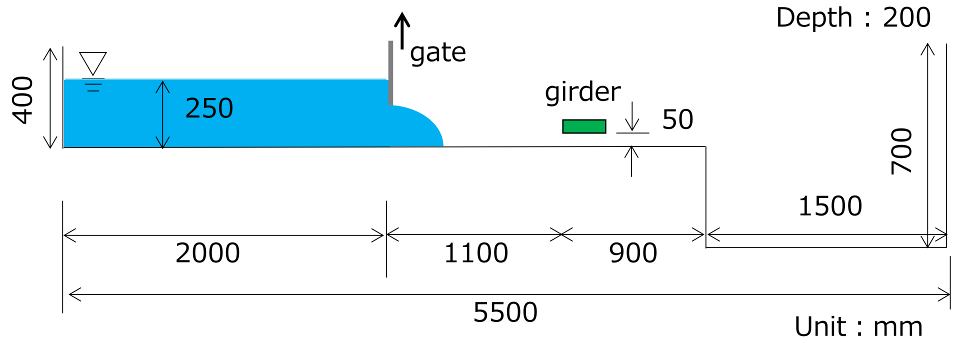

Figure 2. Analysis model

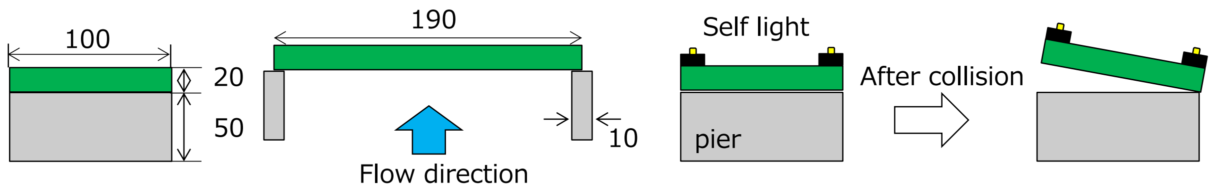

Figure 3. Detail of the bridge model and motion capture system

Fig.4 shows the comparison of rotational angle between the experimental test and the analysis. From this graph, there are some difference between the experimental result and the analysis, however, analysis result behaviors which rotate from positive angle to negative angle shows a good agreement. From this result, our proposed method can evaluate the rigid motion during flow in a small scale.

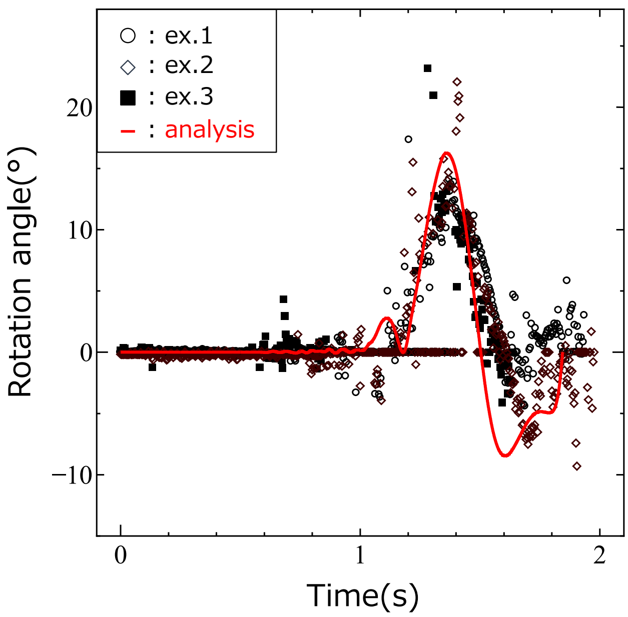

Figure 4. Comparison of rotational angle between the experiment and analysis

3.2 REAL SCALE VALIDATION TEST

One of the segments of a bridge girder, which was pushed away in the Tsunami, is selected as a target structure, and its numerical model is generated in 3D from the CAD data. The wave is modeled for a gentle streem and the initial water level is set to be 15m referring to the report that water levels in many disaster cities reached over 10m in the Tsunami (Fig.5). The initial velocity of the wave is set 10m/s referring to shallow water long-wave equation. In addition, 10m/s is continuously given at the position of 30m from the left corner of the water storage. The particle distance d_0 = 6cm, time increment Δt = 0.001s and total number of particles is about 55millions. The density of particles of the girder model is 2450kg/m^3.

The real scale wash out simulation is shown in Fig.6. According to the disaster report[4], this girder turned over in the Tsunami and the girder motion with rotate can be seen, however, the bridge girder don't overturn in the analysis. Now, the initial water level and the initial velocity is set constantly. It is neccesay to reconsider inflow condition by using tsunami run up simulation in order to simulate in realistic.

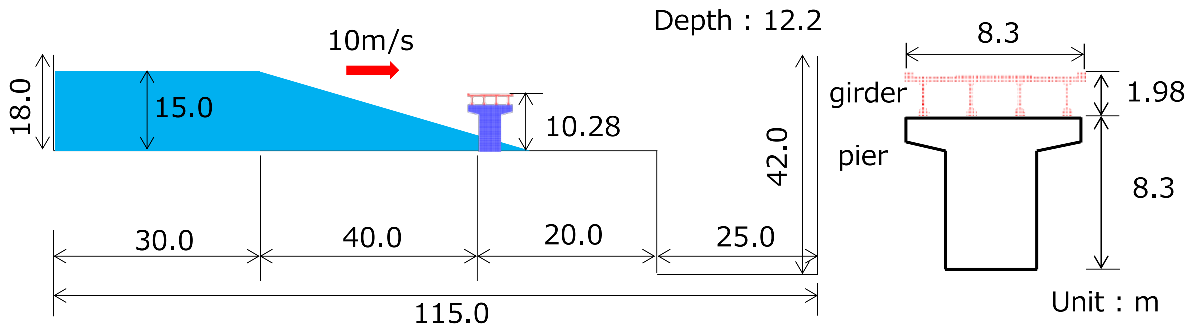

Figure 5. Analysis model and the shape of girder model

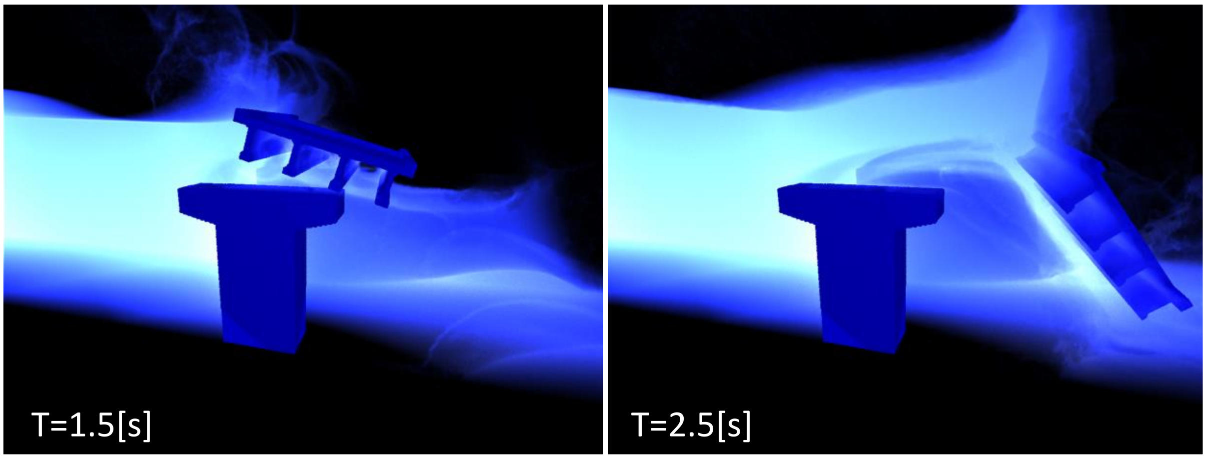

Figure 6. Analysis result (Real scale analysis)

4 CONCLUSIONS

In this study, fluid-rigid interaction formulation is conducted by introducing rigid motion algorithm into ISPH. In the small scale analysis, the validation of our fluid-rigid interaction technique is conducted by comparison with experimental test. From this result, the transition of rotational angle shows a similar tendency and a good agreement quantitatively. Then, real scale validation test is instituted by modeling the bridge which washed out in the Tsunami. There are some difference of rigid motion in report and analysis, however, analysis result behaviors shows largely rotation like the disaster report on March 11, 2011. In the future work, tsunami inflow condition must be reconsidered to predict the bridge wash out accident with accuracy.

ACCKNOWLEGEMENTS

This work was supported by JSPS Grant-in-Aid for Scientific Research(B) 26282106 and Grant-in-Aid for Exploratory Research15K12484.

REFERENCES

[1] Lucy L.B: Numerical approach to the testing of the fusion process, Astron J.; 88: 1013–1024.

[2] Gingold R.A., Monaghan J.J: Smoothed particle hydrodynamics: theory and application to non-spherical stars, Mon Not R Astron Soc, 181: 375–89.

[3] M. Asai, AM. Aly, Y. Sonoda and Y. Sakai : A stabilized incompressible SPH method by relaxing the density invariance condition, *Int.l J. for Applied Mathematics*, Volume 2012, Article ID 139583, 2012.

[4] Japan Society of Civil Engineers : Evaluation of Tsunami Force Acting on Bridge Structures, 2013