11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
E. Oñate, J. Oliver and A. Huerta (Eds)

NUMERICAL EVALUATION OF TSUNAMI IMPACT FORCE ACTED ON A BRIDGE GIRDER DURING TSUNAMI BY USING A PARTICLE METHOD

SHOICHI TANABE[*], MITSUTERU ASAI[†], KENJIRO TERADA[‡], KAZUO KASHIYAMA[§], SHUJI MORIGUCHI[**] AND MAO KURUMATANI[††]

[*] Department of Civil Engineering, Kyushu University
744 Motooka, Nishi-ku, Fukuoka, 819-0395, JAPAN
e-mail: tanabe@doc.kyushu-u.ac.jp

[†] Department of Civil Engineering, Kyushu University
e-mail: asai@doc.kyushu-u.ac.jp

[‡] International Research Institute of Disaster Science, Tohoku University
e-mail: tei@irides.tohoku.ac.jp

[§] Department of Civil and Environmental Engineering, Chuo University
e-mail: kaz@civil.chuo.ac.jp

[**] International Research Institute of Disaster Science, Tohoku University
e-mail: s_mori@irides.tohoku.ac.jp

[††] Department of Urban and Civil Engineering, Ibaraki University
e-mail: maok@mx.ibaraki.ac.jp

Key Words: *Particle Method, Tsunami, Fluid Force, Runoff of Bridge Girders.*

Abstract. On March 11, 2011, the huge Tsunami caused by the great east Japan earthquake devastated the Pacific coast of north-eastern Japan. Many infrastructures including bridges were collapsed by the Tsunami. New generation of Tsunami disaster prevention and mitigation method should be reconsidered toward the next millennium Tsunami. In this study, a stabilized Smoothed Particle Hydrodynamics (SPH) has been utilized for an evaluation of fluid force acted on bridge girders. In addition, a new boundary treatment using a virtual marker and fixed boundary particle is developed to satisfy the slip and no-slip boundary condition for the velocity field and to satisfy the pressure Neumann condition at the same time. The accuracy and efficiencies of our proposed method are validated by comparison between a numerical solution and experimental results. Finally, a simple treatment for the particle method has been introduced to our developed SPH tool to simulate wash out of the bridge girder.

1 INTRODUCTION

On March 11, 2011, the huge Tsunami caused by the great east Japan earthquake devastated the Pacific coast of north-eastern Japan. Many infrastructures including bridges and other Tsunami prevention facilities were collapsed by the Tsunami. Particularly, the damage of outflow of bridge girders caused a traffic disorder and these collapse behaviors led to delay of recovery after the disaster. A couple of years have passed since the huge Tsunami, measures for new Tsunami disaster prevention and mitigation methods are actively prompted toward the next millennium Tsunami. Recently, numerical evaluation of the fluid force during Tsunami is strongly desired for generating the new regulation of Tsunami disaster prevention, because real size experimental tests are almost impossible and too costly. In this study, the numerical evaluation of fluid forces acted on bridge girders is focused. A reasonable numerical simulation technique based on the Incompressible Smoothed Particle Hydrodynamics (ISPH) has been selected to represent complex flow phenomena around bridges. The features of our proposed simulation technique are stabilization of ISPH with a modified source term in pressure Poisson equation and a new boundary treatment using a virtual marker and the fixed boundary particle.

The source term in pressure Poisson equation (PPE) for ISPH is not unique. It has several formulations in the literature as Lee *et al*[1] and Khayyer *et al*[2,3]. The source term is derived from a function of density variation and velocity divergence condition. Both source terms are not complete the density invariant and divergence free condition, so modified schemes have been proposed to satisfy the above two conditions. Relaxation coefficient is multiplied in term of density invariant for smoothing the resultant pressure. Recently, in the framework of MPS, there is a trend to introduce a higher order source term in the PPE, Kondo and Koshizuka[4] and Tanaka and Masunaga[5]. In this paper, the similar approach is implemented with the ISPH for their stabilization and for smoothed pressure evaluation.

Generally, particle methods in fluid dynamics are not so easy to treat boundary conditions like pressure Neumann condition and slip or no-slip conditions on the solid surface. This is one of typical difficulties in mesh-less method. Recently, pressure Neumann condition in SPH is re-focused using a virtual marker and fixed boundary particle. A new boundary treatment using them is proposed to control the slip and no-slip boundary conditions on the solid boundary surface by model's resolution. The accuracy and efficiencies of our proposed method are validated by comparison between a numerical solution and experimental results.

2 IMPROVED ISPH

In this section, a stabilized ISPH (Asai *et al*[6]), which includes a modified source term in the pressure Poisson equation, for incompressible flow is summarized.

2.1 Governing equation

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

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

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

where 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 the 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, $\boldsymbol{u}_i^*$ 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.

3 BOUNDARY CONDITION

In this section, a new boundary treatment using a virtual marker and fixed boundary particle is proposed here. The concept of this treatment is to give a wall particle accurate physical properties, velocity and pressure. The procedure is summarized briefly.

3.1 Procedure for proposed boundary treatment

First, wall particle is placed on a grid-like structure with equally spaced in a solid boundary. Next, virtual marker is put in a position which is symmetrical to the wall particle across its solid boundary. Then, velocity and pressure on the marker are interpolated based on the concept of weighted average of neighboring particles, which is the fundamental equation of SPH shown as below.

$$\phi(\boldsymbol{x}_i, t) \approx \langle \phi_i \rangle = \sum_j \frac{m_j}{\rho_j} W(r_{ij}, h) \phi_j(\boldsymbol{x}_j, t) \tag{5}$$

However, the portion of the particles within the compact support may be in wall domain or vacant domain such as an air layer, in that case, interpolation using the modified weight function ($\tilde{W}$) as below is performed.

$$\phi(\boldsymbol{x}_i, t) \approx \langle \phi_i \rangle = \sum_j \frac{m_j}{\rho_j} \tilde{W}(r_{ij}, h) \phi_j(\boldsymbol{x}_j, t) \tag{6}$$

$$\tilde{W} = \frac{W(r_{ij}, h)}{\sum_j \frac{m_j}{\rho_j} W(r_{ij}, h)} \tag{7}$$

The most important point is that the marker is not directly related to the SPH approximation but uses just a computational point to give the wall particle accurate physical properties. Therefore, the density of the marker does not have a bad influence on accuracies in SPH and hence, it is possible to give the boundary condition more robustly. Moreover, computational cost can reduce compared to the ghost particle method[7] because the marker is created only once at the preprocess assuming that the wall particle is fixed.

Here, the velocity on the virtual marker which is corresponding to the wall particle across its solid boundary is interpolated by the SPH approximation shown as below.

$$\boldsymbol{v}(\boldsymbol{x}_v, t) \approx \langle \boldsymbol{v}_v \rangle = \sum_j \frac{m_j}{\rho_j} \tilde{W}(r_{vj}, h) \boldsymbol{v}_j(\boldsymbol{x}_j, t) \tag{8}$$

where x_v is the position of the targeting virtual marker and v_v is the velocity on the virtual marker. In order to satisfy the slip condition, the wall particle needs to be given the velocity, which is mirror-symmetric to the one on the virtual marker. This mirroring processing ($\boldsymbol{v}_v \rightarrow \boldsymbol{v}_w^{'}$) is given by the following equation.

$$\boldsymbol{v}_w^{'} = \boldsymbol{M}\boldsymbol{v}_v \tag{9}$$

where $\boldsymbol{M}$ is a second order tensor to implement the mirroring processing, and it is represented by the use of inward normal vector of the wall ($\boldsymbol{n} = (n_1, n_2, n_3)^T$) and the kronecker delta δ.

$$M_{ij} = \delta_{ij} - 2n_i n_j \tag{10}$$

On the other hand, in order to satisfy the non-slip condition, the wall particle needs to be given the velocity, which is point-symmetrical to the one on the virtual marker. $\boldsymbol{R}$ is a mirror-symmetric tensor and the velocity is given the same way as the Eq.(9).

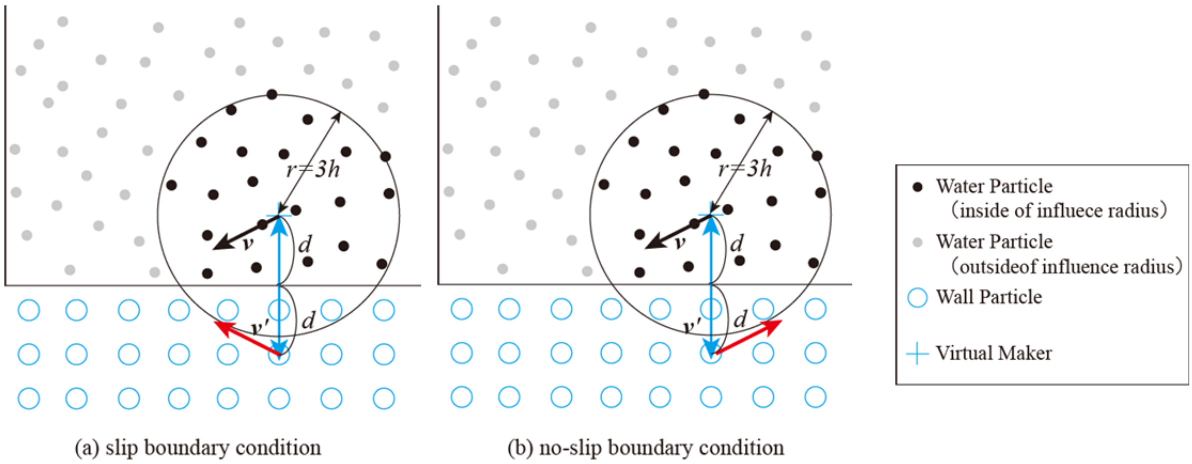


Figure 1: Velocity vectors for the slip and no-slip boundary conditions

$$\boldsymbol{v}_w^{'} = \boldsymbol{R}\boldsymbol{v}_v \quad , \quad R_{ij} = -\delta_{ij} \tag{11}$$

Fig.1 shows the examples of velocity vectors, which should be given the wall particles to satisfy slip or no-slip conditions. Practically, it is nessesarry to discuss a optimized condition between slip and no-slip condition to refer the resolution of its simulation model. Therefore, the following equation is proposed by using the coefficient of β ($0 \leqq \beta \leqq 1$).

$$\boldsymbol{v}_w^{'} = \beta \boldsymbol{M}\boldsymbol{v}_v + (1-\beta)\boldsymbol{R}\boldsymbol{v}_v \tag{12}$$

where β represents the ratio between slip and no-slip conditon.

In order to prevent from penetration of the water particle into the solid boundary, the pressure Neumann condition needs to be satisfied. For this purpose, giving the accurate pressure to the wall particle is necessary by referring to the one on the marker. Since normal component of the velocity on the solid boundary needs to be zero, the following equation needs to be satisfied.

$$\boldsymbol{v}_{w0} \cdot \boldsymbol{n} = 0 \tag{13}$$

where $\boldsymbol{v}_{w0}$ is the velocity of the solid boundary. Referring to the Navier-Stokes equation (Eq.(14)), which describes the same one as Eq. (2), the next non-uniform pressure Neumann condition needs to be satisfied to complete Eq. (13).

$$\frac{D\boldsymbol{u}}{Dt} = -\frac{1}{\rho^0}\nabla P + \underbrace{\nu\nabla^2\boldsymbol{u} + \boldsymbol{g}}_{f} \tag{14}$$

$$\partial p / \partial \boldsymbol{n} = \rho \boldsymbol{f} \cdot \boldsymbol{n} \tag{15}$$

In order to satisfy the non-uniform pressure Neumann condition in SPH method, it is necessary to give the wall particle the pressure, which is evaluated by the following equation.

$$p_w^{'} = \langle p_v \rangle + 2d\rho \langle \boldsymbol{f}_v \rangle \cdot \boldsymbol{n} \tag{16}$$

where p_v and $\boldsymbol{f}_v$ are the pressure and external force on the marker evaluated by SPH approximation. d represents the distance from a solid boundary to the targeting wall particle and triangle bracket < > means SPH approximation.

4 VALIDATION TEST

In the following section, the comparison between a numerical solution and experimental results has been introduced to validate the proposed scheme. The difference of the evaluation of fluid impact force by the boundary conditions is also investigated.

4.1 Analysis model

The analysis model and the detail of the girder model are shown in Fig.2. This experiment was carried out by Nakao *et al*[8], and the fluid impact force is evaluated while the wave acts on the girder model. The shape of the girder models are rectangle and upside down trapezoid. In this study, the numerical simulation is conducted by using the proposed boundary treatment described in section 3. First, velocity and water level are compared with experimental test and then, the fluid impact force is compared. The particle distance d_0 = 0.5cm, time increment Δt = 0.001s and the total number of particles is about 8millions.

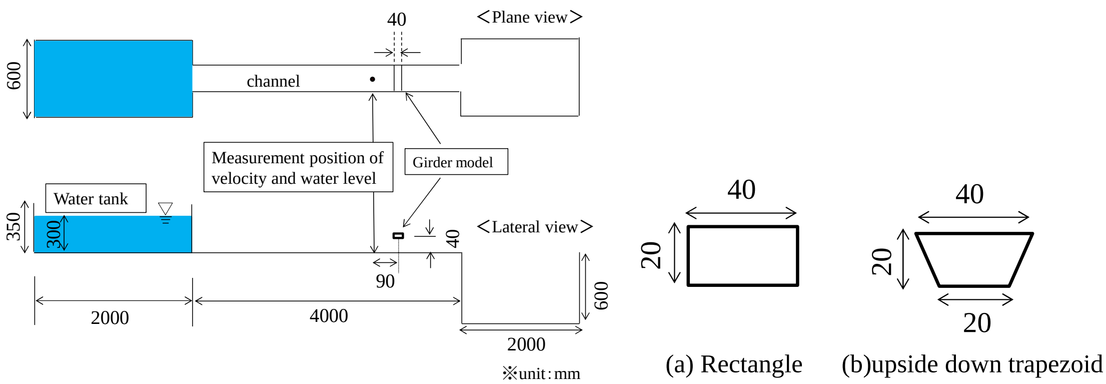


Figure 2: Analysis model and the shape of girder model

4.2 Result

Table 1 shows velocity value at the measurement point of the experimental model as shown in Fig.2. The result with the slip condition is faster than the experimental value, and the one with no-slip condition is later than the experimental value. Then, the calibration as to β which represents the rate of slip and no-slip condition is carried out to match the experimental and simulation result in velocity. In this simulation, the best value of β is obtained as 0.8 in Eq. (12).

Fig.3 shows the comparison of water level in each condition. From these results, the calibrated value is nearest to the experimental result including the rise of water level. Fig.4 shows the comparison of fluid force in rectangle model. This result is given by low pass filtering with 15Hz, which is the same treatment in the experimental result.

Table 1: Velocity in each condition

	Wave velocity(m/s)
Experiment	2.20
Slip condition(β=1)	3.06
Calibrated value(β=0.8)	2.24
No-slip condition(β=0)	1.71

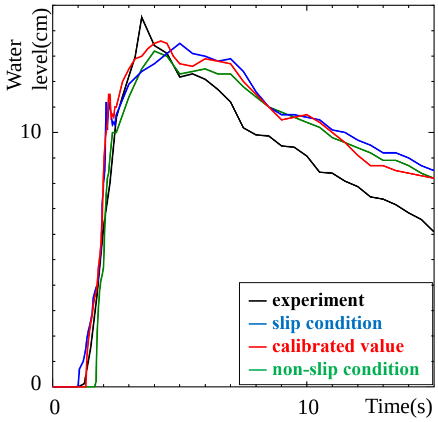


Figure 3: Comparison of water level

From the graph, each result roughly matches with the experimental one. However, considering the velocity and water level, the calibrated value is considered to be the best in this analysis. In addition, Fig.5 shows the comparison of fluid force between our proposed method and conventional one. From this graph, it is shown that the result of our proposed method is much better accuracy than conventional one. Since the conventional method is not completely satisfied the pressure Neumann condition and slip or no-slip condition on the solid boundary, the penetration of water particle into solid wall is occurred and hence, the accuracy of conventional method is not good.

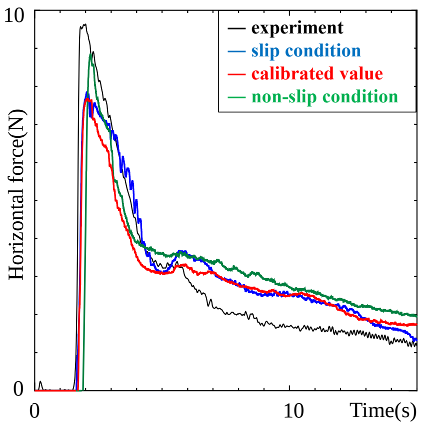


Figure 4: Comparison of fluid impact force

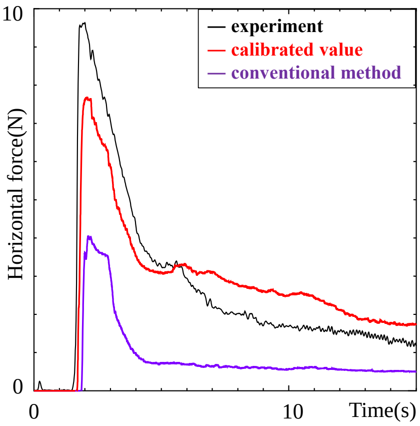


Figure 5: Comparison of accuracy

Fig.6 shows the comparison of fluid force in upside down trapezoid model. Likewise, the good result is obtained including lift force. From these results, our proposed method can evaluate the fluid impact force acted on a bridge girder on the practical level.

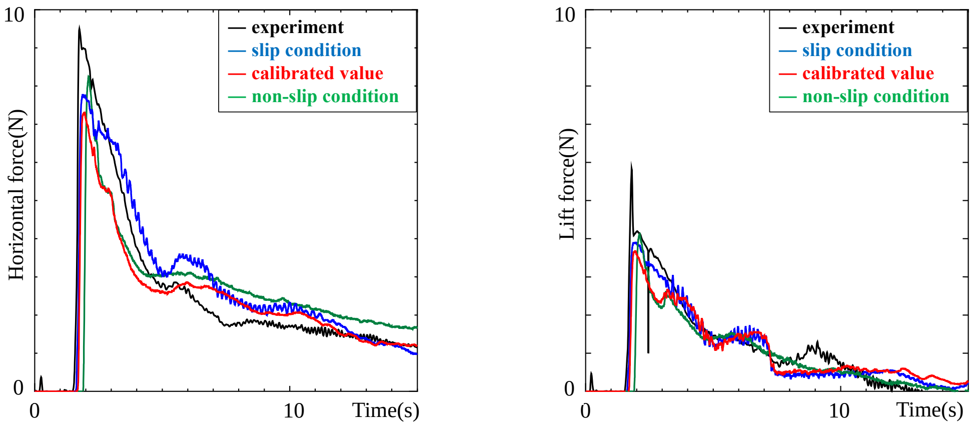


Figure 6: Comparison of fluid impact force in upside down trapezoid model

5 WASH OUT SIMULATION OF BRIDGE GIRDER

Finally, fluid-rigid coupling analysis is conducted by introducing rigid motion algorithm which is proposed by Koshizuka[9] in MPS method into ISPH algorithm(Fig.7). In this rigid motion algorithm, the velocity of the particle consisting rigid body is temporarily evaluated by solving as fluid which has the structure's density. Then, the translational motion and rotation momentum average of the rigid is evaluated by the temporal velocity, and hence, rigid body motion is given. A part of the results for wash out simulation of bridge girders is introduced.

5.1 Fluid-rigid coupling algorithm

First, the rigid particle which has different density to water is assumed as a water particle, and the velocity and pressure of it is evaluated the same as the water particle by ISPH algorithm. Next, for each rigid body, translational velocity $\boldsymbol{T}$ and angular velocity vector $\boldsymbol{\omega}$ are evaluated by following equation.

$$\boldsymbol{T} = \frac{1}{n}\sum_{k=1}^{n}\boldsymbol{v}_k \tag{17}$$

$$\boldsymbol{\omega} = \frac{1}{\boldsymbol{I}}\sum_{k=1}^{n}\boldsymbol{r}_k \times \left(m_k \boldsymbol{v}_k\right) \tag{18}$$

Then, the velocity of the rigid particle is updated by

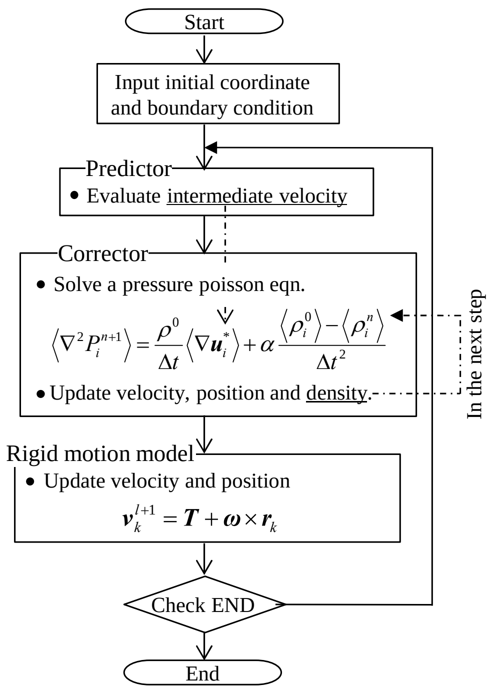


Figure7: Rigid motion algorithm

the above equations. Position of the rigid particle is updated based on the velocity.

$$\boldsymbol{v}_k^{l+1} = \boldsymbol{T} + \boldsymbol{\omega} \times \boldsymbol{r}_k \tag{19}$$

5.2 Analysis model

One of the segments of a bridge girder, which was pushed away in the Tsunami, is selected as a target structure(Fig.8), and its numerical model is generated in 3D from the CAD data. The wave is modeled for two cases: a surge stream and a gentle stream(Fig.9). Initial water level is set to be 15m referring to the report that water levels in many disaster cites reached over 10m in the Tsunami. 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 50m and 30m from the left corner of the water storage in case of a surge and gentle sream respectively. The depth in the models is 12.2m. The particle distance d_0 = 6cm, time increment Δt = 0.001s and the total number of particles is about 55 millions.

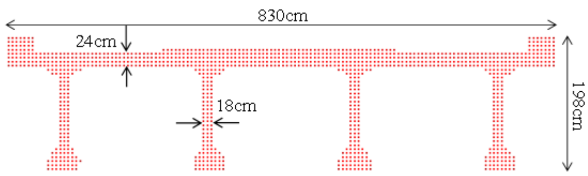
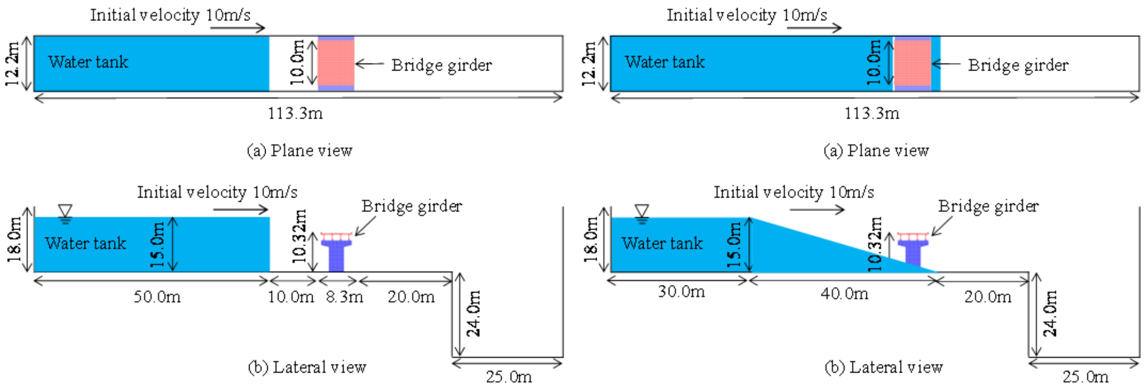


Figure 8: the girder model

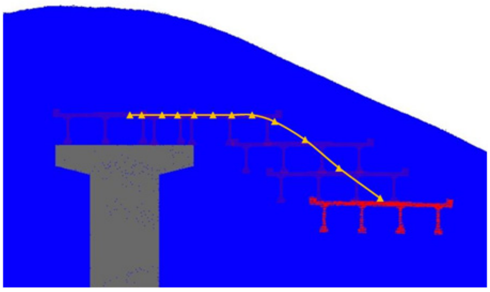


(i) a surge stream (ii)a gentle stream

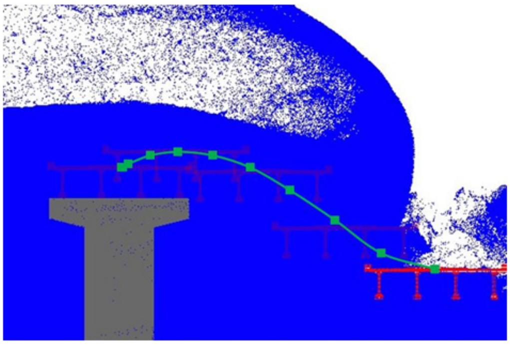
Figure 9: real size and shape girder model

5.3 Result

Fig.10 shows the snapshots of wash out simulation of the bridge girder in case of a surge stream and a gentle one. Focused on the trajectory of wash out behavior of bridge girder, the girder in case of a surge stream is pushed away horizontally, on the other hand, the girder in case of a gentle one is once lifted up by lift force and then push away by horizontal force. From the results, they show good agreements with our predictions for wash out behavior. This fluid-rigid coupling algorithm can represent the difference of wash out behavior by Tsunami shape. However, the motion of rotation cannot be occured by current algorithm in this simulation. Indeed, it is reported that the bridge girder is pushed away with rotating by Tsunami in this disaster. One of the cause is considered that the bearing and anchor are not modeled. In the future work, the better rigid body motion algorithm will be introduced to get much accurate results in the wash out behavior of the bridge girder.



(a) trajectory of a surge stream



(b) trajectory of a gentle stream

Figure 10: wash out behavior of bridge girder

5 CONCLUSION

A stabilized incompressible smoothed particle hydrodynamics is proposed to simulate free surface flow and to evaluate the water induced impact force. The modification is appeared in the source term of pressure Poisson equation, and the idea is similar to the recent development in Moving Particle Semi-implicit method (MPS). In addition, a new boundary treatment using a virtual marker is proposed to control slip and no-slip condition to refer the resolution of the numerical model. The accuracy and efficiency of our proposed method are validated by comparison between a numerical solution and experimental results. Finally, fluid-rigid coupling analysis is conducted by introducing rigid motion algorithm into ISPH. The proposed algorithm can represent the difference of wash out behavior by Tsunami shape. In the future work, better ridigd body motion algorithm will be introduced to get much accurate results in the wash out behavior of the bridge girder.

REFERENCES

[1] Lee E-S, Moulinec C, Xu R, Violeau D, Laurence D, Stansby P. Comparisons of weakly compressible and truly incompressible algorithms for the SPH mesh free particle method. Journal of Computational Physics 2008; 18: 8417–8436.

[2] Khayyer A, Gotoh H, Shao S. Corrected incompressible SPH method for accurate water-surface tracking in breaking waves. Coastal Engineering 2008; 55: 236–250.

[3] Khayyer A, Gotoh H, Shao S. Enhanced predictions of wave impact pressure by improved incompressible SPH methods. Applied Ocean Research 2009; 31: 111-131.

[4] Kondo M, Koshizuka S. Improvement of stability in moving particle semi-implicit method. Int. J. Numer. Meth. Fluids 2010 (in press).

[5] Tanaka M, Masunaga T. Stabilization and smoothing of pressure in MPS method by Quasi-Compressibility. Journal of Computational Physics 2010; 229: 4279-4290.

[6] Asai M, Aly Adbelraheem M., Sonoda Y and Sakai Y. A stabilized incompressible SPH

method by relaxing the density invariance condition. Journal of Applied Mathematics,Vol. 2012, Article ID 139583.

[7] J.J. Monaghan, Simulating Free Surface Flows with SPH, Journal of Computational Physics, Vol. 110, pp.399-406,1994.

[8] Nakao H, Itonaga W, Matsuda R, Izuno K, Kobayashi H. Study on hydrodynamic force on bridge sections subjected to tsunami, *Journal of Japan Society of Civil Engineers*, A2, Vol.67 (2011);PI_481-I_491.

[9] Koshizuka: particle method, Maruzen