Numerical evaluation of fluid force acted on bridge girders during tsunami by using particle method

*** Shoichi Tanabe[1], Mitsuteru Asai[2], and Yoshimi Sonoda[2]**

[1]Department of Civil Engineering, University of Kyushu, JAPAN.
[2] Department of Civil Engineering, University of Kyushu, JAPAN.

*Corresponding author: asai@doc.kyushu-u.ac.jp

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 the fixed ghost boundary method is developed in the model having step-shaped incompatible boundary surface. Finally, the accuracy and efficiencies of our proposed method are validated by comparison between a numerical solution and experimental results.

Keywords: *SPH, Fluid Force, Boundary Condition, Tsunami*.

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. In order to construct safe and secure coastal structures, present tsunami disaster prevention and mitigation methods are necessary to reconsider and to develop an accurate prediction tool toward the next millennium Tsunami. At present, 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 for this purpose are almost impossible and too costly. In this study, focusing on the numerical evaluation of fluid forces acted on bridge girders, a reasonable numerical simulation technique based on the Incompressible Smoothed Particle Hydrodynamics (ISPH) has been selected. 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 the fixed ghost boundary method.

The source term in pressure Poisson equation (PPE) for ISPH is not unique. It has several formulations in the literature as Lee *et al*. (2008) and Khayyer *et al*. (2008,2009). 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 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 (2010) and Tanaka and Masunaga (2010). 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 with fixed ghost boundary method using a virtual marker. A new boundary treatment using the fixed ghost boundary method is proposed to satisfy the slip and no-slip boundary conditions on the solid boundary surface, which is modeled by step-shaped incompatible boundary surface. The accuracy and efficiencies of our proposed method are validated by comparison between a numerical solution and experimental results.

After the validation, an ISPH method with a modified source term and a new boundary treatment are utilized to estimate the fluid force acted on bridge girders in a real scale model, which has a complicated shape. The estimation value is discussed in a couple of cases by differences as to the position of the girders.

Improved ISPH

In this section, a stabilized ISPH (Asai *et al*. 2012), which includes a modified source term in the pressure Poisson equation, for incompressible flow is summarized.

Governing equation

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

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

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

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 $\boldsymbol{\tau}$ 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.

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. That is, the density invariant condition is not satisfied only if the number of nearest neighboring particles is fixed and the uniform distribution of the particles is kept perfectly. Therefore, developing the scheme like that the errors don't occur in the long term is needed by allowing some density errors in a moment. 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.

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

Then, the pressure Poisson equation reformulated as:

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

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.

Boundary treatment

In this section, a new boundary treatment using a virtual marker in incompatible step shaped boundary model 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.

Procedure of proposed boundary Treatment

Wall particle is placed on a grid-like structure with equally spaced in a solid boundary. Virtual marker is put in a position which is symmetrical to the wall particle across its solid boundary. 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.

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

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 ($\tilde{\boldsymbol{W}}$) 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 because the marker is created only once at the preprocess assuming that the wall particle is fixed.

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 ($\boldsymbol{v}_v \to \boldsymbol{v}_w'$) is given by the following equation.

$$\boldsymbol{v}_w' = \boldsymbol{M}\boldsymbol{v}_v \tag{8}$$

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.

$$M_{ij} = \delta_{ij} - 2n_i n_j \tag{9}$$

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.(8).

$$\boldsymbol{v}_w' = \boldsymbol{R}\boldsymbol{v}_v \quad , \quad R_{ij} = -\delta_{ij} \tag{10}$$

Fig.1 shows the examples of velocity vectors, which should be given the wall particles to satisfy slip or no-slip conditions.

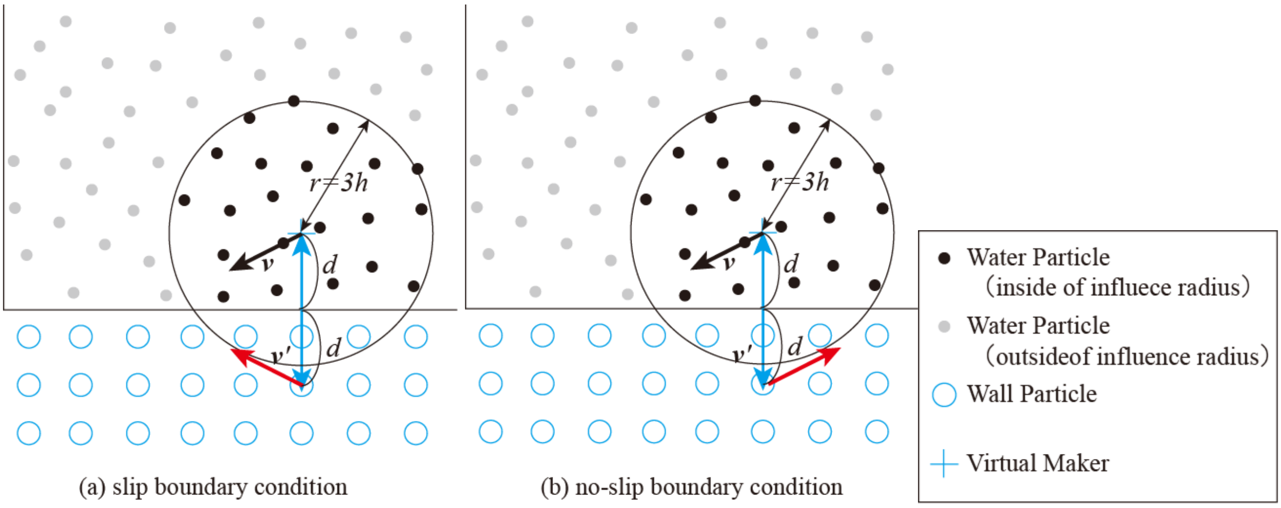

Fig.1 Virtual marker for the slip and no-slip boundary conditions

For the purpose of satisfying the pressure Neumann condition, giving the wall particle accurate pressure is necessary by referring to the one on the marker. Since normal component of the velocity on the solid boundary is equal to zero, the following equation needs to be satisfied.

$$\boldsymbol{v}_{w0} \cdot \boldsymbol{n} = 0 \tag{11}$$

where $\boldsymbol{v}_{w0}$ is the velocity of the solid boundary. Referring to the Navier-Stokes equation (Eq.(12)), which describes the same one as Eq. (2), the next non-uniform pressure Neumann condition needs to be satisfied to complete Eq. (11).

$$\frac{D\boldsymbol{u}}{Dt} = -\frac{1}{\rho^0}\nabla P + \underbrace{\nu\nabla^2\boldsymbol{u} + \boldsymbol{g}}_{f} \tag{12}$$

$$\partial p/\partial \boldsymbol{n} = \rho\ \boldsymbol{f} \cdot \boldsymbol{n} \tag{13}$$

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.

$$p_w' = \langle p_v \rangle + 2d\rho \langle \boldsymbol{f}_v \rangle \cdot \boldsymbol{n} \tag{14}$$

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 ρ describes a water density respectively. Triangle bracket < > means SPH approximation.

Results and discussion

In the following section, the comparison between a numerical solution and experimental results has been introduced to validate the proposed scheme. The relation between the difference of the boundary conditions and evaluation of fluid impact force are also investigated. Next, the degree of the fluid impact force acted on a real size and shape of bridge girder is estimated.

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*. (2011), and the fluid impact force is evaluated while the wave acts on the girder model. The shape of the girder model is upside down trapezoid. In this study, the numerical simulation is conducted by using the proposed boundary treatment described in section 3 and the particle distance $d_0 = 0.5$cm, time increment $\Delta t = 0.001$s and the total number of particles is about 8millions.

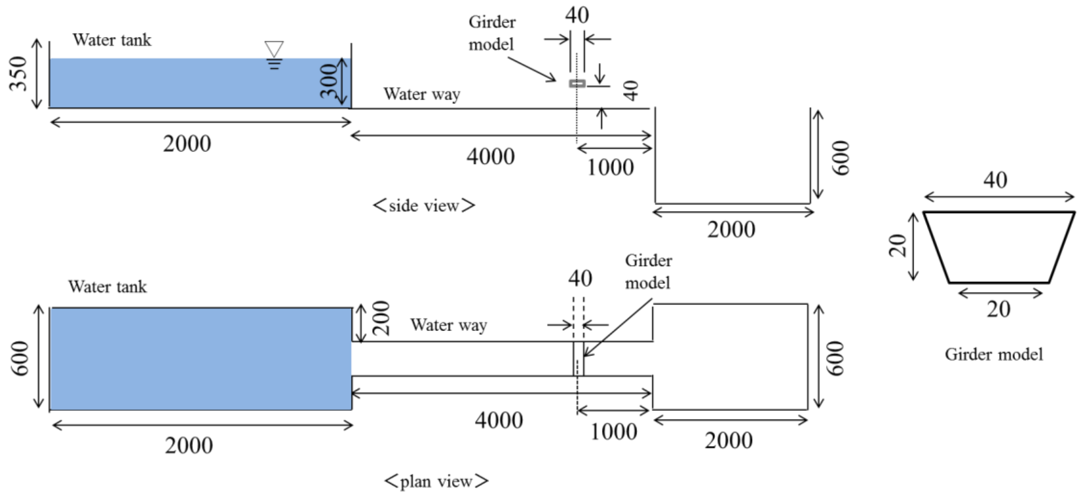

Fig.2 Analysis model (unit: mm)

Analysis results

Fig.3 shows the result of horizontal and vertical force in upside down trapezoid girder model by filtering to cut a component of more than 15 Hz as with the experiment. The chart (a) is given the slip condition and (b) is given the no-slip condition respectively. So far, in the conventional method, the fluid force in the model having complicated boundaries could not be validated sufficiently because the evaluation has been only 2 dimensional or quasi-2 dimensional. Also, the decline of the force which originates from the penetration of water particle into the solid boundary is occurred. However, according to the result shown in Fig.3, this simulation applying our proposed method is confirmed to be useful and this simulation result matches the experimental one at the practical level. In short, the application of our proposed method to the model having incompatible step-shaped boundary is utilized with high accuracy for the purpose of evaluating the fluid impact force. As how to give the boundary conditions, it is not concluded which condition is the best for the simulation, but it seems to be middle or somewhere close to the middle of the two conditions. Further discussion is required in this verification.

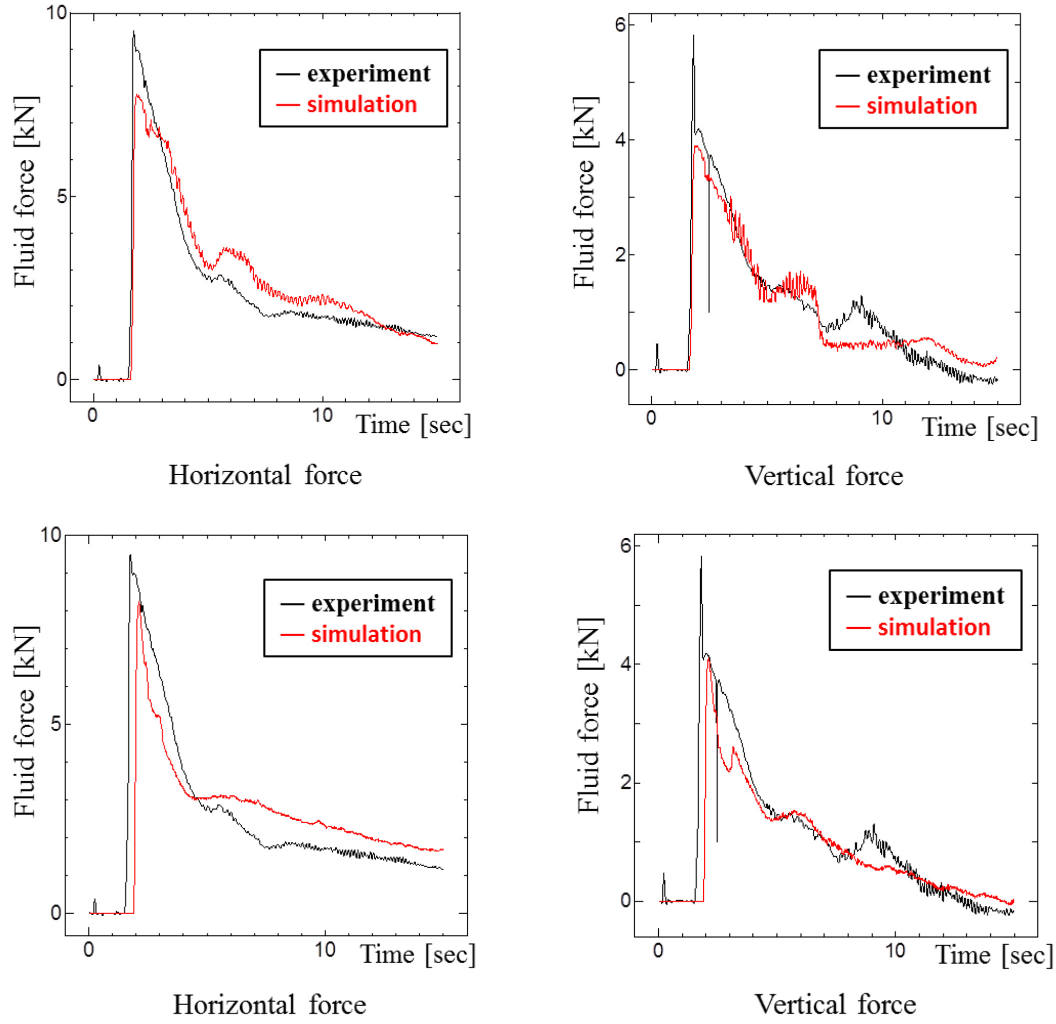

Fig.5 Horizontal and vertical force in upside down trapezoid model

Application for real sized and shape of girders

In this chapter, the fluid impact force acting on a real scale and shape girder model is estimated. The wave is modeled for a gentle stream, and the girders are modeled to three patterns to investigate the change of the value of the fluid force on the assumption that the girders are pushed away with rotating by tsunami. One is horizontal and the others are 10 or 30 degree tilted to the ground respectively. Initial water level is set to be 10m referring to the report that water levels in many disaster cites reached over 10m in the huge tsunami caused by the great east Japan earthquake. The initial velocity of the wave is set 10m/s referring to shallow water long-wave equation assuming that the water level is 10m. Fig.4 shows the analysis model and the detail of the girder models are shown in Fig.5. The depth in the model is 4m. The particle distance d_0 = 12.5cm, time increment Δt = 0.001s and the total number of particles is about 10 millions.

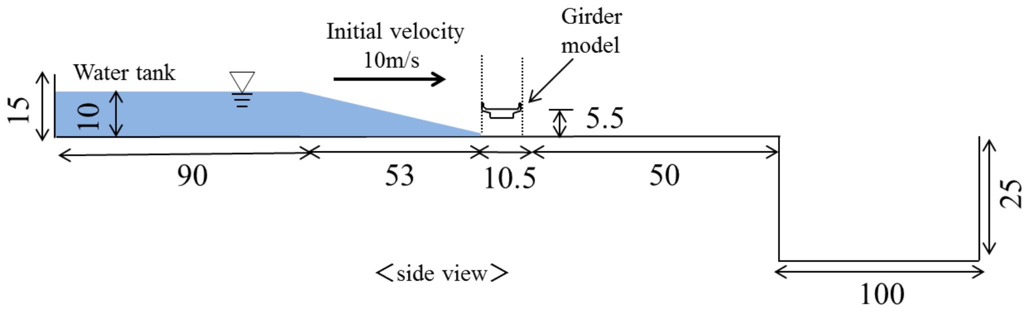

Fig.4 real size and shape of girder model

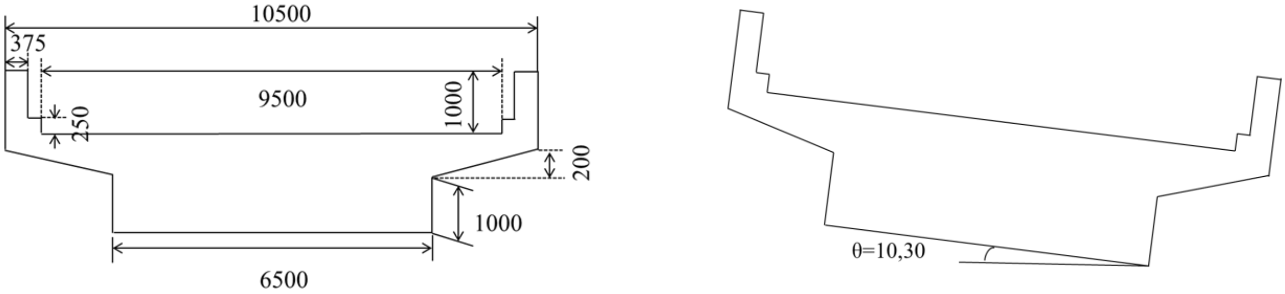

Fig.5 girder models

Result and discussion

Fig.6 shows the results for horizontal and vertical forces acting on the bridge girder model in this simulation. The predicted horizontal force shows a trend that the fluid force increases as the girder is more tilted. On the other hand, the vertical force shows the largest value in 10 degree tilted model. According to these results, it shows that the girder is lifted by the first vertical impact force and then pushed away by the horizontal force. From this result, it can be shown that the lift force is important factor to prevent bridges from tsunami. During the past tsunami disasters, many bridge girders may be pushed away mainly because of the lift force (vertical force) acted on the girders.

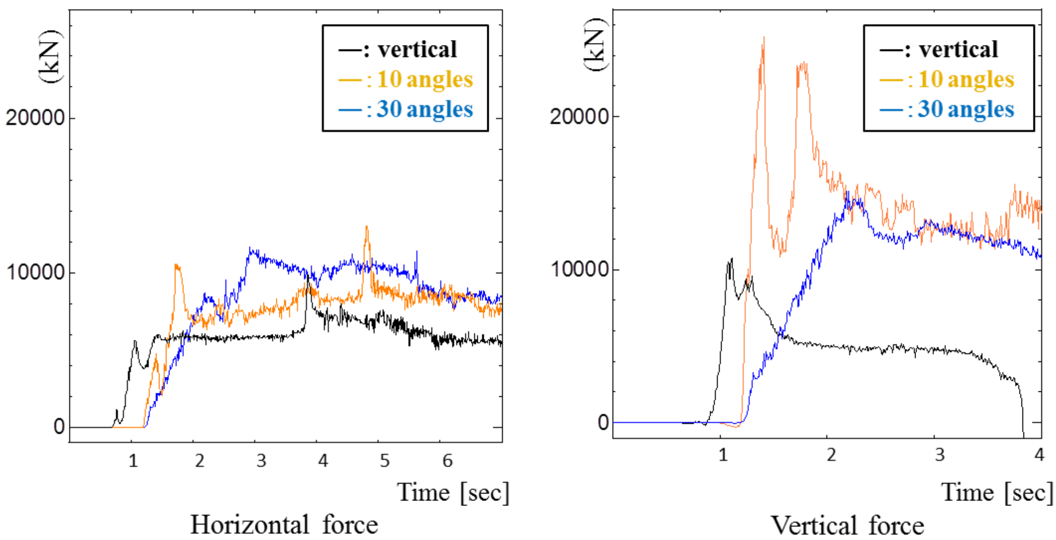

Fig.6 Horizontal and Vertical force in real and shape of girder model

Conclusion

A stabilized incompressible smoothed particle hydrodynamics is proposed to simulate free surface flow. 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 solve an incompatible step-shaped boundary model. The accuracy and efficiency of our proposed method are validated by comparison between a numerical solution and experimental results. From our numerical test, the proposed method can handle the model having a complicated boundary with high accuracy. Finally, the estimation of fluid impact force in a real size and shape girder model is performed by applying the proposed method. This simulation results can show the trend of change of the fluid force value in this model, and also show that the lift force acted on girders is important factor to prevent them from tsunami.

In the future work, a movable girder model as a rigid body should be developed to investigate the process that girder is pushed away, and it may need to discuss the relation between the fluid force and the shape of the girders and waves.

Acknowledgements

Part of the results in obtained by using the K computer at RIKEN Advanced Institute for Computational Science (Proposal number hp130015), and this work is partially supported by JSPS Grant-in-Aid for Young Scientists (B); Grant Number 24760365.

References

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.

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

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

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

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

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.