The 2013 World Congress on
Advances in Structural Engineering and Mechanics (ASEM13)
Jeju, Korea, September 8-12, 2013

A large scale tsunami run-up simulation and numerical evaluation of fluid force during tsunami by using a particle method

*Mitsuteru Asai[1], Shoichi Tanabe[2] and Masaharu Isshiki[3]

[1], [2] *Department of Civil Engineering, University of Kyushu, Fukuoka, JAPAN*
[3] *Graduate School of Science and Engineering, Ehime University, Matsuyama, JAPAN*
[1] *asai@doc.kyushu-u.ac.jp*

ABSTRACT

In this study, the incompressible Smoothed Particle Hydrodynamics (ISPH) is utilized as a 3D tsunami run-up simulator. Firstly, a modeling tool for ground level including structure and sea floor was developed to generate simulation model. The geometrical modeling utilized aerosurvey data for the ground level and bathymetry data for sea floor. A 50 million particles model for a small city was generated by our developed modeling tool for the particle simulations. The numerical solutions were compared with a disaster investigation report. The accuracy of fluid force evaluated by the ISPH has been investigated by comparing with an experimental data.

1. INTRODUCTION

The huge tsunami caused by the Great Tohoku Earthquake devastated the coastal area on March 11, 2011. After the Tohoku Earthquake, the Japanese government and each local government has been discussing the next disaster prevention and mitigation method against millennium Tsunami. In order to construct safe and secure coastal structures, it is necessary to generate an accurate simulator, which can predict not only the flood area but also the structural damage. Therefore, we are developing a 3D tsunami simulator based on a particle method.

In this study, the incompressible Smoothed Particle Hydrodynamics (ISPH) is utilized as a 3D tsunami run-up simulator. Firstly, a modeling tool for ground level including structure and sea floor was developed to generate simulation model. The geometrical modeling utilized aerosurvey data for the ground level and bathymetry data for sea floor. The numerical solutions were compared with a disaster investigation report.

In the next step, the accuracy of fluid force evaluated by the ISPH has been investigated by comparing with an experimental data. In this comparison, the boundary treatment of the particle simulation has been discussed to get accurate solutions.

---

[1] Associate Professor
[2] Graduate Student

2. A stabilized ISPH

In this section, a stabilized ISPH (Asai 2012), which includes a modified source term in the pressure Poisson equation, for incompressible flow is summarized. Then, a conventional turbulence model 'Smagorinsky model' is introduced into the ISPH.

2.1 Governing equations

The mass and momentum equations of the flows are given 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. In the most general incompressible flow approach, the density is assumed by a constant value with its initial value ρ^0.

2.2 Modification in the source term of pressure Poisson equation

The main concept in an incompressible SPH method is solving a discretized pressure Poisson equation at every time step to evaluate the pressure value implicitly. In the sense of physical and theoretical observation, 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.

2.3 Introduction of an eddy viscosity by Smagorinsky model

A simple Large Eddy Simulation has been implemented by introducing the Smagorinsky sub-grid model. The sub-grid stress tensor is modeled through the traditional Boussinesq eddy viscosity assumption as

$$\frac{\tau_{IJ}}{\rho} = 2\nu^T S_{IJ} - \frac{2}{3}k\delta_{IJ} \tag{5}$$

$$\nu^T = \left(C_s\Delta\right)^2 |\boldsymbol{S}| \tag{6}$$

in which ν^T in the Smagorinsky model is an eddy viscosity, C_s is Smagorinsky constant, Δ is constant and it taken as smoothing compact support in this scheme. The local strain rate $|\boldsymbol{S}|$ is calculated as (Violeau 2007). The turbulent kinetic energy incorporated in the pressure term, and then the current viscous term is given by SPH assumption as follows:

$$< \nabla \cdot (\nu \nabla \cdot \boldsymbol{u}_i) >= \sum_j m_j \left(\frac{\rho_i \nu_i + \rho_j \nu_j}{\rho_i \rho_j} \frac{r_{ij} \nabla W}{r_{ij}^2 + \eta^2} \right) \boldsymbol{u}_{ij} \tag{7}$$

The numerical examples have illustrated that the proposed finite elements could be very useful for geometrically nonlinear analysis as well as free vibration …

3. Tsunami Simulation

By using the stabilized ISPH tool, a tsunami simulation has been conducted with a realistic geometrical model in 3D.

3.1 Tsunami simulation at Taro

In this simulation, tsunami run-up behavior is simulated with the real geometrical map generated from a multi-resolution aerial survey data. The tsunami is inputted on the right boundary edge in our model shown in Fig.1, and the input of tsunami is assumed from the investigation of Tohoku earthquake. The height is 3m, and its velocity is 10m/s.

The constant tsunami is given during our tsunami simulation for 5 minutes in the real time. Fig 2 shows the snapshot of tsunami simulation after 20s, 2m20s, and 4m20s. In the figures, contour color indicates the velocity magnitude in the horizontal direction of the figure. These run-up behaviors show a good agreement with the damage investigation results.

The computation was mainly carried out using the facilities at Research Institute for Information Technology at Kyushu University. It spends about 3days for the 5minutes behaviors.

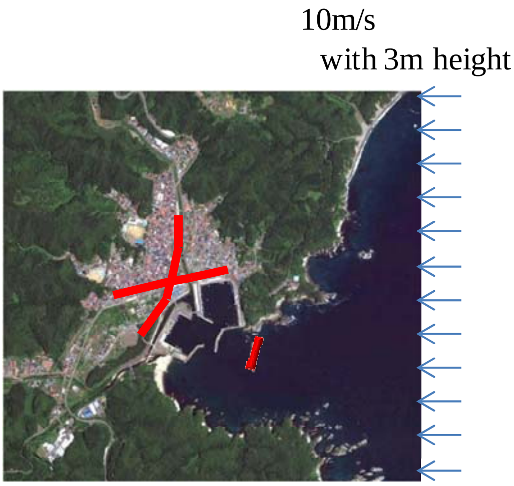


Fig. 1 Taro city and tsnami inflow condition

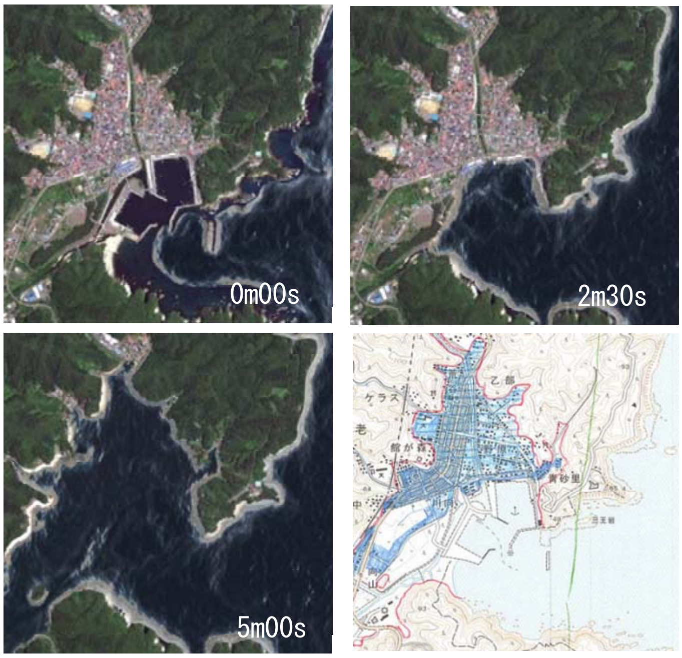


Fig. 2 Taro city and tsnami inflow condition

4. Validation of fluid impact force by comparing with an experimental test

In this section, the accuracy and efficiencies of our proposed method are validated by comparison between a numerical solution and experimental results. The necessity of our proposed boundary treatment, which is described in previous section, is discussed at the same time, and 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 real size and shape of bridge girders is estimated.

4.1 Details of the validation test

The analysis model and the detail of the model for girders are shown in Fig.3 and Fig.4 respectively. This experiment was carried out by Nakao et al. (2011), and the fluid impact force is evaluated while the wave acts on the model of girders. The shape of the model for girders is rectangle and upside down trapezoid. The particle distance d0= 0.5cm, time increment dt= 0.001s and the total number of particles is about 8millions.

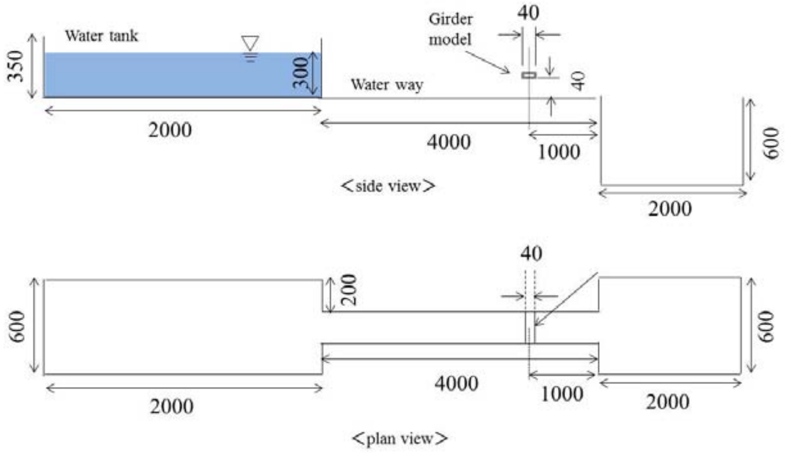


Fig. 3 Validation model (unit: mm)

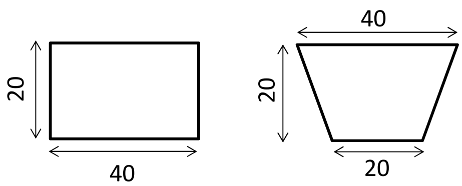
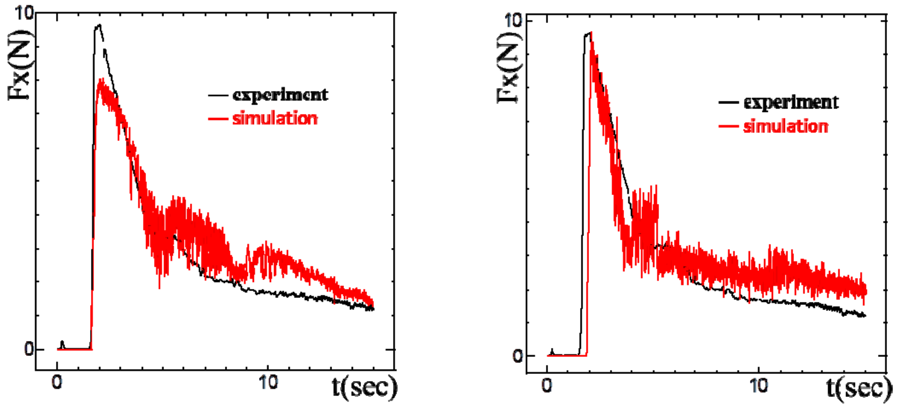


Fig. 4 Detail of the Bridge girder models

4.2 Validation results

At fast, Fig.5 shows the result of horizontal force in rectangle model and (a) is given the slip condition and (b) is given the noslip condition respectively. So far, in the conventional method, the accuracy of fluid impact force could not be validated sufficiently because of the problem such as the decline of maximum force, which originates from penetration of water particle into the solid boundary.

According to the result shown in Fig.5 this simulation applying our proposed method is confirmed to be useful and this simulation result matches the experimental one at the practical level. In addition, from the detail of the result, the maximum force given the noslip condition is closer to the experimental result. This reason may be concerned with the tip shape of the wave. In the slip condition, the tip shape of the wave is more acute angle than the experiment, on the other hand, the one in the no-sip is rounded, and since the latter wave attacks the girder model as a mass of water, it is considered that the maximum force given the noslip condition is more accurate to the experiment.



(a) Horizontal force (slip condition) (b) Horizontal force (no-slip condition)

Fig. 5 Horizontal force in rectangle model

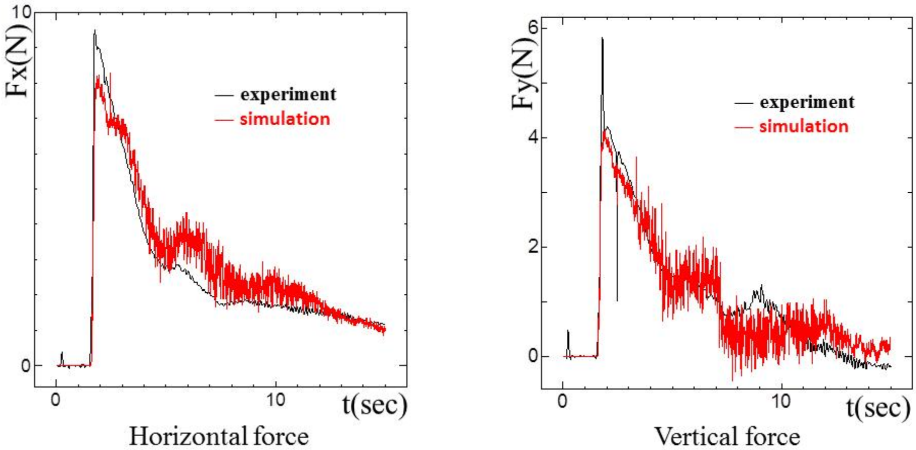
From the result of the moment to attack the girder model, the moment in the slip condition almost matches the experimental result. This may be related to the velocity (Table.1) and shape of the wave. Referring to Table.1, the wave velocity in the slip condition is faster than the experimental one, so it is thought that the moment to attack the girder model is also faster, but the result is almost equal to the experiment. It can be because that the increase of water level is delayed by the acute-angled shape of the wave.

In the no-slip condition, the velocity of the wave is slower than the experimental one, because it becomes a flow as the friction acts on the boundary and the waterway is long enough to cause the effect of that friction.

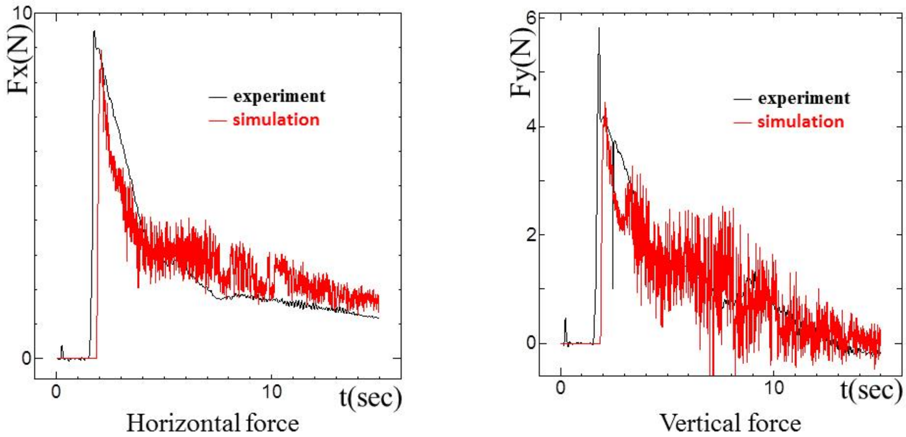
Table.1 Comparison of the wave velocity

The velocity of wave (unit: m/s)	
Experiment	2.2
Simulation (slip condition)	2.9
Simulation (no-slip condition)	1.7

Next, Fig.6 shows the result of horizontal and vertical force in upside down trapezoid model and (a) is given the slip condition and (b) is given the no-slip condition respectively. Referring to Fig.11, the trend similar to the rectangle model is obtained, that is, the maximum fluid impact force in the no-slip condition is closer to the experiment and the moment to attack the model for girders in the slip condition almost matches the experimental result. From the above results, the accuracy and efficiencies of our proposed method were validated by comparison between a numerical solution and experimental results. It showed that the application of our proposed method to the model having incompatible step-shaped boundary is also utilized in a practical level to evaluate 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.



(a) Slip condition



(b) No-slip condition

Fig. 6 Horizontal and vertical force in upside down trapezoid model

4. Validation of fluid impact force by comparing with an experimental test

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. The ISPH has been applied to a tsunami run-up simulation with a high performance computation, and then the accuracy of the proposed model has been verified with an experimental test for the estimation of tsunami wave force. In the validation test, treatment of the boundary condition has been carefully investigated to get accurate solution especially for estimating the fluid impact force.

REFERENCES

Asai M, Aly Adbelraheem M., Sonoda Y and Sakai Y. (2012) A stabilized incompressible SPH method by relaxing the density invariance condition. Journal of Applied Mathematics,Vol.2012, Article ID 139583.

Violeau D and Issa R. (2007) Numerical modeling of complex turbulent free-surface flows with the SPH method: an overview, International Journal for Numerical Methods in Fluids, 53(2), pp.277-304.Cadappa, D.C., Sanjayan, J.G. and Setunge, S. (2001), “Complete triaxial stress-strain curves of high-strength concrete,” *J. Mat. Civil Eng., ASCE,* **13**(3), 209-215.