SIMULATION OF
THE COLLAPSE PROCESS OF INFRASTRUCTURE
USING GENERAL-PURPOSE PHYSICS ENGINE

Yoshihiro Kabeyama[1], Sunao Fujimura[2] and Shinichiro Okazaki[3]

[1,2]Graduate School of Engineering, Kagawa University, Japan;
[3]Faculty of Engineering and Design, Kagawa University, Japan;

ABSTRACT: In recent years, tsunamis caused by earthquakes and flooding associated with heavy rainfall have occurred in Japan. It has often been reported that bridges collapsed because of the outflow of the bridge superstructure and the scouring of the bridge pier resulting from these causes. However, to simulate these phenomena, it is necessary to carry out a coupled analysis of three types of behavior of the structure, ground, and fluid, and the calculation cost is very large. Therefore, the examination of this phenomenon has not currently achieved much progress. The purpose of this research was to perform a fluid–ground–structure simulation (such as for the collapse of a bridge pier because of flooding) at low cost, using a general-purpose physics engine. In this study, the focus was on the simulation of large deformations of the fluid, ground, and structure, and all objects were modeled with particles. First, the parameters that can simulate the behavior of the fluid were set appropriately by using the dam break problem. Subsequently, the flooding phenomenon in a river and the scouring phenomenon on a bridge pier on the ground were simulated using these parameters.

Keywords: Physics simulator, Scouring phenomenon, SPH method

1. INTRODUCTION

In recent years, damage such as deep deformation of the lower structure has frequently occurred in Japan, along with the damage of the upper structure of river bridges due to floods and tsunamis [1]. Heavy rains in the western Japan region that occurred in July 2018 caused damage to many bridges in the Japan Railway Shikoku area. This forced the trains to be shut down and affected many users. In the case of the Saita River Bridge between the Yosan Line Motoyama and Kannonji Temple, the bridge pier sloped because of the scouring, causing the deformation of the superstructure. The Saita River Bridge is built on a river that usually has a small amount of water. However, on the day of heavy rains, the water flow was higher than expected. As a result, the ground was scoured by the river flow, and the bridge pier was inclined. This kind of damage has been experienced frequently since ancient times, and it is thought that it is possible to avoid damage by studying disaster cases. In recent years, with the development of numerical calculators, the study of disaster risk by machine learning has been carried out, and it is currently possible to extract a pier having a high risk of scouring with high accuracy. However, the construction of a more accurate forecasting model requires a high-level understanding of the scouring phenomenon, and studies considering changes in the varied topography of the river, ground conditions, and the river flow are necessary. The method of numerical calculation makes it possible on a computer. To simulate these phenomena, it is necessary to couple the three-phase behavior of the fluid–ground–structure, and the time and load required for modeling and calculation are very large. The examination of this phenomenon using numerical analysis has not significantly progressed, especially in three dimensions. Handling river flow is very important and plays an important role in revetment work and the design of the bridge.

In particular, it is difficult for the finite-element method (FEM) to reproduce complicatedly changing free surfaces in fluid simulation. [2]-[5] Also, it is necessary to reproduce the fluctuation of the riverbed caused by the river flow scooping the ground. Moreover, in this research, more-complicated analysis is required, because the calculation data for the bridge are included. Although three-dimensional software that can easily simulate such a large deformation has been developed, it is expensive. Therefore, in this research, an attempt was made to solve the soil-fluid-structure interaction problem, which has been difficult to study because of the large computational load, at low cost and in a relatively short time by using a physical simulator that can be used free of charge and has high versatility.

2. SIMULATION METHOD

2.1 Software

In this study, a general-purpose physics simulator

(Blender) was used. Blender is equipped with a physics engine called Bullet, and this software can simulate an object from a rigid body to a soft body as a discrete body or a continuum. In addition, it supports optimization to the Graphics Processing Unit (GPU) using Compute unified device architecture (CUDA) and can perform three-dimensional simulation at high speed [6]. This simulator is based on the smoothed-particle hydrodynamics (SPH) method and can simulate large deformation of both fluids and solids. However, the setting of the parameters that govern the behavior between particles is very different from general numerical analysis, such as the FEM. Therefore, to perform accurate simulation, it is necessary to confirm the validity of the behavior of the fluid, which is originally a continuum, as a connected particle group, and it was examined by comparison with the existing experimental results when simulating.

3. REPRESENTATION TECHNIQUE OF MODEL

3.1 Particle and Grid Methods for Rigid-Body Simulations

When simulating a rigid body, it is common to use polygons to calculate and express accurately. However, as the shape becomes more complex, the amount of information processed by the model becomes larger, and complex calculations are required for continuum collision calculation. In the process of calculating such complicated behavior, the deformation of the mesh may break the calculation. Therefore, by using particles instead of polygons for rigid-body representation, mesh problems do not occur, because the interaction between particles is calculated. It is also possible to reduce the cost of computation by limiting the range of interaction. By this means, physical simulation close to reality is performed at high speed [7].
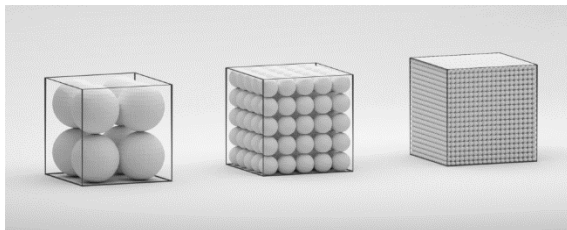

Fig. 1. Structure of cubes

3.2 SPH Method

In this research, the aim was to solve the ground-fluid-structure coupled problem of a slope by scouring of the bridge by creating a fluid model, soil particle model, and bridge pier model using the SPH method.

The SPH method is a numerical analysis method and is classified as a meshless method. Among the meshless methods, the SPH method can replace the behavior of a continuum with the motion of a discrete body by expressing the continuum with a finite number of particles by the Lagrange method. The SPH method is suitable for problems with large-scale changes in objects, and it is used in such fields as structural mechanics and fluid dynamics [8], [9]. In general, calculation of simulation by a continuum requires enormous time, and it frequently cannot be calculated by a general-purpose PC. The method of using discrete fields is possible without incurring much time. However, the calculation accuracy of the SPH method itself, and the handling of boundary conditions, are not well established. Further, it is difficult to reproduce the fluid by discrete bodies; hence, it is not performed very often. It is necessary to verify the validity by combining an infinite number of parameters (with the result of the element experiment in each component), and the experimental result in which the interaction between the components is considered, considering the setting of parameters, etc.

3.3 Particle System

Blender is equipped with Particle System. Particle System is a computer graphics technology, and very small objects called "particles" can represent irregularly shaped materials that exist in the natural world. By placing an object called Emitter in Blender, one can release and control particles from there. Also, particles can be influenced by gravity and other external forces, and the effects of interference of different particles can also be expressed. In other words, it is considered that the simulation of scouring can be expressed by interacting different fluid and soil particles represented by particles.

4. PARTICLE MODEL SETTING

Blender particles have various parameter settings. The Bullet installed in Blender has features that allow parallel computation by a CPU or GPU, and it can solve various physical phenomena very fast. However, the code is a black box, and viscosity, stiffness, and mass parameters governing particle motion are unitless, so it is impossible to divert actual fluid and solid parameters as they are. Therefore, in the case of fluid simulation, for example, it is necessary to set parameters that can simulate this behavior appropriately after conducting element experiments to understand the characteristic behavior of the fluid.

In this research, the focus is on the behavior of the fluid among the phrases that describe the scour phenomenon first, and the fluid-related parameters are set to be able to reproduce the dam breaking

problem with high accuracy. Subsequently, a model was constructed that can reproduce the behavior of soil particles under the action of fluid appropriately.

5.   SIMULATION RESULT

5.1 Dam Break

5.1.1 Simulation method

The dam break problem is often used to verify the validity of a model that describes the free surface flow field. In this study, the fluid was represented by particles, and the experimental device was created by a mesh frame. The scale of the model to be created was $50 \times 100 \times 60$ mm, as shown in Fig. 2. Then, the fluid was set to a portion 30 mm from the bottom in the model, the weir was removed at once, and the flow was tracked. This frame ignores the friction with the particle. In the particle parameter settings, the initial settings were set with reference to books or the Internet site of simulation [10]-[12]. Above all, size

and rest density had a large influence on the interaction range and fluid behavior, and other parameters were set based on these. Figure 3 shows the simulated results for each set value of each parameter. The validity of this simulation was

confirmed by image comparison with experimental fluid behavior. Table 1 shows the values that are
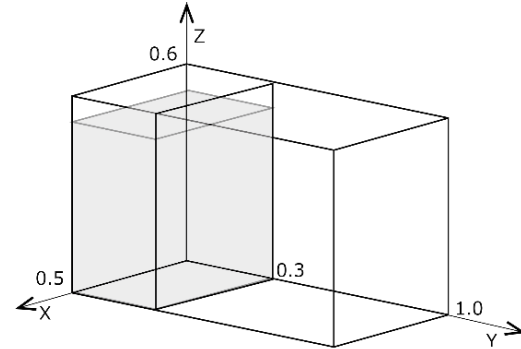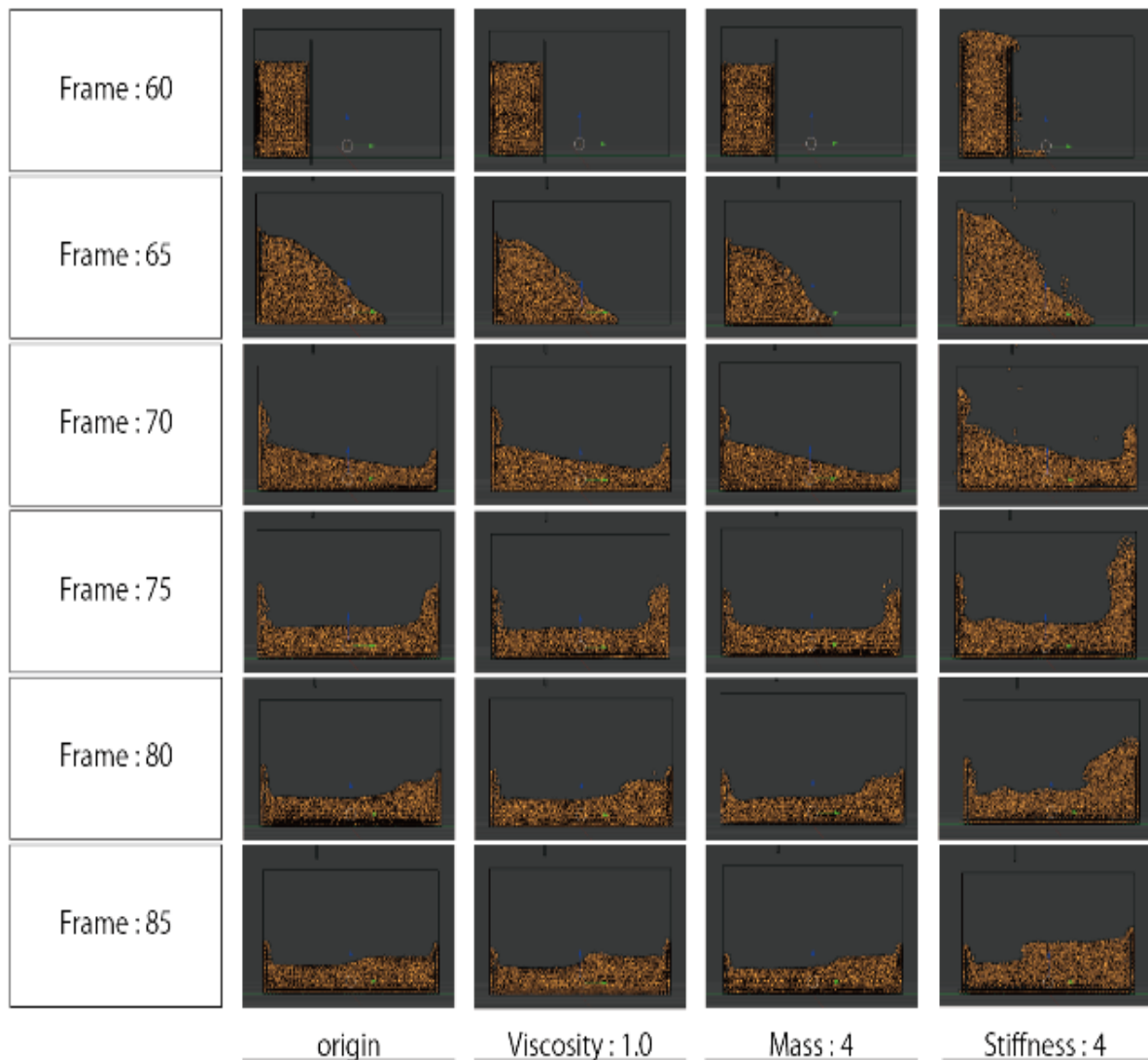

Fig. 2.   Mesh frame model

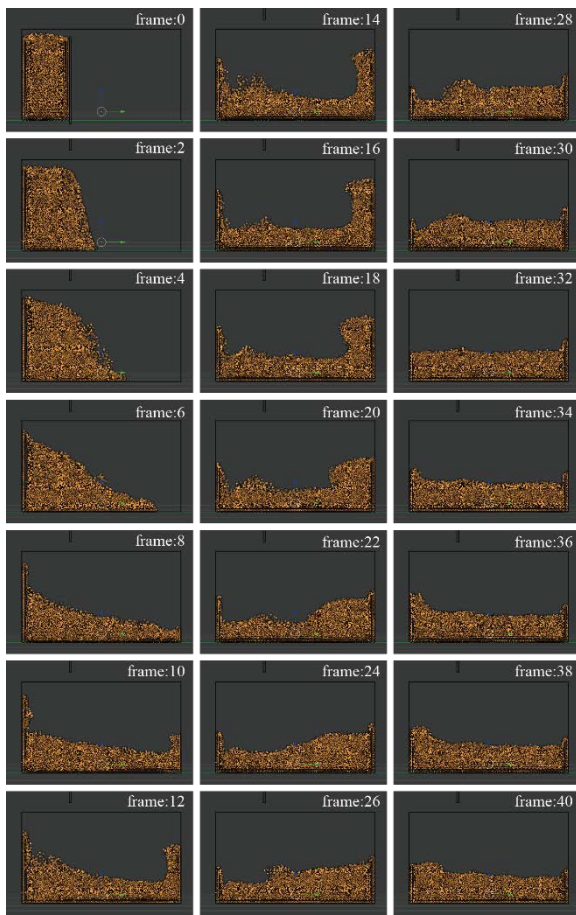Fig. 3.   Comparing dam break simulation result with parameter

Fig. 4.   Result of dam break simulation

closest to the experimental fluid behavior as a study of the parameter value.

The simulated result of the dam break using this setting is shown in Table 1 and particle moving is shown in Fig. 4. At frame 0, the stored fluid drains by removing the weir. At frame 8, The fluid collides with the opposite wall and bounces up. Then the fluid bounces back and is approaching equilibrium. This simulation of the frame rate is 24 fps.

Table 1. Setting of fluid

Fluid	
Particles	7000
Size	0.011
Rest Density	0.5
Mass	4.0
Stiffness	4.0
Viscosity	1.4

5.1.2 Simulation results

In Fig. 5, the left side of the figure is the experimental behavior of the fluid by Koshizuka and Oka [3], and to the right is the result of the simulation by Blender. Comparing these shows that the fluid did

not bound significantly at 0.6 s, but at other times it showed almost the same behavior as the experiments. This value was applied in the following simulations.

5.2 Collapse Simulation

5.2.1 Simulation method

Infiltration, overflow, erosion, earthquakes, etc. can be mentioned as factors that cause the dike and bridge of the river to be destroyed. The scouring phenomenon is included in the erosion. Scouring is a phenomenon in which local erosion, which did not occur in steady-state water flow, happens when the river flow increases because of floods associated with heavy rainfall and the water flow is disrupted. Scouring makes the superstructure unstable because of the settlement and outflow of the foundation part. The progress of such scouring may cause the bridge pier to a slope, and even collapse.

In the simulation of the scouring phenomenon, three types of the particle (fluid, foundation, and bridge structure) were set to solve the fluid–ground–structure interaction problem. Here, to express the foundation and the pier, Molecular Script was used as the setting of particle. In Molecular Script, there is a method called activated particle linking that connects particles, and it is possible to determine the strength of the connection (search length) and the number to be connected (max links) as the setting value among them. These settings are shown in Table 2.

Table 2. Settings of understructure and pier

	Foundation	Bridge pier
Search length	1.7	4.0
Max links	2	16
Broken	5.0	100.0
% linking	0.06	0.0

5.2.2 Mesh frame model

The experimental equipment for simulation was created in the same way as a dam break. In this model, the water flow path, the foundation to be the ground, and the bridge pier installed on this foundation was set. In past research, experiments on such conditions were not conducted, so, in consideration of the performance of the PC used (Intel Core i7-7820X CPU @ 3.60 GHz, System x64 base processor, Compute Device: CUDA GeForce GTX 1080 Ti(Display)), a model was set up and solved, as shown in Fig. 6. Figure 6 shows the setting values of the mesh frame.
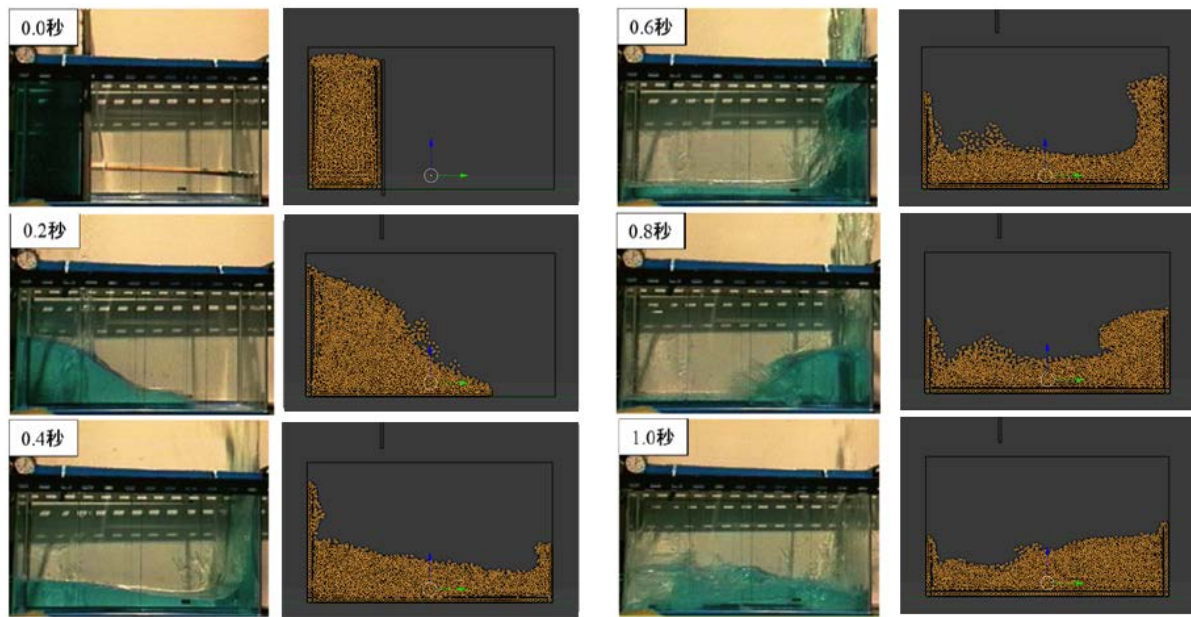
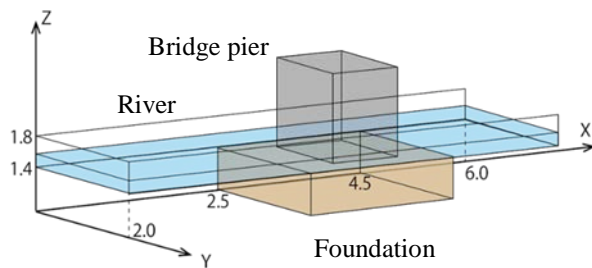Fig. 5. Comparing dam break simulation result with the experiment

Fig. 6. Mesh frame model
Water channel:
Height 0.4 m, width 2.0 m, length 6.0 m
Foundation:
Height 1.4 m, width 2.0 m, length 2.0 m
Bridge pier:
Height 2.0 m, width 1.2 m, length 0.9 m

5.3 Simulation Results

The foundation was excavated by the running water, and it was confirmed that the bridge piers were eventually inclined. In addition, it was confirmed from the calculation process that the collapse of the bridge pier did not stop and became unnatural. Therefore, it was decided to handle new linking as a new setting. With this setting6, the disconnected particles can be rejoined, and the bridge was successfully made to collapse gently. The results are shown in Fig. 7. It was also confirmed that the connection between particles with these settings affects not only the set particles but also other particles. To improve simulation accuracy, it is inevitable to reduce the size and increase the number of particles. In the future, it will be necessary to study the object in a more subdivided model by particles.

6. CONCLUSIONS

In this study, the problem of the scouring of a bridge pier by a flood was addressed using a general-purpose physics simulator. To carry out the simulation at low cost, an attempt was made to solve the fluid–ground–structure three-phase coupled problem using the SPH method. The physics engine Bullet used in Blender does not disclose the physics formula, so it is very difficult to control the parameters used in the computation. An attempt was made to confirm the validity by comparing a dam break with an experiment to reproduce water by particles. Although the characteristic movement of the fluid could be captured partially, it was not possible to reproduce the surface tension of the water and the associated bounce. In the simulation of scouring, three kinds of the particle (fluid, ground foundation, and bridge structure) were treated at the same time. The aim was to solve the fluid–ground–structure interaction problem by this. Here, to express the foundation and the bridge pier, New links was used as the setting of the particles. New links (% linking) can reconnected particles disconnected by the fluid impact. It transpired that New links not only works on the particle that has been activated but also on other particles. Although there was a limit to the segmentation of the model because of the performance of the PC, it was possible to simulate the inclination of the bridge pier by scouring.

As a result, although valid parameters were obtained, when the basic part was subdivided to perform a more accurate simulation, the correct simulation could not be calculated with the same

setting. In other words, the size of the particle and each parameter value are considered to have a strong relationship with each other.
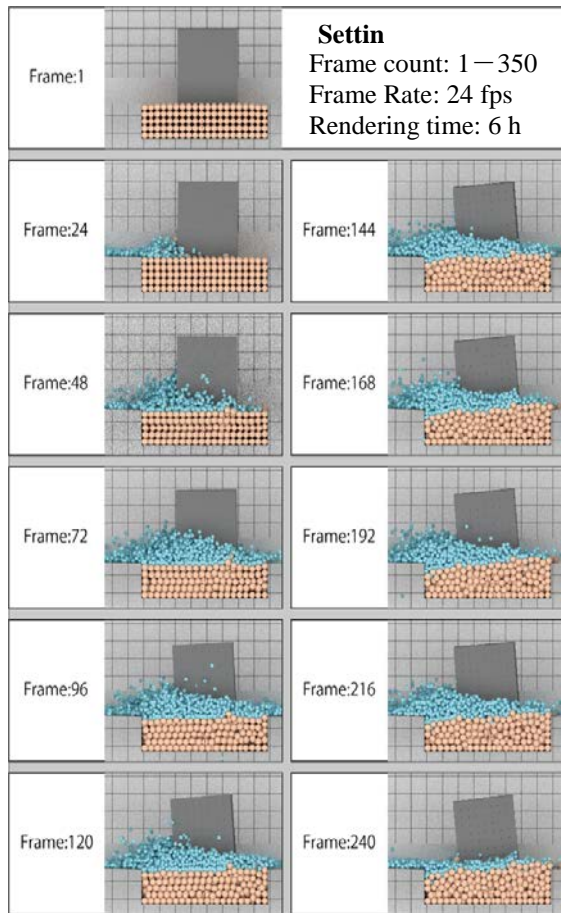

Settin
Frame count: 1－350
Frame Rate: 24 fps
Rendering time: 6 h

Fig. 7. Collapse simulation result

7. ACKNOWLEDGMENTS

8. REFERENCES

[1] Ishino K., and Tamai N., A collapse factor and the measures of the bridge by the heavy rain that occurred in Japan after 2004, Japan Society of Civil Engineers, Vol. 5, 2010.

[2] Goto H., Ikari H., Sakai T., and Oku K., 3D simulation of tsunami flood with floating bodies. Proceedings of Coastal Engineering, Vol.53, 2006, pp. 196-200

[3] Koshizuka S., and Oka Y., Moving-Particle Semi-Implicit Method for Fragmentation of Incompressible Fluid. Nuclear Science and Engineering, Vol.23, 1996, pp.421-434

[4] Suzuki K. Kubota J., and Ohtsubo H., Rigid body simulation Using volume based collision detection. JSCE, Vol.6. 2003. pp.131-139

[5] John Dingliana, Carol O'Sullivan. Graceful Degradation of Collision Handling in Physically Based Animation. COMPUTER GRAPHICS forum. Vol.19, 2000. pp.239-248

[6] Tomihisa Y., PU and GPU performance comparison-using matrix calculation and N-body problem- 2011, pp.1-28.

[7] Tanaka M., Sakai M., and Koshizuka S., Particle-based Rigid Body Simulation and Coupling with Fluid Simulation, Transaction of JSCES, Paper No.20070007, 2007.

[8] Koshizuka S, Numerical Analysis of Flow using Particle Method, Flow21, 2002, pp.230-239.

[9] Tanabe S., Asai M., Miyagawa Y. and IsshikiI M., A fluid－rigid body simulation by using the SPH method and ITS application to brigde run off simulation　SPH, J.JSCE, Ser.A2（Applied mechanics）Vol.70, No.2（JSCE VoL.17）, (2014), pp.329-338.

[10] Todo＋＋，Blender 2.7 Master Book Sculpt & Simulation. Tokyo, Cut system Inc., 2015.6, 459p

[11] Tony Mullen, Physics simulation Blender, Tokyo, ASCII Media Works Inc., 2008, 12. 458p.

[12] Introducing particle fluids parameters, available from <https://farsthary.wordpress.com/2010/03/19/introducing-particle-fluids-parametters-function/>, (accessed 2018-07-22)