VALIDATION OF A NUMERICAL MODEL FOR TSUNAMI WAVES OVERTOPPING A COASTAL EMBANKMENT

Nguyen Xuan Dao[1], Mohammad Bagus Adityawan[2], Hitoshi Tanaka[3]

ABSTRACT

A numerical model based on Reynolds Averaged Navier Stokes (RANS) equations, NumErical Water FLUME called NEWFLUME has been successfully simulated and validated for various types of flows and their interaction with different structures, porous or impermeable, moving or stationary with a short period waves. For a short period waves, the boundary layer is very thin that even the theory of inviscid flow can be applied in predicting and explaining the overall flow pattern. However, in most of sophisticated flow control strategies in engineering applications, we are concerned with the action of viscosity and turbulence. In addition, the flow formed by the more steady current is turbulent over the entire water depth and does not have the same boundary layer characteristics as the thin boundary layer formed by waves. Furthermore, the currently NEWFLUME model has not validated for a long period waves or a steady flow yet. Therefore, this study aims to validate the NEWFLUME model for tsunami waves overtopping a coastal embankment, the simulated results including water level, flow velocity and piezometric head were used to compare with those from hydraulic experiments carried out by Kato et al. (2013). In overall, NEWFLUME model has been found to be a useful tool for evaluating the influences of tsunami overtopping on coastal structures.

1. INTRODUCTION

The recent Great East Japan earthquake with M9 on March 11th, 2011 provided many lessons regarding the huge consequences and potential impact of megaearthquake and megatsunamis. Due to underestimated potential earthquake magnitude M9 generated tsunami off the East coast of Honshu Japan lead to many sea dikes and breakwaters along the country destroyed. Therefore, there is a need for scientist to understand tsunamis and to construct a highly resilient structure which can mitigate the impacts of tsunamis. In the coastal structures design, it is important to understand the wave interaction onshore and offshore structures. Also, wave force induced by tsunami is one of the factors leading to the failure of coastal embankment.

Coastal and port-related structures have been designed based on design formula as well as hydraulic model experiments. Although hydraulic model experiments can precisely reproduce actual physical phenomenon, however, it often requires time and cost to create seabed configurations and model structures, and to measure various kinds of data such as wave height, wave pressure, overtopped water and the movement of targeted structures Hanzawa et al. (2012). In order to investigate more information about the causes of damages from the view point of fluid hydraulics, numerical model is currently a valuable tool to access coastal damages. Numerical studies for solitary wave interaction with porous breakwaters have been performed by Liu and Wen (1997) while Lin and Karunarathna (2007) studied solitary wave interaction with fully emerged rectangular porous breakwaters with different length and particle size. There are several numerical models, for example, a numerical wave flume called CADMAS-SURF (Super Roller Flume for Computer Aided Design of Maritime Structure), Isobe et al. (1999). CADMAS-SURF has been applied mainly to ordinary wave conditions such as wind waves, e.g. wave force on breakwaters and wave overtopping of seawalls Isobe et al. (2002). However, both of NEWFLUME model and CADMAS-SURF model have not validated for a long period wave or steady flow which has fully developed of boundary layer yet. In most of sophisticated flow control strategies in engineering applications, we are concerned with the action of viscosity and turbulence. Therefore, this study aims to validate NEWFLUME model for tsunami waves overtopping a coastal embankment by comparison simulated results to those from hydraulic experiments carried out by Kato et al. (2013).

2. NUMERICAL MODEL

[1] Doctoral student, Department of Civil Engineering, Tohoku University, 6-6-06 Aoba, Sendai 980-8579, Japan
[2] Ph.D., Department of Civil Engineering, Tohoku University, 6-6-06 Aoba, Sendai 980-8579, Japan
[3] Professor, Department of Civil Engineering, Tohoku University, 6-6-06 Aoba, Sendai 980-8579, Japan

2.1 Governing equations

For a turbulent flow, both the velocity field u_i and the pressure field p can be split into mean component and turbulent fluctuations as follows:

$$u_i = \langle u_i \rangle + u_i^{'}; \quad p = \langle p \rangle + p^{'} \tag{1}$$

in which i = 1,2 for a two-dimensional flow direction, $\langle u_i \rangle$ and $\langle p \rangle$ are the mean velocity and the mean pressure, $u_i^{'}$ and $p^{'}$are the turbulent velocity fluctuation and turbulent pressure fluctuation.
If the fluid is assumed to be incompressible, the mean flow field is governed by the Reynolds equations as follows:

$$\frac{\partial \langle u_i \rangle}{\partial x_i} = 0 \tag{2}$$

$$\frac{\partial \langle u_i \rangle}{\partial t} + \langle u_j \rangle \frac{\partial \langle u_i \rangle}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \langle p \rangle}{\partial x_i} + g_i + \frac{1}{\rho}\frac{\partial \langle \tau_{ij} \rangle}{\partial x_j} - \frac{\partial \langle u_i' u_j' \rangle}{\partial x_j} \tag{3}$$

where ρ is the fluid density, g_i is the gravitational acceleration in the i-th direction, $\langle \tau_{ij} \rangle$ is the viscous stress tensor, and $\langle u_i' u_j' \rangle$ the Reynolds stress.

An alternative to the Reynolds stress closure model is the so-called $k-\varepsilon$ model in which the Reynolds stress tensor is assumed to be related to the strain rate of the mean flow through the algebraic nonlinear Reynolds stress model.

$$\langle u_i' u_j' \rangle = -2\upsilon_t \langle \sigma_{ij} \rangle + \frac{2}{3} k \delta_{ij} \tag{4}$$

The governing equations for $k-\varepsilon$ model are as follows:

$$\frac{\partial k}{\partial t} + \langle u_j \rangle \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\frac{\nu_t}{\sigma_k} + \nu\right)\frac{\partial k}{\partial x_j}\right] + \nu_t \left(\frac{\partial \langle u_i \rangle}{\partial x_j} + \frac{\partial \langle u_j \rangle}{\partial x_i}\right)\frac{\partial \langle u_i \rangle}{\partial x_j} - \varepsilon \tag{5}$$

$$\frac{\partial \varepsilon}{\partial t} + \langle u_j \rangle \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\frac{\nu_t}{\sigma_\varepsilon} + \nu\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1c}\frac{\varepsilon}{k}\nu_t \left(\frac{\partial \langle u_i \rangle}{\partial x_j} + \frac{\partial \langle u_j \rangle}{\partial x_i}\right)\frac{\partial \langle u_i \rangle}{\partial x_j} - C_{2c}\frac{\varepsilon^2}{k} \tag{6}$$

in which k is the turbulent kinetic energy ($k = 1/2 \langle u_i' u_i' \rangle$), ε is the dissipation rate of turbulent kinetic energy ($\varepsilon = \nu \langle (\partial u_i' / \partial x_j)^2 \rangle$), ν is the kinematic viscosity, ν_t is the eddy viscosity ($\nu_t = C_d\, k^2/\varepsilon$), σ_k, σ_ε, C_{1c} and C_{2c} are empirical coefficients. The recommended values for these coefficients are as follows (Rodi, 1980):

$$C_d = 0.09;\; C_{1c} = 1.44,;\; C_{2c} = 1.92;\; \sigma_k = 1.0;\; \sigma_\varepsilon = 1.3 \tag{7}$$

2.2 Boundary conditions

Appropriate boundary needs to be specified for the model. For rigid boundary conditions, the values of k and ε are specified in the turbulent instead of right on the wall. They are expressed as functions of distance from the boundary and the mean tangential velocity outside of the viscous sublayer. The velocity on the bottom are equal zero (no-slip condition). The initial for the mean flow is treated as still water with no current motion. For the free surface boundary condition, the zero gradient is imposed for both k and ε. The free surface motion is tracked by the volume of fluid (VOF) technique, Hirt and Nichols (1981).

3. VALIDATION METHOD

Hydraulic experiments in the case of tsunami overflow were carried out by Kato et al. (2013) and used in this study. A coastal embankment model was placed in a water channel with a length of 40 m, a width of 1 m, and a height of 1.5 m to reproduce tsunami overflow by supplying water from channel end using a pump, then flow velocity and pressure near coastal embankment were measured. The reduction scale of the experiment was $1/25^{th}$. Water level and the piezometric head were measured at 60 points using a point gauges and manometer, respectively Kato et al. (2013).

A computational domain was set as similar as a cross-sectional of physical experiment in real scale which was conducted by Kato et al. (2013) as seen in **Figure 1**. A coastal embankment was placed in a water channel with a height of 6 m, crown width 3 m and gradient 1:2. An initial steady flow runs from the left side of the numerical domain.

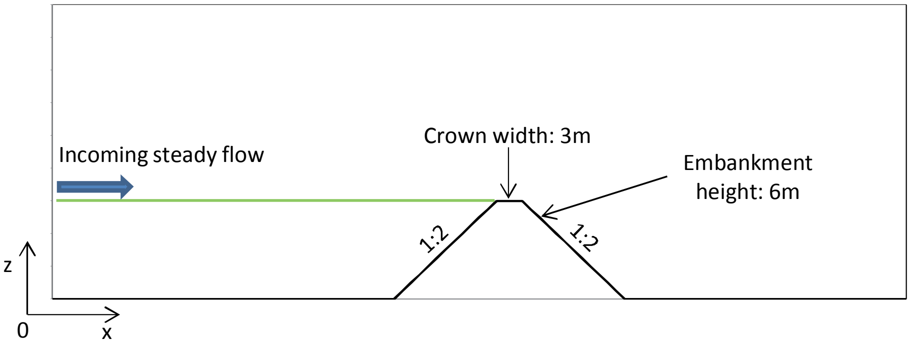

Figure 1. Domain sketch for numerical model

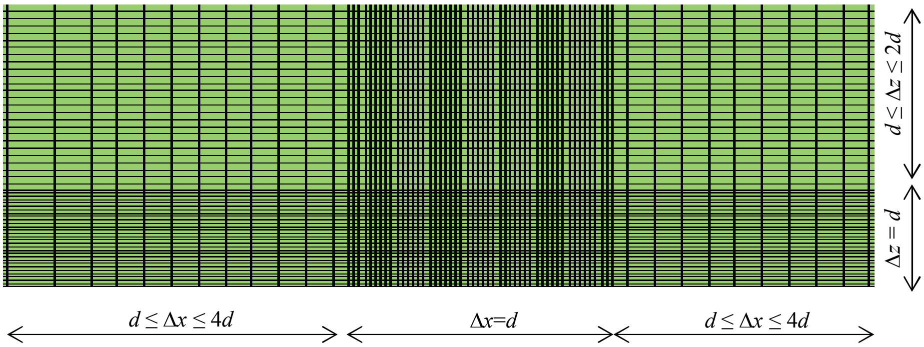

Figure 2. Typical mesh arrangement for the numerical model setup

In which, d is the minimum grid spacing in both x and z direction as can be seen in **Table 1**. The domain has the length of 200 m and the height of 20 m. There are three mesh system in x direction and two mesh system in z direction. A uniform mesh system $\Delta x_{min} = \Delta z_{min} = d$ is employed around the coastal embankment. In order to reduce simulation time, a non-uniform meshes system which is gradually increasing and employed far from the coastal embankment in both x and z direction as can be seen in **Figure 1** and **Figure 2**. The time step Δt was dynamically adjusted according to the stability criteria.

There are three cases with different overflow water depth 2 m, 6 m and 10 m as similar as hydraulic experiment were carried out by Kato et al. (2013) as shown in **Table 1.** Simulated results of water level, flow velocity and piezometric head were then used to compare with those hydraulic experiments by Kato et al. (2013) as can be seen in **Figure 3**.

Table 1 Three cases used for validation of the numerical model

Cases	Embankment height (m)	Overflow depth (m)	Minimum grid spacing *d* (m)
1	6	2	0.04
2	6	6	0.05
3	6	10	0.05

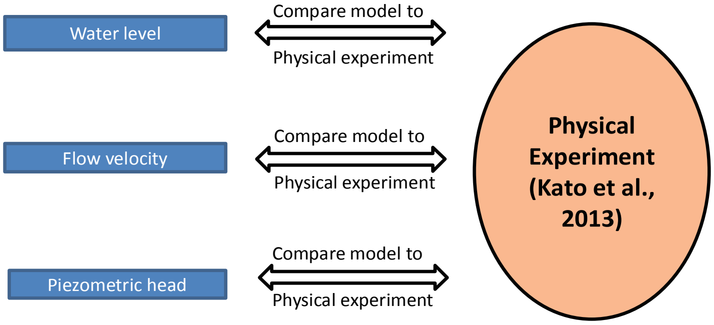

Figure 3 Three simulated results compare to those from physical experiments

4. RESULTS AND DISCUSSION

Figure 4 to **Figure 6** show the cross-sectional distribution of water level, flow velocity and piezometric head for the case of 2 m and 6 m and 10 m incoming waves, respectively. It can be seen that for all of the simulated cases, the calculated results for water level coincide well with the experimental results. The velocity in the experiment was obtained based on the flow discharge per water level at the given output points. In this study, depth-average flow velocity was calculated along the vertical axis. The calculated flow velocity was then compared to the experimental data. The magnitude of the computed velocity shows good agreement to that of the experimental data for all cases. The flow velocity increases from the embankment crest (x = 95 m) to landward areas (x = 117 m).

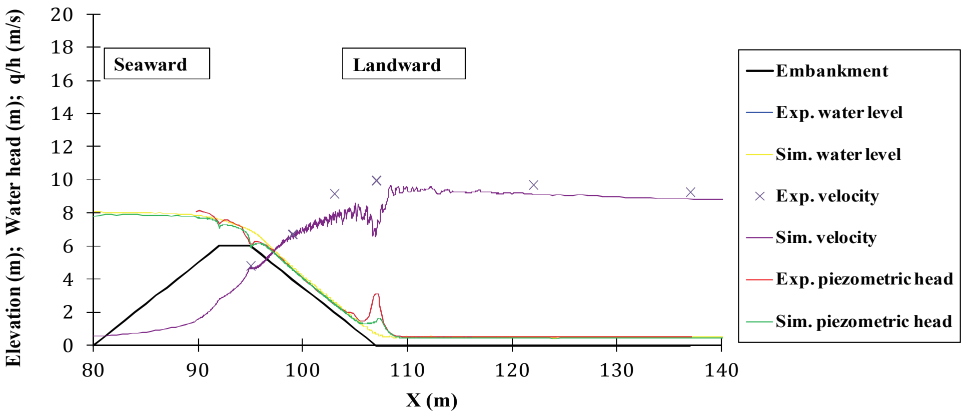

Figure 4 Comparison of distribution of cross-sectional average flow velocity, water depth and piezometric head in the case overflow depth 2 m (smallest grid spacing 0.04 m x 0.04 m)

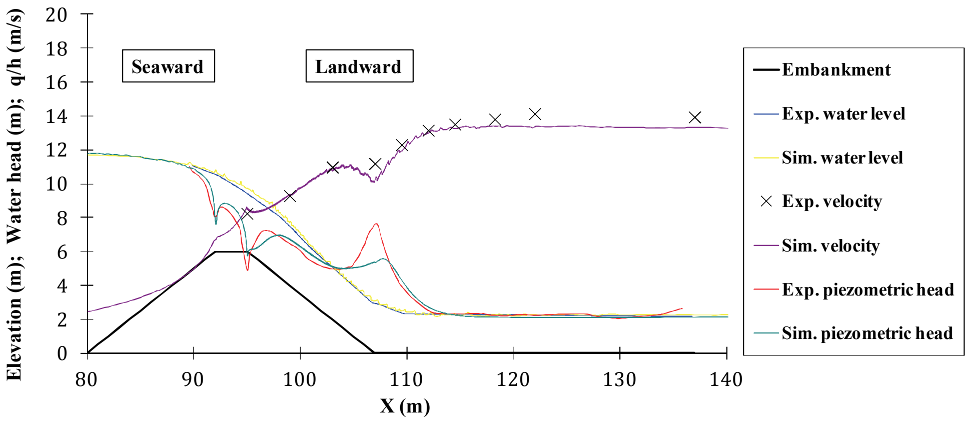

Figure 5 Comparison of distribution of cross-sectional average flow velocity, water depth and piezometric head in the case overflow depth 6 m (smallest grid spacing 0.05 m x 0.05 m)

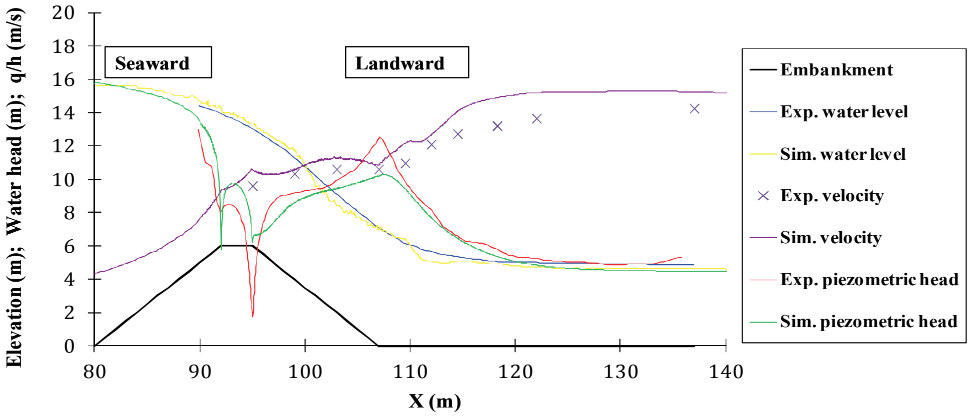

Figure 6 Comparison of distribution of cross-sectional average flow velocity, water depth and piezometric head in the case overflow depth 10 m (smallest grid spacing 0.05 m x 0.05 m)

As mentioned above, in the experiment, the piezometric head were measured at 60 points using a manometer. In this study, the pressure was computed and extracted from the grid at the bed. In general, the computed pressure correlates well and shows the same trend with the measured value. The pressure in the upstream and the downstream of the embankment shows similar value to the water level. This suggests that the pressure distribution relation to depth, as expected. On the top of the embankment, it was observed that there is a tendency of both simulated and experiment pressure on the surface of coastal embankment are reduced locally due to centrifugal action. This means that pressure on the coastal embankment surface is reduced in the landward slope top. In addition, negative pressure is generated at overflow depths of 6 m and 10 m in both simulated and measured. In contrast, the piezometric head increases locally in the landward toe with increased pressure. However, the simulated peak of the piezometric head at the embankment toe is lower than the experiment. This might be affected by the effect of grid spacing because at this location, the flow streamline separates from the bed. Thus a higher grid resolution will provide a higher accuracy here. It also can be seen from **Figure 4** that the flow velocity pattern at the landward embankment slope is oscillated in comparison to the other two cases. This might be caused by the grid spacing in Case-1 at the embankment toe is relatively large compared to the overflowing water depth.

5. CONCLUSIONS

This study has shown that the present NEWFLUME model is able to simulate well tsunami overtopping a coastal embankment. The tendency simulated results of piezometric head, water level and flow velocity show a good agreement with those from hydraulic experiment, especially flow velocity and water level. The results can still be improved with finer grid spacing around the embankment. In overall, the model has been validated for tsunami overtopping a coastal embankment. Finally, NEWFLUME model will be a valuable tool to assess tsunami risks, for example, reconstruction and implementation plans for post-tsunami reconstruction of coastal infrastructures such as coastal embankment etc.

ACKNOWLEGEMENTS

The authors would like to express their grateful thanks to Professor Pengzhi Lin of Sichuan University for providing the latest version of NEWFLUME. This research could not be conducted without financial supports from the Grant-in-Aid for Specific Research Project, International Research Institute of Disaster Science, Tohoku University. The appreciation is extended to the Grant-in-Aid for Scientific Research from the River Environmental Fund (REF) in charge of the Foundation of River and Watershed Environmental Management (FOREM).

REFERENCES

Hanzawa, M., Matsumoto, A., and Tanaka, H., 2012. Applicability of CADMAS-SURF to evaluate detached breakwater effects on solitary tsunami wave reduction. Earth Planets Space, 64, pp. 955-964.

Hirt, C.W. and Nichols, B.D., 1981. Volume of fluid (VOF) method for dynamics of free boundaries. J. Comput. Phys., 39, pp. 201-225.

Isobe, M., Xiping, Y., Umemura, K., and Takahashi, S., 1999. Study on development of numerical wave flume. Proc. Coast. Eng., JSCE, 46, pp. 36-40 (in Japanese).

Isobe, M., Hanahara, Y., Xiping, Y., and Takahashi, S., 2002. Numerical simulation of waves overtopping a breakwater. Proc. Int. Conf. Coast. Eng., ASCE, 28, pp. 2273-2285.

Kato, F., Suwa, Y., Watanabe, K., and Hatogai, S., 2013. Damages to shore protection facilities induced by the Great East Japan earthquake and tsunami. J. Disaster Research. 8(4), pp. 612-625.

Lin, P., 1998. Numerical modelling of breaking waves. Ph.D dissertation, Cornel University, Ithaca, USA.

Liu, P.L.-F. and Wen, J., 1997. Nonlinear diffusive surface waves in porous media. J. Fluid Mech., 347, pp. 119-139.

Lin, P. and Karunarathna, S.A.S, 2007. Numerical study of solitary waves interaction with porous breakwaters. J. Waterw. Port Coast. Ocean Eng. 133(5), pp. 352-363.

Rodi, W.: Turbulence models and their application in hydraulics – IAHR Publication. A state of the art review, 1980.