NUMERICAL MODEL STUDY ON TSUNAMI OVERTOPPING A COASTAL EMBANKMENT

Tohoku University Student Member ○ Nguyen Xuan DAO
Tohoku University Member Mohammad Bagus ADITYAWAN
Tohoku University Fellow Member Hitoshi TANAKA

1. INTRODUCTION

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 study on tsunami overtopping a coastal embankment by using the NEWFLUME model. The simulated results including water level, flow velocity and piezometric head were used to compare with those from hydraulic experiments for validation purposes. In overall, NEWFLUME model has been found to be a useful tool for evaluating the influences of tsunami overtopping on coastal structures.

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) Lin and Liu (1998). 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

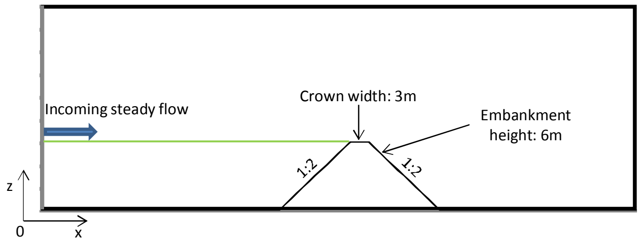

Figure 1. Domain sketch for numerical model

Table 1 Three cases used for numerical model validation

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	0.04

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 **Fig. 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.

In addition, 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 **Fig. 1**. 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.

Keywords: Tsunami; numerical model, validation, coastal embankment.

4. RESULTS AND DISCUSSION

Fig. 2a to **Fig. 2c** 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 is 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.

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

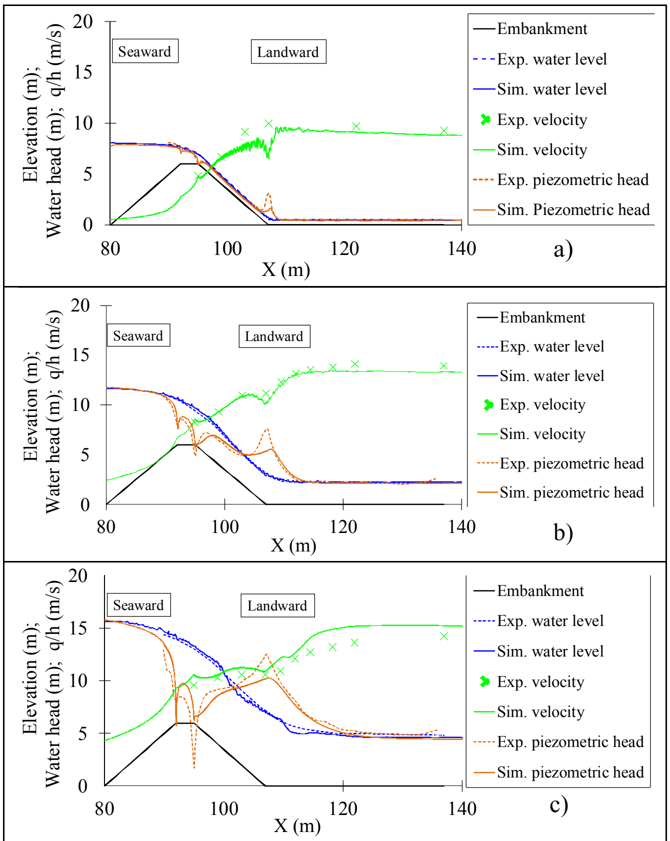

Figure 2 Comparison of distribution of cross-sectional average flow velocity, water depth and piezometric head in three cases.

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 **Fig. 2a** 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. CONCLUSION

This study has shown that the capability of the present NEWFLUME model simulates well tsunami overtopping a coastal embankment. The simulated results show a good agreement with those from hydraulic experiments, 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.

ACKNOWLEDGEMENTS

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 Scientific Research International Research Institute of Disaster Science, Tohoku University.

REFERENCE

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.

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., & Liu P. L.-F (1998). A numerical study of breaking waves in the surf zone. *J. Fluid Mech.* 359, pp. 239-264.