3D NUMERICAL SIMULATION FOR EQUIVALENT RESISTANCE COEFFICIENT FOR FLOODED BUILT-UP AREAS

JUNWOO CHOI

Department of Civil Engineering, Hanyang University, 17 Haengdang-dong, Seongdong-gu, Seoul, 133-791, Korea

KWANG OH KO

Department of Civil Engineering, Hanyang University, 17 Haengdang-dong, Seongdong-gu, Seoul, 133-791, Korea

SUNG BUM YOON

Department of Civil & Environmental Engineering, Hanyang University, 1271 Sa-1-dong, Sangnok-gu, Ansan, Gyeonggi-do 426-791, Korea

An equivalent resistance coefficient including the effect of drag caused by structures as well as a bottom friction was investigated for flooded built-up areas. As applying the coefficient into the bottom friction term in the shallow water equation model, relatively larger grids can be used for the drag effect of structures in inundation simulations. In order to investigate the equivalent resistance coefficient, the existing hydraulic experiment using unsubmerged square piers was simulated by using the RANS equation model of FLOW-3D, a 3-dimensional computational fluid dynamics code. The coefficient (n -value) was evaluated by applying the computed data into the Manning's equation. The computations were compared with the experimental results and the semi-analytical formula derived from momentum analysis including drag interaction effects. The computations agreed well with the experimental results and the semi-analytical formula. From the agreement it was confirmed that the drag interaction coefficient evaluated by the measurements was strongly dependent on the interval of piers. From the computations for the case, in which the water depths were much larger than those of the measurements, it was found that when the unsubmerged piers were arranged in a fully developed open channel, the equivalent resistance coefficient increased with 2/3 power of water depth.

1 Introduction

The frequency of destructive floods due to localized heavy rainfall and ocean earthquake has increased in the last decades, and it might enhance the possibility of urban flood hazard. During urban flood, flow resistances due to structures in built-up area play a significant role in increasing water surface elevation. Numerical models based on the nonlinear shallow water equation are generally utilized for inundation simulation. For the inundation simulation of built-up area, denser-grids approach to resolve all structures in computational domain is not only inefficient but also does not guarantee the accurate and reliable solutions because of deficiency of vertical momentum. The efficient approach to apply an equivalent resistance coefficient (i.e., Manning's n value [1]) into relatively larger grids of computational domain can be employed. However, since the equivalent resistance coefficient includes the effect of flow resistances due to structures as well as bottom friction in an inundation area, it should be a function of shape, projected area, placement, intervals of resistant bodies, water depth, Reynolds number and so on.

In this study the hydraulic experiment using unsubmerged square piers [2] was simulated by using the RANS (Reynolds Averaged Navier-Stokes) equation model with the VOF model in FLOW-3D, a 3-dimensional computational fluid dynamics code [3]. The equivalent resistance coefficient n values, which were evaluated by applying the computed data into the Manning's equation, were compared with the results of the hydraulic experiment and the semi-analytical formula derived from momentum analysis including drag interaction effects. The results of the experimental, numerical, and theoretical analyses can be mutually verified from the comparisons. And from the computations for the case, in which the water depths were much larger than those of the measurements, the relation between the equivalent resistance coefficient and water depth can be found.

2 Analytical Approach

In the fully developed open channel flow in which unsubmerged square piers arranged with an equal interval, a semi-analytical formula for an equivalent resistance coefficient including their drag effect were proposed [2] as

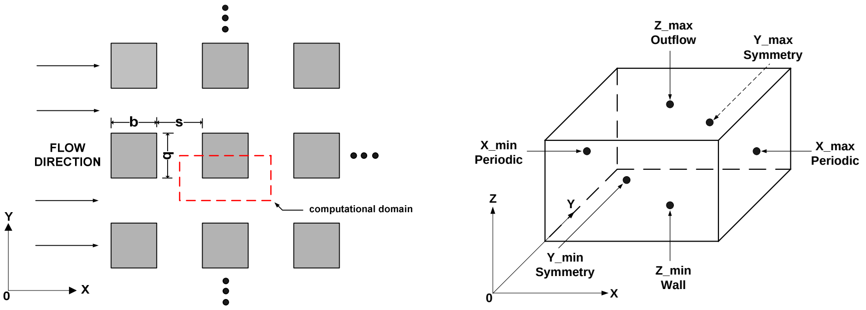
$$n = \sqrt{n_b^2 + C_{DI} C_D \frac{1 - r_0}{1 + s/b} \left(\frac{h}{b} \right) \left(\frac{h^{1/3}}{2g} \right)} \tag{1}$$

where n_b is the Manning's roughness coefficient, C_D is drag coefficient, s is the pier interval in the longitudinal direction, b is the pier width, h is water depth of a wide channel, r_0 is the porosity defined as the fraction of flow area excluding the projected area of piers in the channel, g is gravitational acceleration, and C_{DI} is called drag interaction coefficient. The semi-analytical formula was derived from the momentum equation of equilibrium under the assumption that bottom friction and form drags of the piers independently act in a mean steady and uniform flow. Therefore, in order to consider drag resistance influenced by their adjacent piers, the drag interaction coefficient C_{DI} was introduced as

$$C_{DI} = C_{DIT} C_{DIF} = C_{DIT} \left(1 - 0.95 \exp \left[-0.39 (s/b)^{1.8} \right] \right) \tag{2}$$

where C_{DIF} is longitudinal drag interaction coefficient and C_{DIT} is transverse drag interaction coefficient. The drag interaction coefficients were estimated by using the experimental data, but the experimental data were not enough to estimate C_{DIT} in terms of porosity. From the experiment [2], it was found that C_{DIT} was equal to 1.0 for two row-case with r_0 =0.715, C_{DIT} was equal to 3.1 for one row-case with r_0 =0.43, and C_{DI} was dependent on only the interval of piers in the longitudinal and transverse direction (porosity) in the turbulent region $\mathrm{Re} > O(10^4)$, in which drag coefficient $C_D = 2.1$ is constant for square pier.

3 Numerical Simulation



(a) Computational domain (b) Boundary condition

Figure 1. Schematic diagrams of computational domain and boundary conditions of numerical simulations using FLOW3D.

For the numerical experiment, a computational domain presenting a half of 11.4cm-wide square pier was chosen as shown in Figure 1(a). The square piers continuously aligned in rows for the experiment was modeled by the computational domain with symmetric and periodic boundary conditions shown in Figure 1(b). For upstream and downstream boundaries, periodic boundary condition, which interconnects the data such as velocity components and water depth between two boundaries, was set to reach fully developed and uniform flow as in an infinitely long channel. For lateral boundaries, symmetry boundary condition was set to model infinite numbers of rows in the transverse direction. For bottom boundary, smooth wall (no-slip) condition was set and for upper boundary, pressure outlet condition was set. The channel slope 0.0049 was set by using the X-component gravity. And, uniform water depth, which was less than pier heights, was set by using the initial volume fraction in the VOF model. Reynolds number was maintained in the region $\mathrm{Re} \sim O(10^4)$ for a constancy of drag coefficient of square pier. For the turbulence model, the standard $k-\varepsilon$ model was chosen with the standard wall function. In order to see the variation of the n-value according to longitudinal pier interval, the longitudinal pier intervals were changed in the range from 0.023 to 0.912m with Y-directional 0.1m-domain for $r_0 = 0.43$ and Z-directional 0.11m-domain for water depth 0.05 and 0.09m. And in order to see the variation of the n-value according to water depth, water depths were changed in the range from 0.0285 to 0.342m with Y-directional 0.1m-domain (r_0 =0.43) and X-directional 0.362 and 0.614m-domain.

4 Results of Numerical Simulations

4.1. *Resistance Coefficient according to Pier Interval*

Numerical test of the case, in which drag resistances of the piers do not interact with each other, was performed to analyze and compare its computations with the analytical results.

The conditions of the numerical test were that the porosity r_0 was 0.72, for which the resistances of piers were independent in the experiment, the uniform water depth was 0.09 m, and the ratio of pier width and longitudinal pier interval s/b was 4.0, 5.0, 6.5. Figure 2 shows the computations and the analytical results obtained by using Eq. (1) with C_{DI} =1.0. For the region $s/b \geq 4$, in which the resistances of piers are independent so that Eq. (1) with C_{DI} =1.0 was valid, the results agreed with each other. From the agreement it can be said that the numerical model gives a reasonable result when the piers were arranged with wide enough to do not interact with each other in terms of resistance.

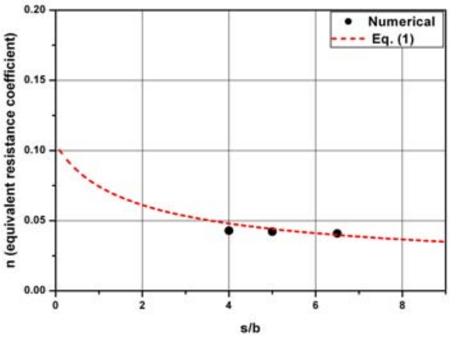
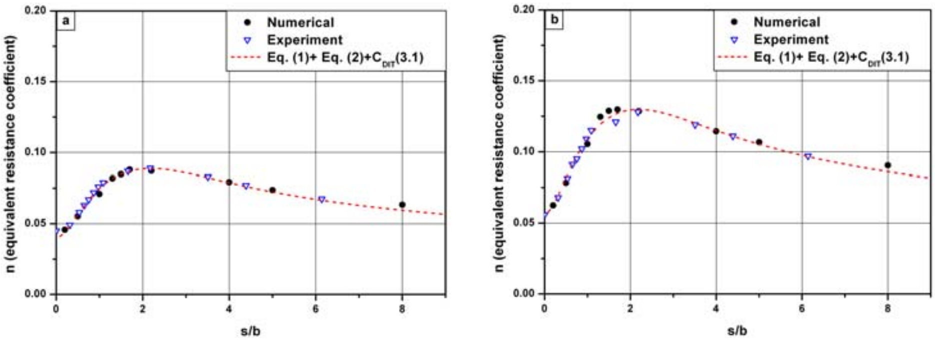


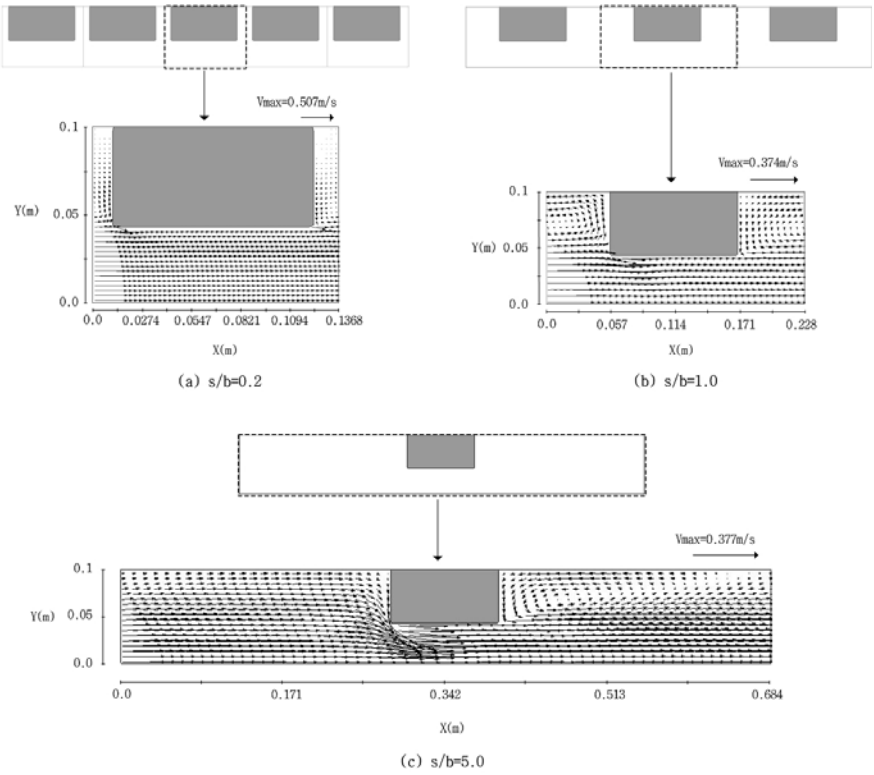
Figure 2. Variation of the equivalent resistance coefficient n -value according to the dimensionless pier interval s/b in the longitudinal direction (h =0.09m, r_0 =0.715).



(a) h =0.05m (b) h =0.09m

Figure 3. Variation of the equivalent resistance coefficient n -value according to the dimensionless pier interval s/b in the longitudinal direction (r_0 =0.43).

Second, numerical simulation of the case, in which drag resistances of the piers interact with each other because of narrow pier interval in the longitudinal direction, was performed to analyze and compare its computations with the semi-analytical results. The conditions of the numerical simulation were that the porosity r_0 was 0.43, the uniform water depth was 0.05 and 0.09 m, and the various longitudinal pier intervals ranged 0.023~0.912. Figure 3 shows the computations, the existing measurements and the analytical results obtained by using Eq. (1) and Eq. (2) with C_{DIT} =3.1 in order to present the variation of the n -value according to longitudinal pier interval s/b. They agreed well with each other. Above all, the agreement implies that the results of the three different approaches (numerical, experimental and analytical analysis) are reliable. From the measurements and the semi-analytical formula, Choi et al. [2] described as in the followings: For $s/b < 4$, the resistance of upstream piers affects the resistance of downstream piers, since the approach velocities decrease due to the turbulent eddies in an insufficient longitudinal space. At $s/b \approx 2.2$, the n value had its maximum. For $s/b < 2.2$, the n value decreased as the longitudinal space behind piers was reduced because the approach velocities were reduced. For $s/b > 2.2$, the n value decreased as the number of pier per a unit channel length decreased. The numerical results also confirmed the above relation between the equivalent resistance coefficient and the pier interval. Figure 4 shows the velocity vectors near water surface at s/b =0.2, 1.0 and 5.0. In the figure, as the longitudinal space behind piers decrease, the approach velocities and the vortex scales decrease. Based on the agreement, the numerical results shown in figure 4 can be said to be reasonable.



(a) s/b=0.2

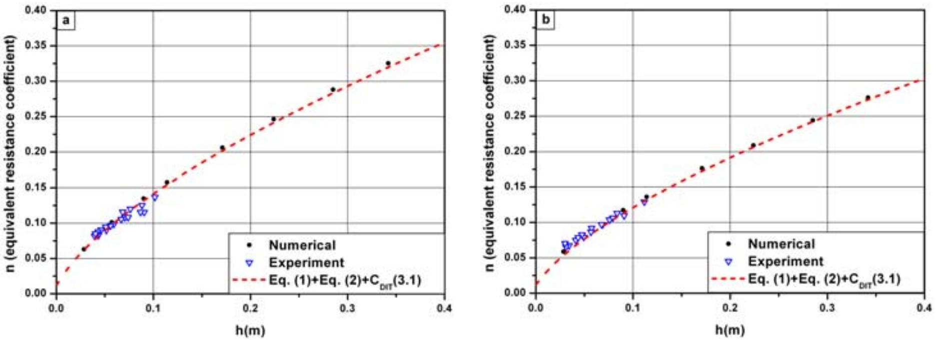
(b) s/b=1.0

(c) s/b=5.0

Figure 4. Snapshots of computed velocity vectors at the different longitudinal pier intervals (X-Y plane).

4.2. *Resistance Coefficient according to Water Depth*

Numerical simulation of the case, in which drag resistances of the piers interact with each other in the larger water depths than those of the measurements, was performed to analyze and compare its computations with the semi-analytical results. The conditions of the numerical simulation were that the porosity r_0 was 0.43, the longitudinal pier intervals were s/b =2.175 and 4.386, and the various water depths ranged from 0.0285 to 0.342m. Figure 5(a) and 5(b) show results of the computational, the experimental and the analytical approaches in the case for s/b =2.175 and in the case for s/b =4.386, respectively. Based on the experiments, the case for s/b =2.175 had drag interactions due to piers in both of the longitudinal and transverse direction, and the case for s/b =4.386 had drag interactions due to piers in the transverse direction. The analytical results were obtained from the equivalent resistance coefficient Eq. (1) using Eq. (2) for C_{DIF} and C_{DIT} =3.1. From the comparisons, all results of the three different approaches agreed well with each other. Especially in the higher water depth-cases, which were not able to be performed in the hydraulic experiments, the computations and the analytical approach show a good agreement. This agreement confirmed that when unsubmerged square piers were aligned in rows in the fully developed turbulent open channel flow, the equivalent resistance coefficient n –value is proportional to the 2/3 power of water depth as shown in the analytical formula Eq. (1).



(a) *s/b*=2.175 (b) *s/b* =4.386

Figure 5. Variation of the equivalent resistance coefficient n -value according to the water depth *h* (r_0 =0.43).

5 Conclusions

In order to investigate the equivalent resistance coefficient n -value of unsubmerged square piers aligned in rows in a fully developed open channel flow, numerical simulation of the existing experiment was performed, and the n -value was evaluated by applying the computed water depth, discharge and slope into the Manning's equation. The results of the numerical simulations were compared with the experimental data and the semi-analytical formula derived from momentum analysis including empirical drag interaction coefficient, and they agreed well with each other. From the agreement it can be inferred that the results of the three different approaches are reliable. From the results of larger water depth-cases, which were not able to be performed in the hydraulic experiments, it was confirmed that when unsubmerged piers were arranged in the fully developed open channel flow, the equivalent resistance coefficient n –value increased with the 2/3 power of water depth.

Acknowledgments

This research was conducted by the financial support by the research fund from the Ministry of Education, Science and Technology in Korea through Brain Korea 21 project granted to Hanyang University.

References

1. V.T. Chow, *Open Channel Hydraulics*, McGraw-Hill (1959).
2. J. Choi, K.K. Kwon, K.O. Ko, S.B. Yoon, *Proceedings of the 5th Intenational Conference on Asian and Pacific Coasts, Singapore* (2009).
3. Flow Science, Inc. *FLOW-3D User Manual*. (2000).