A30 2-D AND 3-D NUMERICAL ANALYSIS OF FLOW AROUND SUBMERSIBLE GROINS IN SERIES

HIROSHI KAWAGUCHI
Graduate School of Engineering, Hiroshima University, 1-4-1 Kagamiyama, Higashi Hiroshima, 739-8527, Japan
E-mail: kawaguti@hiroshima-u.ac.jp

SHOJI FUKUOKA AND AKIHIDE WATANABE
Department of Social and Environmental Engineering, Graduate School of Engineering, Hiroshima University, 1-4-1 Kagamiyama, Higashi Hiroshima, 739-8527, Japan
E-mail: sfuku@hiroshima-u.ac.jp and awata@hiroshima-u.ac.jp

1. Introduction

Groins are used in curved channels against bank erosion and for maintenance of watercourses used for transportation. The use of groins has been investigated through experiments, numerical analysis and direct construction in rivers. Recently, in addition to such conventional purposes, groins have been used as structures capable to create favorable environment as they permit the diverse variation of flow and bed topography. Also, riparian trees have been used as permeable groins to maintain the existing environment along riverside. It is important to evaluate the effect of groins against flow and the streambed, not only qualitatively, but also quantitatively for each purpose, because groins are used for various purposes. In this paper, we discuss the following; First, to examine the applicability of 2-D numerical analysis for flow around submersible groins. Second, to estimate applicability of 3-D numerical analysis and hydrodynamic forces acting on a groin by this model.

2. 2-D numerical analysis for flow around a submersible groin

The ϕ - ω method was used to solve the flow with free surface around the groin. Here, ϕ is stream function and ω is vorticity in the x-y plane. Vorticity and pressure equations are developed by taking rotation and divergence of depth-averaged momentum equation for the hydrostatic pressure distribution and the hydrodynamic force acting on the submersible groin. Stream function in steady flow with free water surface was defined from depth-averaged continuity equation. An equation relating stream function with vorticity was obtained from the definition of vorticity and stream function. The governing equations were discretized in x and y direction by finite differences method and the Galerkin method respectively. The hydrodynamic force per unit volume was represented using the Heaviside function. Drag and lift coefficients C_D and C_L are 5.6 and 0.2 obtained from the measured results in laboratory experiments.

3. Comparison of experimental results and numerical results of the 2-D model

Two-dimensional analysis was performed for the case of the steady flow in a flat bed channel with submersible groins in series, as performed by the authors. The calculated water levels are higher at the groin section and immediate downstream of the groin. This discrepancy results from the assumption of hydrostatic pressure distribution in the computation, which is not valid due to flow separation occurring in this region. At other cross sections, calculated water levels are close to the observed water levels. The depth-averaged velocity U of the computation are smaller than that of the observed in the region of x=600cm and y=0～50cm. This result is consistent with the small water level gradient in x direction. But at the other cross sections, the reduction in velocity between groins is computed well. The difference between computed and experimental depth-averaged velocity V is recognized at x=600~620cm. This result also is consistent with the relatively small computed water level gradient in y direction.

From these results, it is observed that 3-D analysis is required to provide better consideration of pressure and velocity distributions in flow separation area.

4. 3-D numerical analysis for flow around a submersible groin

In the 3-D numerical analysis, pressure distribution is solved by the SMAC method and turbulent flow is solved by LES model.

The grid-scale-momentum equation is used for LES. The velocity vector and pressure are filter-averaged value in a grid scale. In this momentum equation, the shear stress is produced by the sub-grid-scale turbulent flow. This shear stress is calculated using the Smagorinsky model for the turbulence energy equation in sub grid scale. Only coefficient C_s has a value between 0.1~0.25 in general.

This SMAC method is an implicit method to solve the pressure term in grid-scale-momentum and continuity equations. The momentum and continuity equations are solved through the following two steps. At first, the momentum equation is solved by Euler's explicit method. In the second step, to satisfy the continuity equation in the next step, the increment of pressure is solved by pressure equation. It is noted that the kinematic boundary condition is solved with pressure equation, as the free water surface is a boundary condition for pressure.

The calculation for this calculation are $\Delta x=\Delta y$=5cm, Δz=1cm, Δt=0.004s. The momentum equations are discretized on a staggered grid. The convection term is discretized by the first-order upwind method. The pressure is zero at the closest mesh to the water surface. And the gradient of the pressure is zero at the wall. A cyclic boundary condition is used in the x direction. The average water level from the experiment is specified. The shear stress on the wall is represented by the use of a logarithmic law for the velocity distribution. To be consistent with the experimental water discharge, k_s was set to 0.6cm and C_s=0.35.

5. Comparison of experimental results and numerical results of the 3-D model

The 3-D model is solved for the same steady flow case used in the 2-D model. The 3-D numerical analysis cannot represent the water level drop in the downstream part of the groin. The maximum difference between computational and experimental water level was about 5mm. But water level distribution in x direction around groin is improved than that obtained by the 2-D model, especially around x=600cm y=50cm. So, the gradient of water level in y direction around that point is steeper and close to the experimental data. The computed water level is close to that of measured except for the separation area. Regarding the velocity in the x direction u, the 3-D model still underestimates the velocity u near groin section for the same reason explained before. The experimental data is found to be negative at the downstream of the groin indicating that separation flow occurred in the experiment. However, the numerical result is not negative there. It is believed that this 3-D model is limited by the relative coarse mesh employed, because in zones of large gradients, the computed magnitude of viscosity in the Smagorinsky model is underestimated (underestimated velocity gradients). This is the reason why Cs was made larger in order to fit the experimental discharge and counterbalance the low value of $2S^2$ in the Smagorinsky model. The result of 3-D model is closer to experiment than that of 2-D model, numerical water surface being improved around the point x=600cm y=50cm. This numerical analysis shows relatively well the characteristics of the experiment.

The difference between computed and measured hydrodynamic force acting on the groin is due to the insufficiency of flow representation around a groin.