FLOW PATTERN AROUND A COMPOUND BRIDGE PIER

Ehime University, Graduate School of Science and Eng. Ricardo de Aragão (Ph.D. Student)
Ehime University, Dep. of Civil and Environmental Eng. Akihiro Kadota (Dr. of Eng., Assistant Professor)
Ehime University, Dep. of Civil and Environmental Eng. Koichi Suzuki (Dr. of Eng., Professor)
Federal Univ. of Campina Grande, Brazil, Dep. of Civil Eng. Vajapeyam S. Srinivasan (Ph.D. Professor)

INTRODUCTION

Scour is the erosive action of water, which excavates soils from streambeds and banks and is classified into general, contraction, and local scour. The last two types are direct consequences of the structures set inside the main river channel as bridge foundations (piers and abutment), for instance. Much has been learned about the mechanism of scour around bridge pier; however, the bridge designers are still unable to predict with confidence the depth of the scour hole. The empirical equations mostly used by them for determination of this value rely on experiments conducted under controlled conditions. Therefore, turbulence, vortexes, flow pattern, and the three-dimensionality of the approaching flow are factors seldom considered. To achieve a reliable methodology for obtaining the scour depth, the flow pattern around these structures should be well known. This can be done through the use of numerical models that consider a computational fluid dynamic (CFD) approach. In this sense, several researches have been conducted on flow field around cylindrical pier. However, no research has been found about flow patter around different pier shapes.

In this paper, the simulation of flow pattern around a compound pier has been conducted by means of a CFD methodology. Hereafter the equations used and the results obtained will be presented.

THREE-DIMENSIONAL FLOW SIMULATION

The simulation of the velocity components was done by solving the three-dimensional Navier-Stokes equations, which consist of conservation of mass, momentum, and energy. These equations are written as follows:

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho U_i}{\partial x_i}=0 \tag{1}$$

$$\frac{\partial \rho u_i}{\partial t}+\frac{\partial \rho u_i u_j}{\partial x_i}=\frac{-\rho\partial \overline{u_i u_j}}{\partial x_i}-\frac{\partial p}{\partial x_i} \tag{2}$$

where ρ is the local fluid density, t is time, x_i is the position vector in the i_{th} coordinate direction, U_i is the i_{th} fluid velocity component, p is the pressure, $\overline{u_i u_j}$ is the Reynolds stress term. To evaluate this term, a turbulent model is required. Here the well-established two-equation standard κ-ε turbulence model has been applied by means of the Boussineq eddy-viscosity concepts:

$$-\overline{u_i u_j}=\nu_t\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)-\frac{2}{3}\delta_{ij}\kappa \tag{3}$$

where ν_t is the eddy viscosity coefficient, δ_{ij} is the *Kronecker* delta function, κ is the turbulent kinetic energy.

The eddy-viscosity coefficient, ν_t, is obtained by the relationship between the size and the energetic of individual eddies in fully developed, isotropic turbulence, $\nu_t=C\mu\rho\kappa^2/\varepsilon$, where C_μ is a dimensionless model constant, assumed as being equal to 0.09; ρ is the local fluid density, and κ and ε are the specific turbulent kinetic energy and turbulent kinetic energy dissipation rate, respectively. The following equations are used to calculate κ and ε:

$$\frac{\partial \rho k}{\partial t}+U_i\frac{\partial \rho\kappa}{\partial x_i}=\nu_t G-\varepsilon+\frac{\partial}{\partial x_i}\left[\left(\nu+\frac{\nu_t}{\sigma_\kappa}\right)\frac{\partial \kappa}{\partial x_i}\right] \tag{4}$$

$$\frac{\partial \rho\varepsilon}{\partial t}+U_i\frac{\partial \rho\varepsilon}{\partial x_i}=\frac{\varepsilon}{\kappa}(C_{\varepsilon 1}\nu_t G-C_{\varepsilon 2}\rho\varepsilon)+\frac{\partial}{\partial x_i}\left[\left(\nu+\frac{\nu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right]. \tag{5}$$

$$G=-\overline{u_i u_j}\frac{\partial U_i}{\partial x_j} \tag{6}$$

The values of the empirical constants have been established as $C_{\varepsilon 1}$=1.44, $C_{\varepsilon 2}$=1.92, σ_k=1.0, and σ_ε=1.3.

CASE STUDY

The concepts above mentioned were used to analyze the flow pattern around the piers of the Shimanto River Bridge, whose foundations have been scoured. This bridge was built in Nakamura City, Kochi prefecture, Japan. The Shimanto River Bridge is 508 m in length and is pillared by 13 compound piers, whose shape is shown in Figure 1. Because the annual typhoons, the area around the bridge has been flooded almost every year. As a consequence, the peak discharge has reached values up to 13,000 m^3/s.

MODEL APPLICATION

To conduct the simulations, data about flow depth and velocity, and scour depths collected during previous experiments were considered. The geometry applied had the following dimensions: 0.71 m x 0.1 m x 0.38 m in the x, y, and z directions, respectively. The pier model was similar to the one shown in Fig. 1, but the diameter was D=0.048 m. The sediment was uniform (D_{50}=1.42 mm). These characteristics were used in the calculation. The bed was set no-slip with a roughness equal to the D_{50} above cited. From one of the experiments a water depth h=0.1 m, a mean velocity U=0.3 m/s and a respective shear velocity U_*=0.017 m/s had been chosen and applied to the log-law velocity distribution.

In order to avoid the influence of the grid size on the numerical calculation, a fine and non-uniform mesh was used. It had 200 x 140 x 60 cells in the streamwise, cross-streamwise, and vertical direction, respectively. As it is known, when flowing water approaches a pier, it is forced to separate at its sides encloses downstream of it. Water also turns downward the impiping on the bed, acting as a jet. This will dislodge the material, creating a hole upstream of the pier,

Keywords: Scour, turbulence, Navier-Stokes, computational fluid dynamic, model
Dept. of Civil and Environmental Engineering, Ehime University, 3 Bunkyo-cho, Matsuyama-shi, Ehime-ken, Japan

and vortexes at the sides. A wake vortex is generated at the downstream of the piers, which entrain the loose particle and remove the bed material. To be realistic, a 3D flow simulation model should present a result similar to the above described.

Based on the results acquired, a downflow along a vertical line near the pier nose can be clearly seen (Fig. 2). The velocity field and the contour of the velocity close to the bed , y/D=0.1, (Figs. 3 and 4) show that the shape of this pier induce a creation of vortices past the first cylinder, which slow the flow, reducing the energy necessary for transporting material and, consequently, the scour effects. The wake vortices seen in this figure, at downtream face, are effective in removing the sediment at the downstream face of the pier. The material removed from both upstream and downstream of the pier will settle past the pier where the shear stress is below the critical value (0.06). The correspondent shear distribution at the same depth is seen in Fig. 5. From this result, one can see that the 3D-flow model can simulate flow features around the pier very well. Through a comparison of results from experiments with circular and compound pier, (Fig. 6), it is seen that for the same value of the dimensionless number *Ns* ($Ns=U/\sqrt{sgD}$ - U is the mean flow velocity (m/s), s is the sediment specific gravity (dimensionless), g is the gravitational acceleration (m/s^2), and D_g is the typical grain diameter of the surface particle (m)), the scour depth at the cylindrical pier is deeper than the one at the compound pier, stressing the hole of the shape on the scour process.

Considering the flow pattern for several values of flow velocities and depth, the results from the simulations could be useful in developing a reliable scour depth equation.

CONCLUSION

In summary, the flow patter around a compound pier, similar in shape and scaled 1/100 of the Shimanto River Bridge, has been analyzed by using a computational fluid dynamic model. The results obtained allow us to better understand the flow field distribution around this structure and the influence of the pier shape on the scour process. Furthermore, it seems compatible with the results from the experiments. This approach could be use to develop a scour depth equation more close to the physical process.

REFERENCES

Rodi, W. (1980) Turbulence models and their application in hydraulics, IAHR – Monograph, Balkema.

Aragão, R., Kadota, A, and Suzuki, K. Local scour around the piers of the Shimanto River, In: Annual Journal of Hydraulic Engineering, JSCE, Vol.47, pp. 673-678, 2003, March

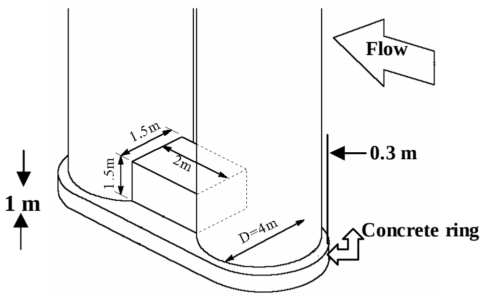

Figure 1. Shimanto River Bridge pier.

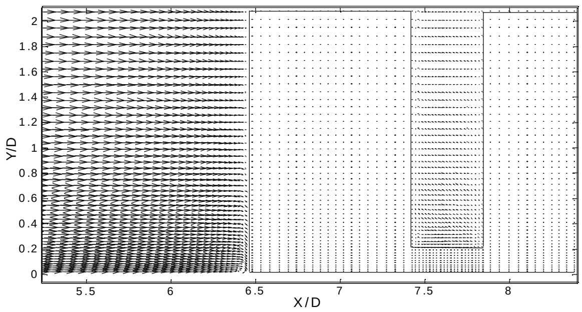

Fig. 2 Velocity field on the vertical plane section (XY plane).

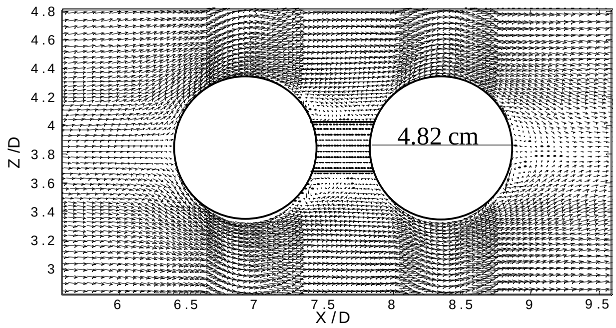

Fig. 3 Velocity field on the horizontal plane at y/D=0.1

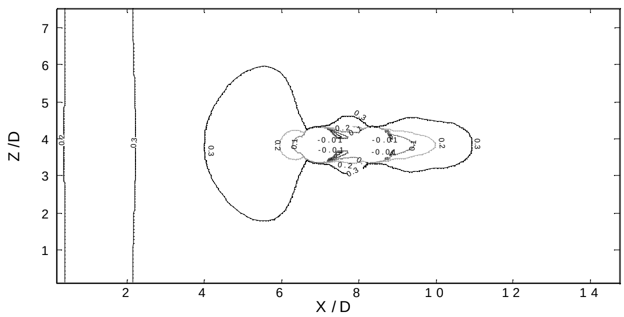

Fig. 4 Contour of velocity in U direction, m/s

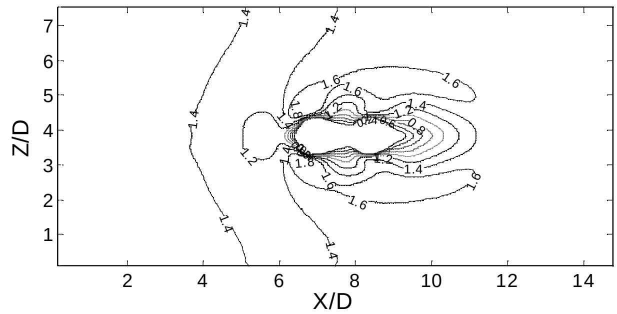

Fig. 5 Contour of shear stress above the bed, dimensionless

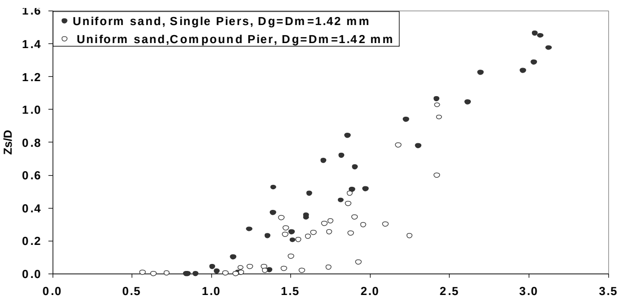

Fig. 6 *Zs/D* vs *Ns* for single and compound pier using a uniform sand and attack angle=0º.