# Bed Shear Stress Around Rectangular Pier: Numerical Approach

Prahoro Nurtjahyo[1], Hamn-Ching Chen[2], Jean-Louis Briaud[3], Ya Li[1], Jun Wang[1]

## ABSTRACT

*A Reynold-Averaged Navier-Stokes (RANS) method has been employed in conjunction with a chimera domain decomposition technique for time-domain simulation of flow around a rectangular pier. The main objective of the numerical model is to find the maximum shear stress which exists around the pier at the beginning of the scour process. It is found that for ratios of length (L) over width (B) greater than 1, the maximum shear stress remains constant. There is a significant influence of the effect of L/B on the maximum shear stress for L/B less than 1.*

## INTRODUCTION

In recent years, computational fluid dynamics (CFD) has quickly become one of the most popular and efficient methods to determine fluid flow behavior in industrial and environmental applications. In the scour research area, some numerical code have been used to simulate flow and scour around vertical pile.

BRIdge Stream Tube model for Alluvial River Simulation (BRI-STAR, Molinas, 1990) is a semi-two-dimensional code that is used to simulate laboratory bridge scour experiments. Hoffman and Booij (1993) applied Duct-model and Sustra-model to simulated the development of local scour holes behind the structure. Olsen and Melaaen (1993) simulated scour 3D around a cylinder by using finite volume SSIIM. The SSIIM model solves Reynold stresses by the $k-\varepsilon$ turbulence model. They observed and reported that there is agreement between the pattern of the vortices in front of the cylinder and the model. Wei et al. (1997) performed a numerical simulation of the scour process in cohesive soils around cylindrical bridge piers. A multi-block Chimera RANS method was incorporated with a scour rate equation to compute scour processes. A linear scour rate equation was assumed in which the scour rate was expressed as a linear function of the streambed shear stress. The simulation captured the important flow features such as the formation of horseshoe vortices ahead of the pier and the flow recirculation behind the pier. Comparison was showing an agreement with experimental data when the model simulated the time history of scour depth. Wei found out that the

[1] *Graduate Research Assistant, Department of Civil Engineering, Texas A&M University*
[2] *Professor, Ocean Engineering Program, Department of Civil Engineering, Texas A&M University*
[3] *Professor, Geotechnical Engineering, Department of Civil Engineering, Texas A&M University*

value of the critical shear stress has a significant influence on the scour process around a cylinder in cohesive soils. Xibing Dou (1997) simulated the development of scour holes around piers and abutments at bridge crossings. A stochastic turbulence closure model which includes an isotropic turbulence has been incorporated into a three-dimensional flow model, CCHE3D. Roulund et al (1998) present a comprehensive description on the flow around a circular pile and the development of the scour by use of a numerical study as well as an experimental study. The numerical model solves the three-dimensional Reynolds averaged Navier-Stokes equations with use of the $k$-$\omega$ turbulence closure model. The method implemented a fully three-dimensional bed load formulation including the effect of gravity.

Briaud et al (2001a, b) proposed a method to predict scour depth versus time curve around a cylindrical bridge pier of diameter D founded on cohesive soils, so called SRICOS-EFA. A series of numerical simulations were performed to develop an equation for the maximum shear stress $\tau_{\max}$ around the pier the scour start to develop. The method has been extended to cover complex single pier which include the effect of water depth, shape, group pier, and attack angle. This paper is part of the development of the new SRICOS-EFA method.

Bed shear stresses in cohesive soils around rectangular piers are presented in this research. The scour equation is incorporated with the 3-D RANS method (Wei et al., 1997) and the procedure of scour calculation will be coupled with the multi-block RANS flow solver (Chen and Chen 1998).

## CHIMERA RANS METHOD

In the present study, the Chimera RANS method of Chen and Chen (1998) has been employed for a detailed resolution of the unsteady, viscous flow around a vertical pile. The method solves the Reynolds-Averaged Navier-Stokes equations for incompressible flow:

$$U^i_{,i} = 0 \tag{1}$$

$$\frac{\partial U^i}{\partial t} + U^j U^i_{,j} + \left(\overline{u^i u^j}\right)_{,j} = -g^{ij} p_{,j} + \frac{1}{\mathrm{Re}} U^i_{,jk} = 0 \tag{2}$$

where $U^i$ and $u^i$ represent the mean and fluctuating velocity components, and $g^{ij}$ is conjugate metric tensor, $p$ is pressure, and Re is the Reynolds number based on the pier diameter.

The present method solves the mean and turbulence quantities on embedded, overlapped or matched multiblock. Within each computational block, the finite analytic method of Chen, Patel and Ju (1990) was employed to solve the unsteady RANS equations in general curvilinear, body fitted coordinate system. Both the kinematic and dynamic boundary conditions were implemented on the exact free surface for accurate resolution of the effect of the free surface. The two-layer model of Chen and Patel (1988) was employed for an accurate resolution of the turbulence boundary layer flow including the laminar sublayer and buffer layer in the near wall region. A more detailed description of the chimera RANS method is given in Chen and Chen (1998).

## RESULTS AND DISCUSSION

The scour process is highly influenced by the bed shear stress developed by the flowing water at the soil-water interface. When a cylinder obstructs the flow in an open channel with a flat bottom, the maximum shear stress $\tau_{max}$ is many times larger than the value given when there is no obstructions. Numerical simulation was performed (Wei et al., 1997) to obtain $\tau_{max}$. It was found that for large water depth ( $y/D > 2$ ), $\tau_{max}$ was dependent on the Reynolds number $Re$, the mean flow velocity $V$, and the mass density of water $\rho$. The equation is rewritten as:

$$\text{For a circular pier: } \tau_{\max} = 0.094 \rho V^2 \left[ \frac{1}{\log Re} - \frac{1}{10} \right] \qquad (3)$$

where the Reynolds number $Re$ is defined as $VD/\nu$ where $V$ is the mean flow velocity, $D$ is the pier diameter, and $\nu$ is the kinematic viscosity of water ($10^{-6}$ $m^2/s$ at $20^0C$). If this value of $\tau_{max}$ is larger than the critical shear stress $\tau_c$ that the soil can resist, scour is initiated. As the scour hole deepens around the cylinder the shear stress at the bottom of the hole decreases. Once the scour hole becomes deep enough, the bottom shear stress becomes equal to $\tau_c$ (the critical shear stress for the soil), the soil stops scouring, and the final depth of scour $z_{max}$ is reached.

Wei's approach (equation 3) is developed based on the assumption of a single pier and a large water depth so that the effect of free surface on riverbed can be ignored. Further improvement of the code has been done to capture the effect of water depth, the effect of pier spacing, the effect of shape, and the effect of attack angle. A number of parametric studies were performed to determine the relationship between the maximum bed shear stress and the effect of water depth, the effect of spacing, the effect of shape, and the effect of attack angle. In this present study, the objective is to obtain the relationship between the maximum bed shear stress and various rectangular shapes.

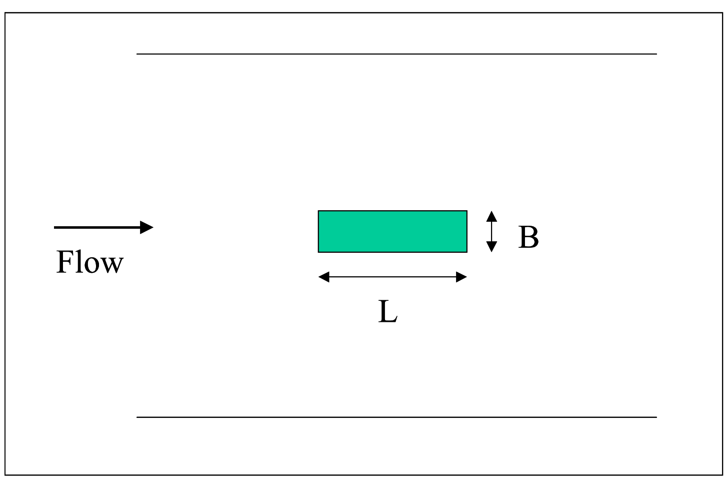


Figure 1. Problem definition

A rectangular pier with a width (B) of 0.061 meter was placed vertically in a 1.5 meter wide flume as shown in figure 1. The flow is a steady flow with a velocity of 0.33 m/s and the water depth is 0.375 meter. Four different pier length L/B=1, 4, 8, 12 are investigated. The value of Reynolds number is obtained based on the width of rectangular pier (Re=20130) and the Froude number based on the width of rectangular pier (Fr=0.4267). In order to reduce the amount of CPU time, a half domain is chosen because of the symmetry. The grid is constructed based on a half domain and divided into 4 blocks. Block#1 has dimensions 12x16x41 grid points in x, y and z direction respectively, Block#2 51x25x41, Block#3 32x16x41, and Block#4 61x7x41. In the case of L/B=1, the grid system is shown in figure 2.

The grid is very fine near the pier and riverbed to be able to apply the two-layer approach of the turbulence model. A few grid layers are placed within the viscous sublayer with the distance from the wall (i.e. pier and riverbed). The closest distance from the wall must satisfy $0.1 < y^+ < 1$, where $y^+ = \mathrm{Re}..u_\tau.y_n$, $u_\tau$ is the dimensionless friction velocity at the wall, and $y_n$ is the dimensionless normal distance from the grid layer to the wall.

The velocity vector around pier is shown in figures 3, 4, 5 and 6. The horseshoe vortex system that develops at the base of the pier is the major feature of the flow around the upstream half of the pier. The term "horseshoe" is derived from the shape that the system takes as it wraps around the upstream base of the pier and tails downstream. Two other features to note is the velocity component downward on the front face of the pier and a boundary-layer separation upstream of the pier. The pier redistributes the vorticity normally present in the flow. Most of the vorticity is diffused to the boundaries - namely the bed and the pier surface.

Figure 7, 8, and 9 shows the pressure field induced by the pier. If the pressure field is sufficiently strong, it causes a three-dimensional separation of the boundary layer which in turn rolls up ahead of the pier to from the horseshoe vortex system. A blunt nosed pier is defined as being one which the pressure field induced by the pier is sufficiently strong to form the horseshoe vortex system.

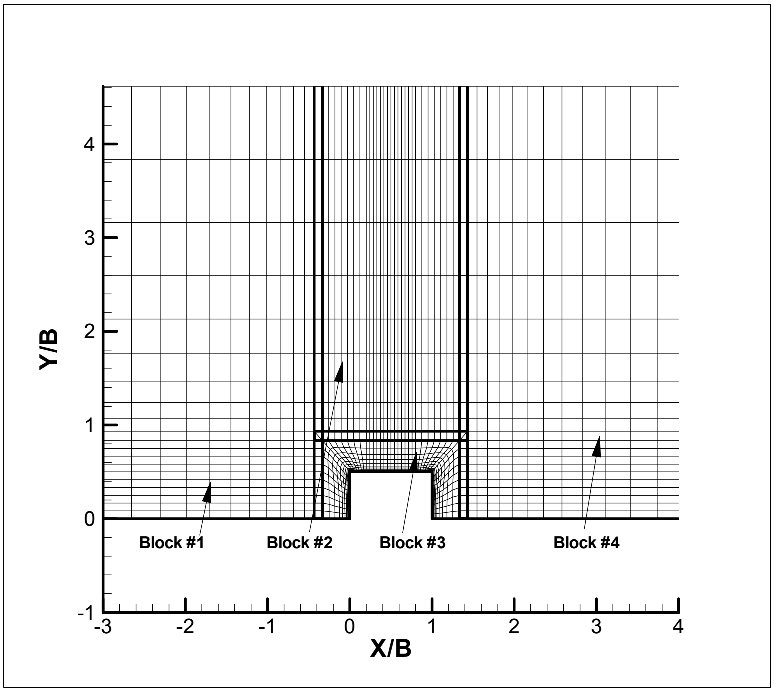


Figure 2. The Grid System for L/B=1

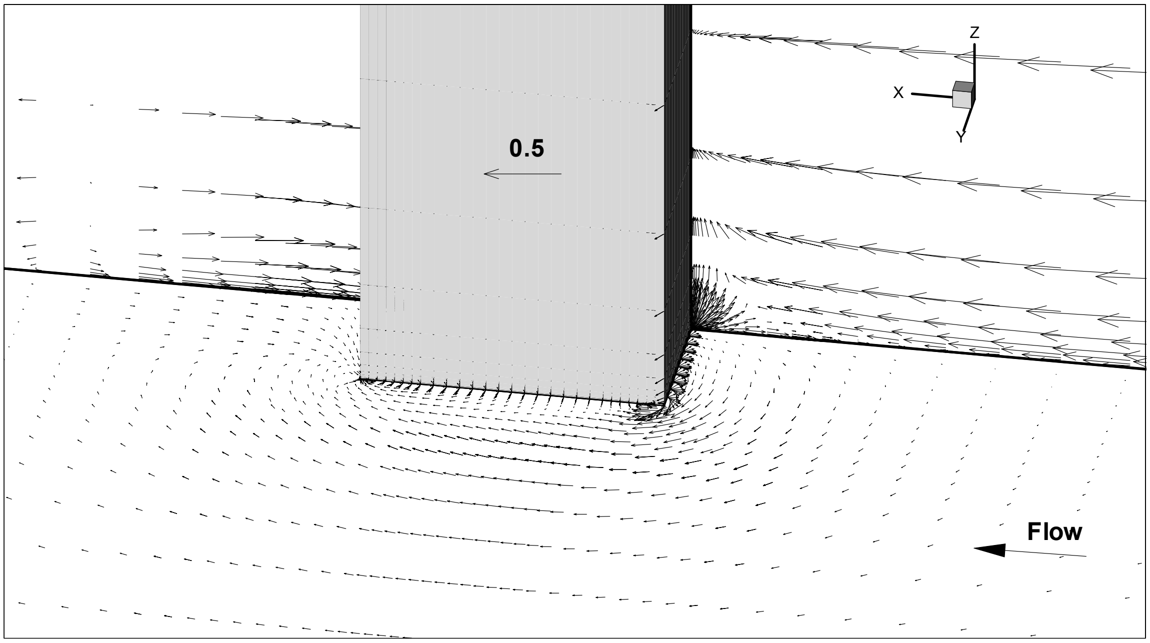


Figure 3. Velocity vector around rectangular pier (L/B=1) at initial condition

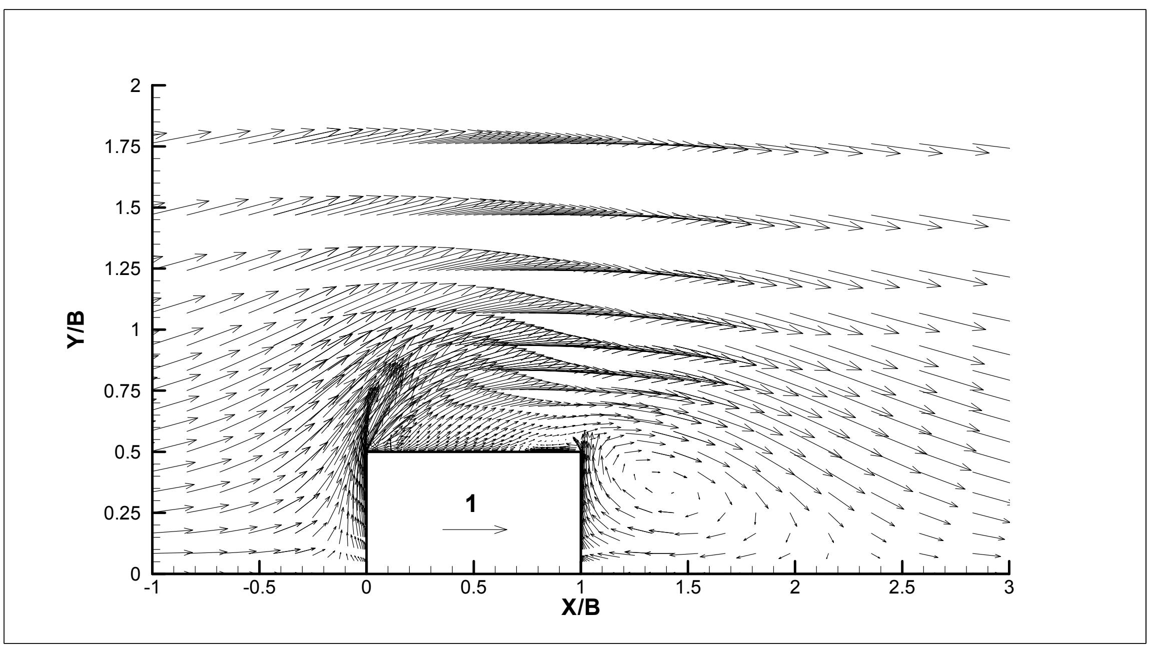


Figure 4. Velocity vector around rectangular pier at free surface (L/B=1)

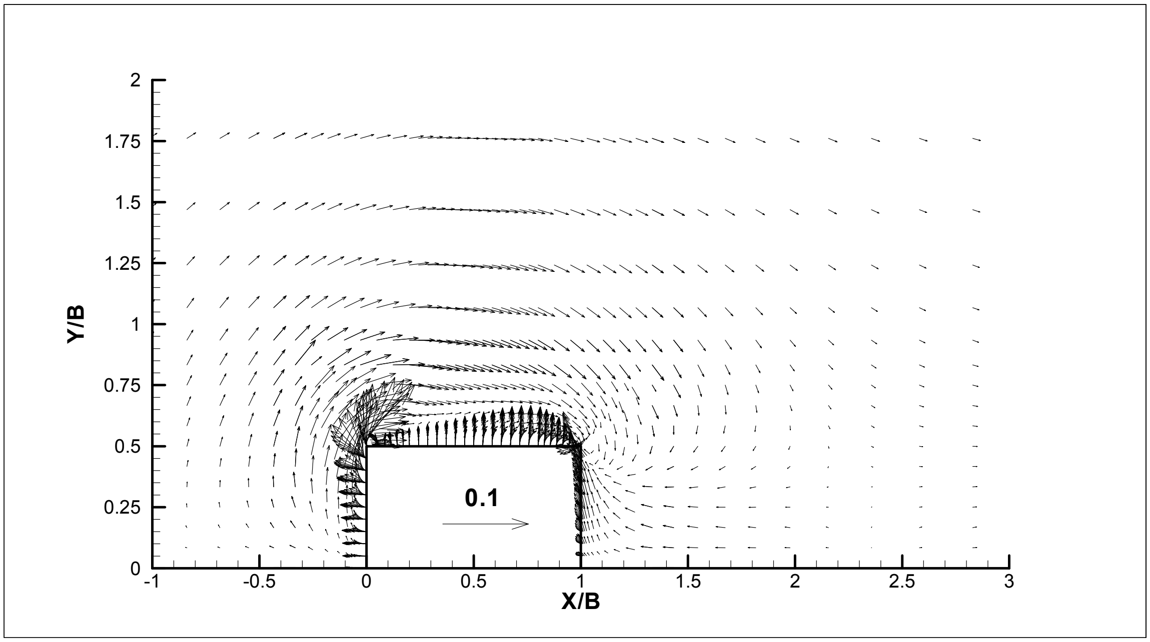


Figure 5. Velocity vector around rectangular pier at z/B=0.0001 above riverbed (L/B=1)

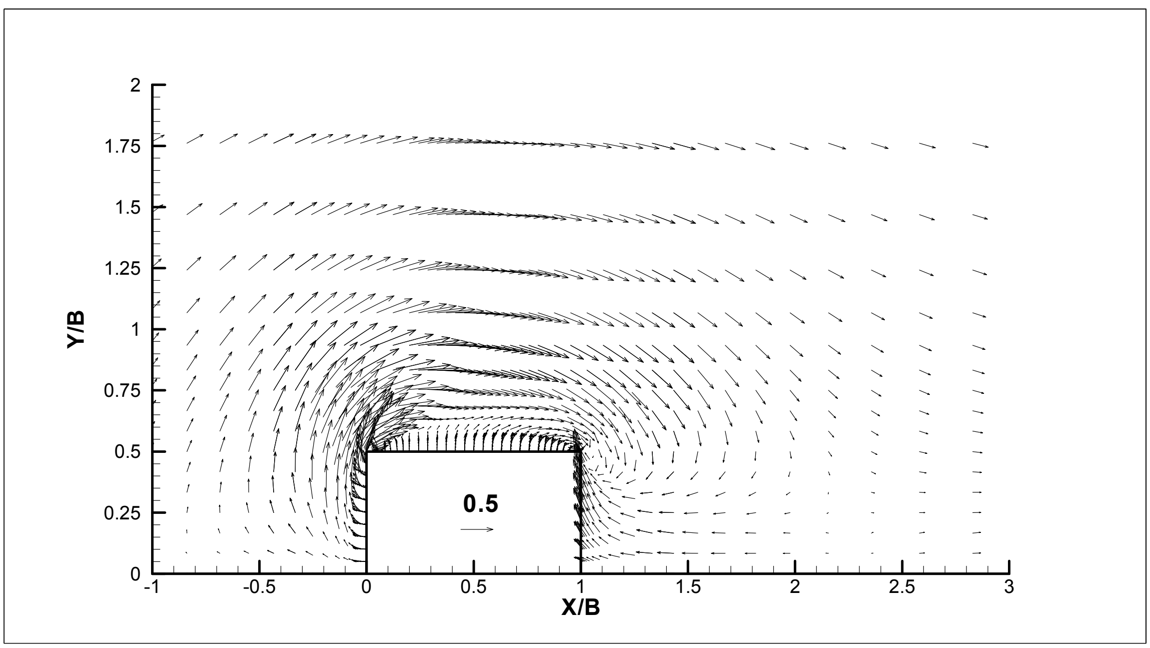


Figure 6. Velocity vector around rectangular pier at z/B=0.004 above riverbed (L/B=1)

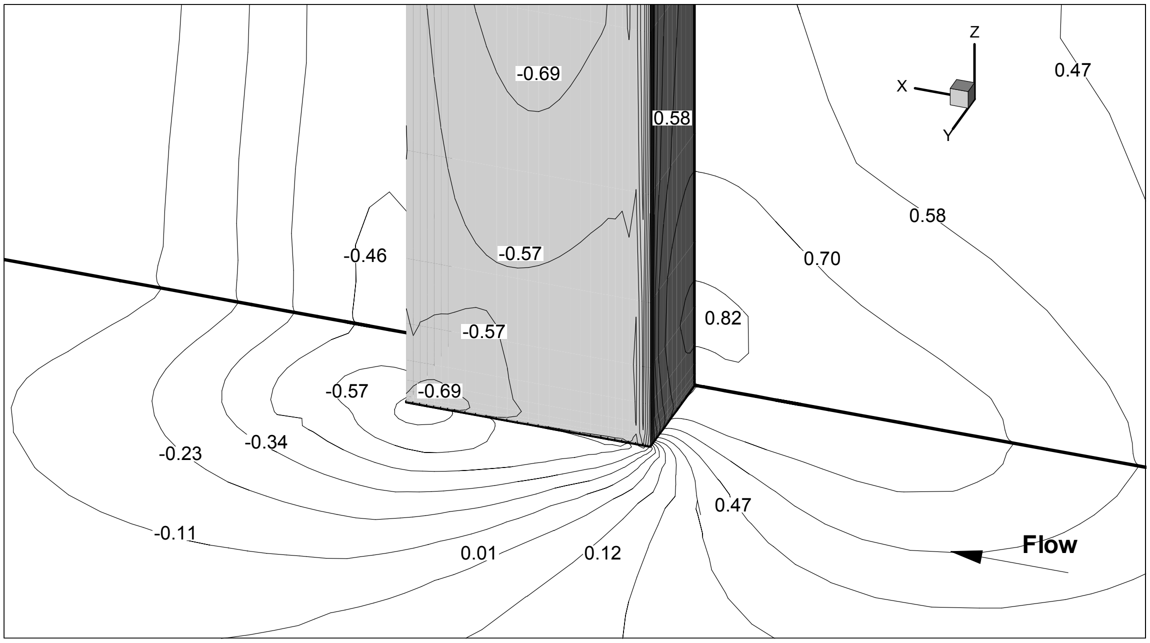


Figure 7. Pressure contour around rectangular pier (L/B=1) at initial condition

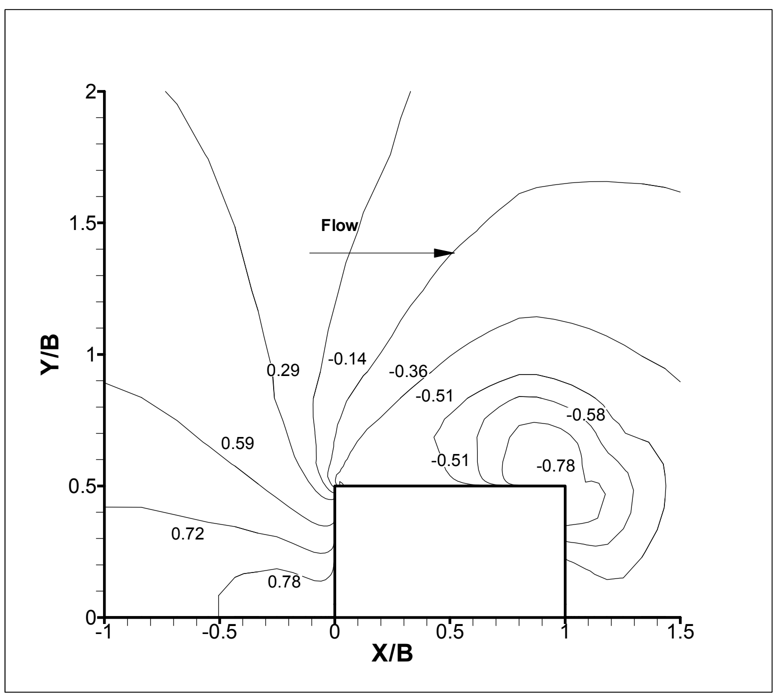


Figure 8. Pressure contour around rectangular pier (at bed) for L/B=1

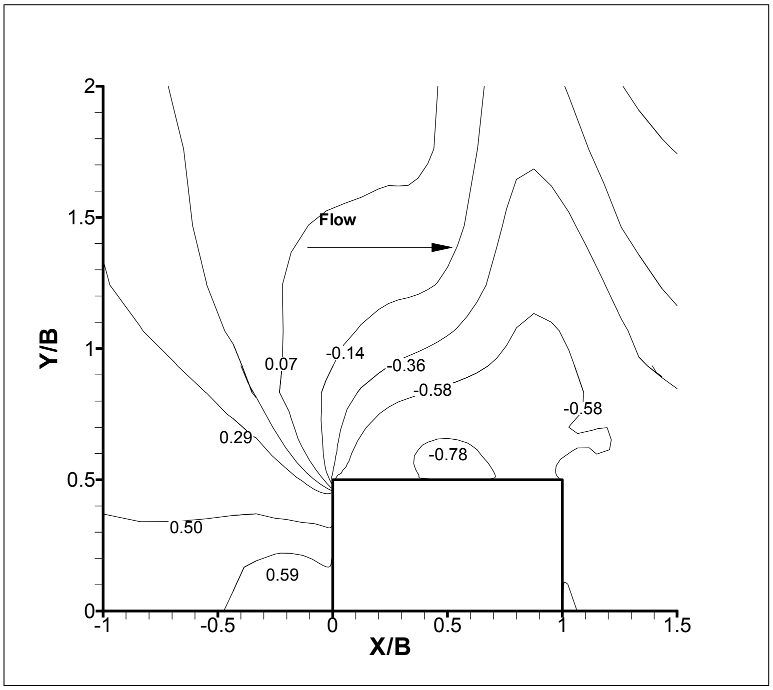


Figure 9. Pressure contour around rectangular pier (at free surface) for L/B=1

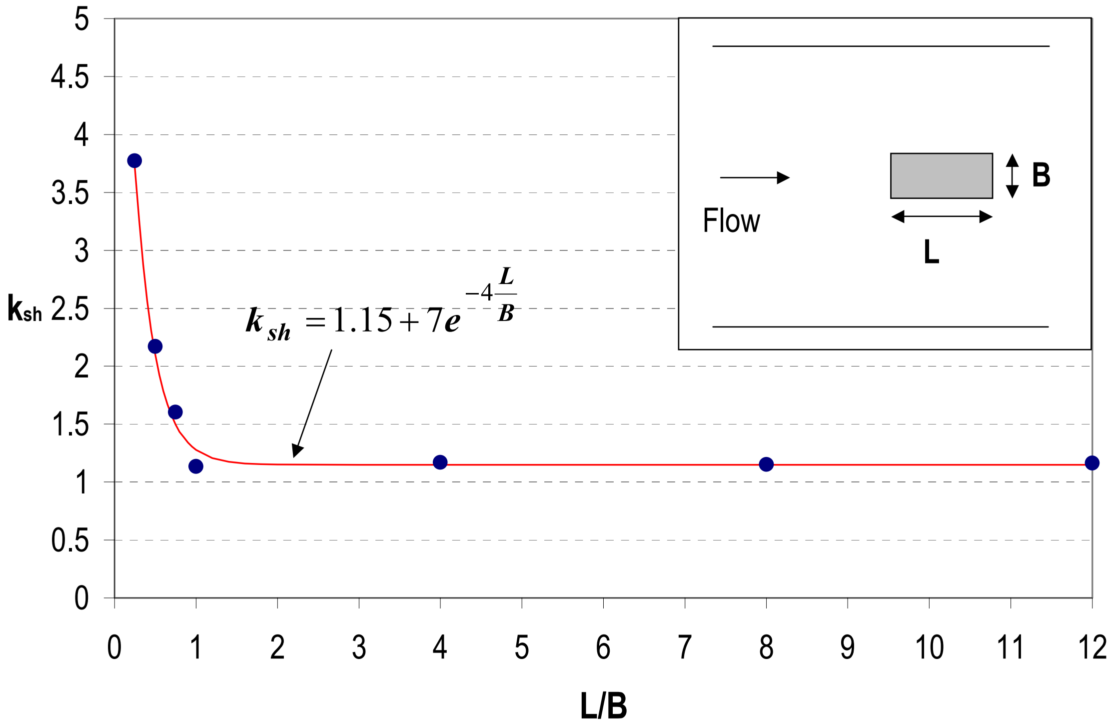


Figure 10. The relation between $k_{sh}$ and **L/B** at deep water (y/D>2), $k_{sh} = \frac{\tau_{\max}}{\tau_{\max}(circle)}$

Figure 10 shows the relationship between the ratio of the maximum bed shear stress and rectangular pier for different L/B. It is found that the correction factor for shape (rectangular shape) is 1.15 only for L/B>1. The value is no longer valid when L/B is less than 1. The flow pattern around the rectangular pier for L/B=0.25 is quite different from the pattern for L/B=4. The flow is separating at the sharp corner. Since the length of pier (L) is smaller than the wide of pier (B), the case of L/B=0.25 allows the flow to go behind the pier while the flow for L/B=4 follows the length of the pier. In the case of L/B=4, the length of separated flow is about B hence we found similar flow behavior for L/B>1. Beyond the separated flow, the velocity remains uniform. On the contrary, for case of L/B<1, there is no region of separated flow and the decreasing pressure behind the pile may increase the velocity around the corner. Bed shear stress contours around the pier are shown in figure 11, 12, 13 and 14 for L/B = 0.25, 0.75, 1, and 4. The difference of flow pattern and pressure around the two rectangular pier (L/B=0.25 and L/B=4) are shown in figure 15, 16, 17 and 18.

The correction factor for the effect of pier shape is $k_{sh}$ defined as $k_{sh} = \frac{\tau_{\max}(L/B)}{\tau_{\max}(circle)} = 1.15 + 7e^{-4\frac{L}{B}}$ for rectangular pier, and for circular pier $k_{sh} = 1$. The equation for maximum bed shear stress around rectangular pier can be written as:

$$\tau_{\max} = k_{sh} 0.094 \rho V^2 \left[ \frac{1}{\log \mathrm{Re}} - \frac{1}{10} \right] \tag{4}$$

## CONCLUSION

A chimera RANS method has been employed in conjunction for simulating bed shear stress in soils around rectangular pier. A smooth interface was assumed (cohesive soils). The method successfully resolved many important flow features for local scour around bridge piers including the formation of horseshoe vortices. The ratio of length of pier (L) over width of pier (B) may be ignored for L/B greater than 1. However, for L/B less than 1, the effect of L/B becomes significant. The correction factor is included into the equation of bed shear stress to capture the effect of a rectangular pier shape with different L/B ratio.

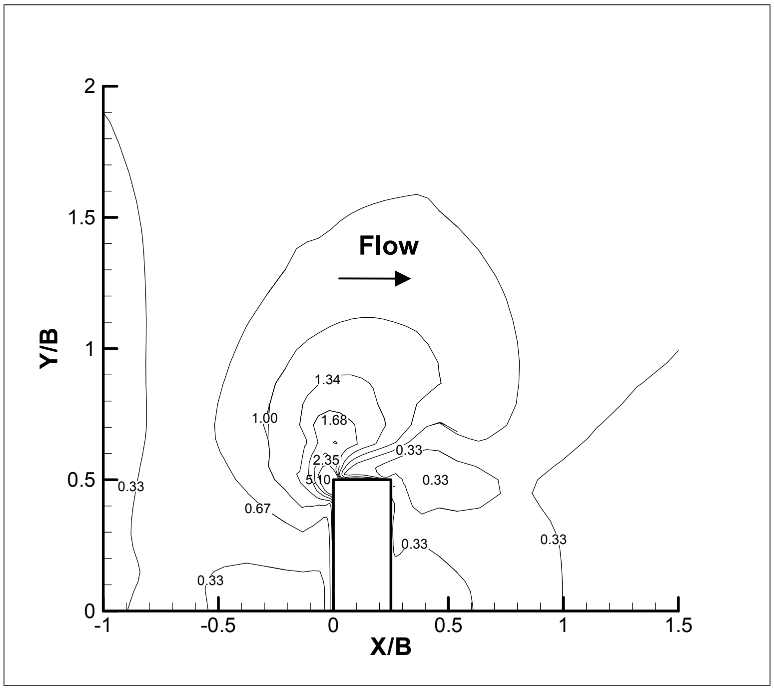


Figure 11. Bed shear stress contour ($N/m^2$) around rectangular pier for L/B=0.25

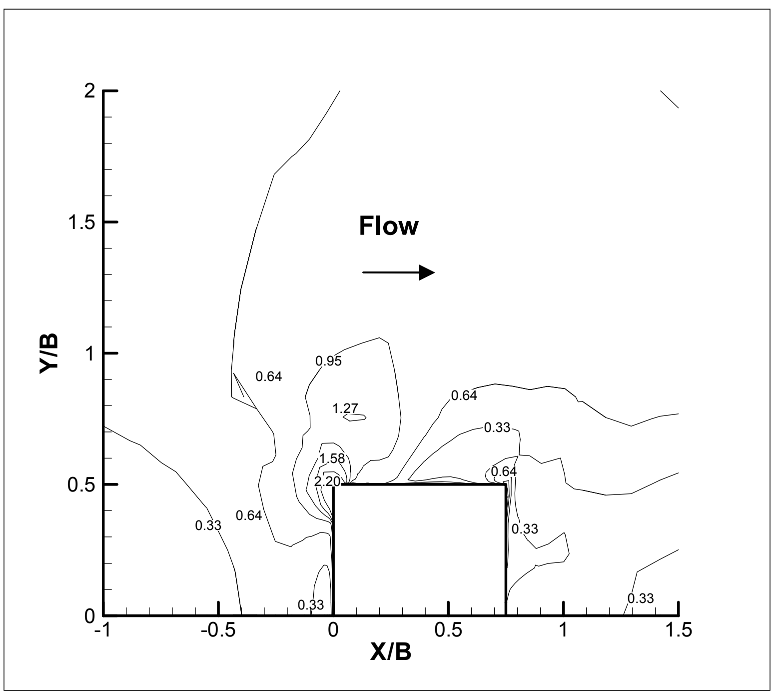


Figure 12. Bed shear stress contour ($N/m^2$) around rectangular pier for L/B=0.75

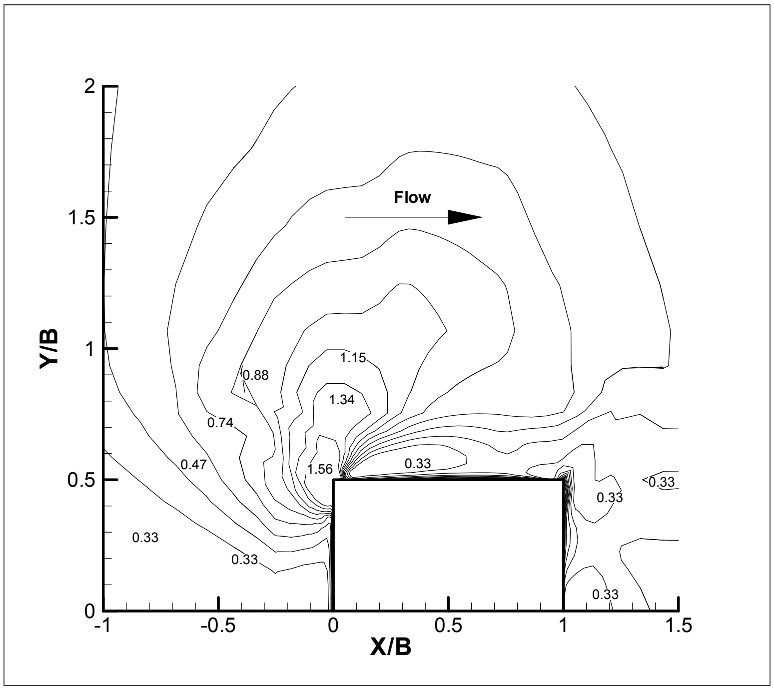


Figure 13. Bed shear stress contour ($N/m^2$) around rectangular pier for L/B=1

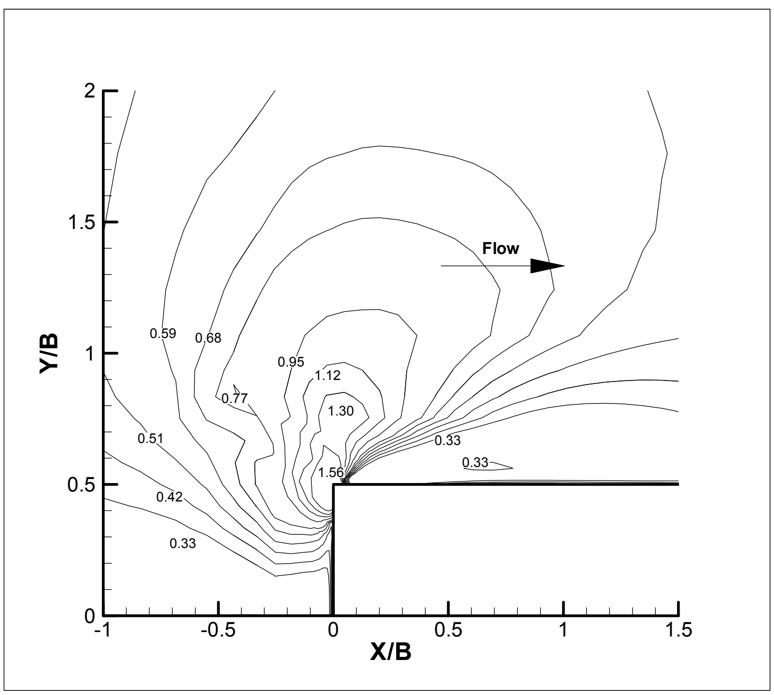


Figure 14. Bed shear stress contour ($N/m^2$) around rectangular pier for L/B=4

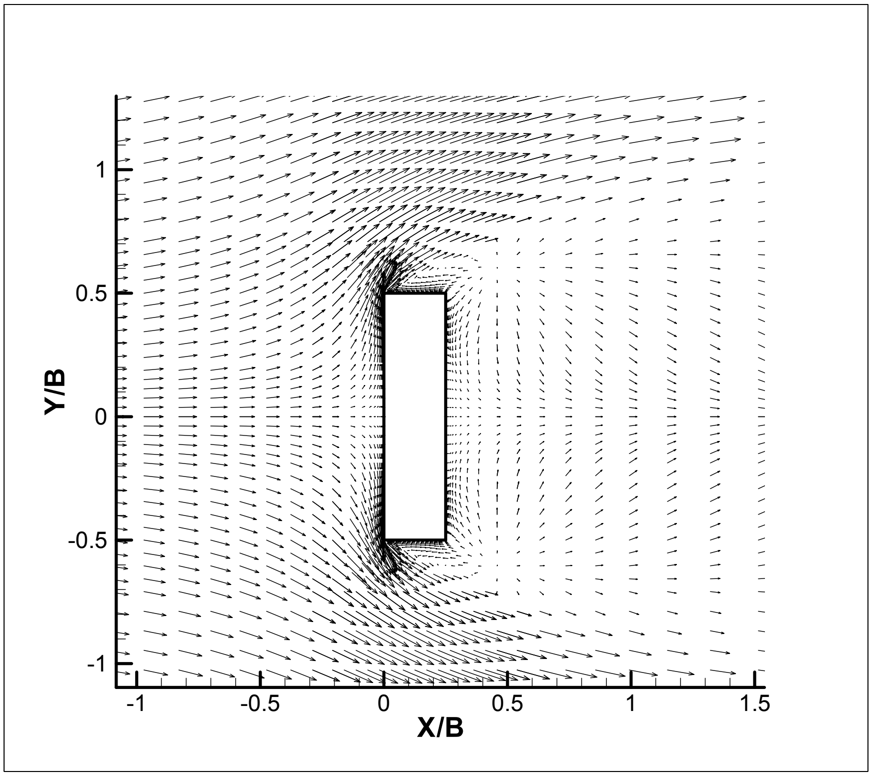


Figure 15. Velocity vector around rectangular pier for L/B=0.25

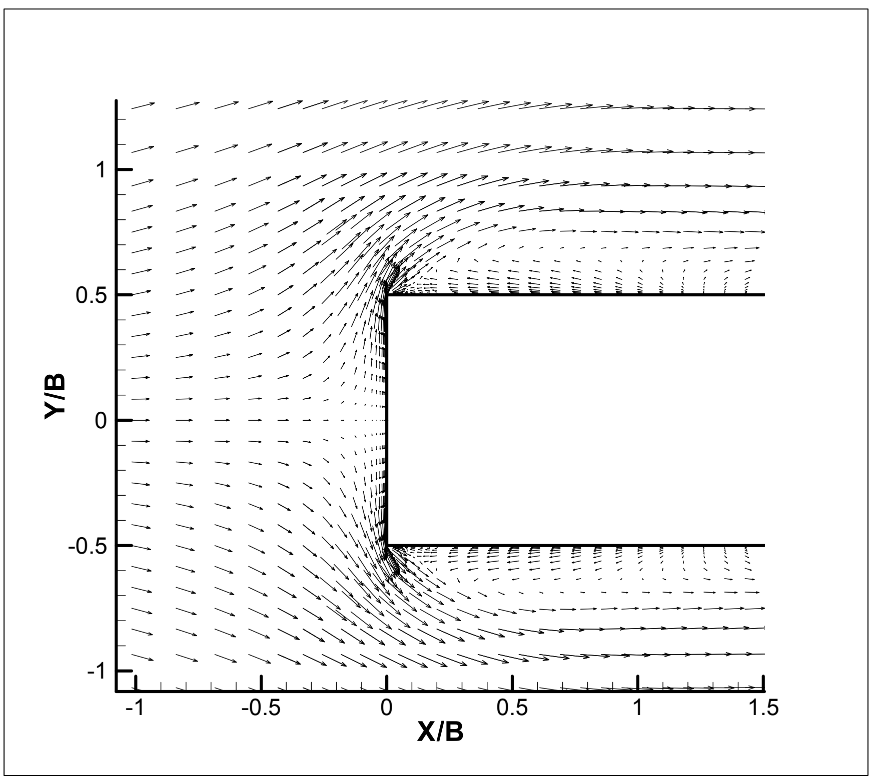


Figure 16. Velocity vector around rectangular pier for L/B=4

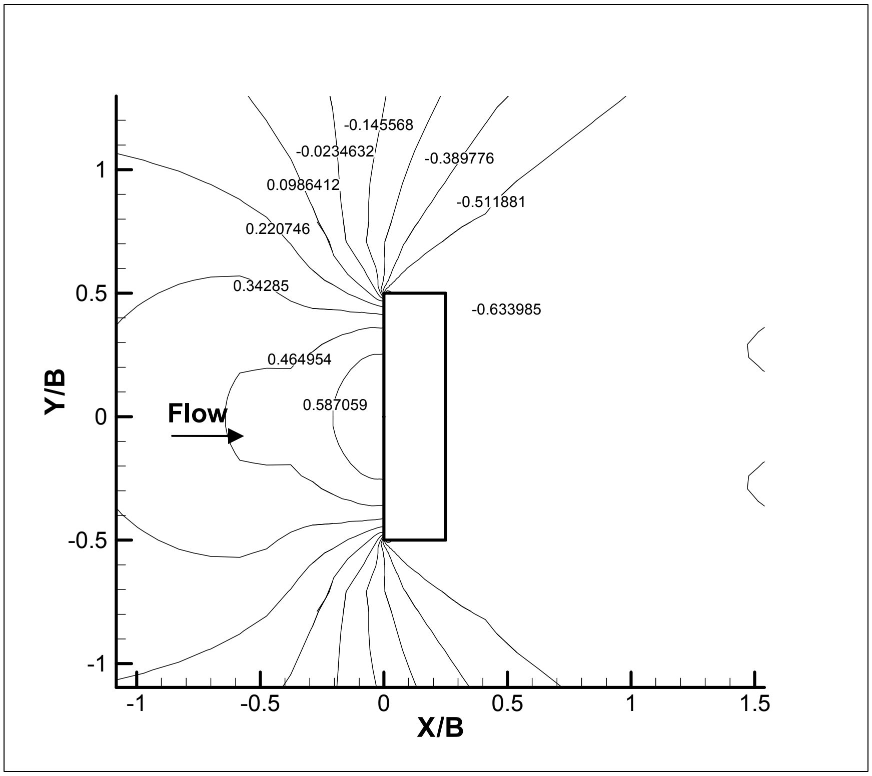


Figure 17. Normalized Pressure ( )contour around rectangular pier for L/B=0.25

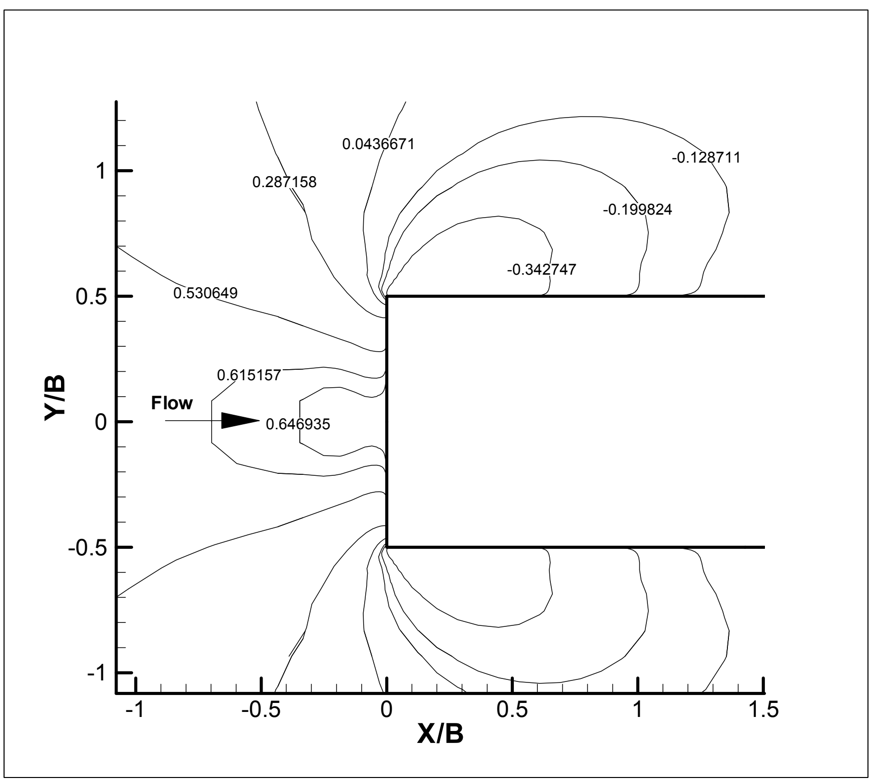


Figure 18. Pressure contour around rectangular pier for L/B=4

## ACKNOWLEDGMENT

This work was supported by NCHRP-Project 24-15 where Mr. Tim Hess, the contact person, was very helpful.

## REFERENCES

1. Briaud J.-L., Ting F., Chen H.C., Cao Y., Han S.-W., Kwak K., 2001a, "Erosion Function Apparatus for Scour Rate Predictions", Journal of Geotechnical and Geoenvironmental Engineering, Vol. 127, No.2, pp. 105-113, Feb. 2001, ASCE, Reston, Virginia.
2. Briaud J.-L., Chen H.-C., Kwak K., Han S., Ting F., 2001b, "Multiflood and Multilayer Method for Scour Rate Prediction at Bridge Piers", Journal of Geotechnical and Geoenvironmental Engineering, Vol. 127, No.2, pp. 114-125, Feb. 2001, ASCE, Reston, Virginia.
3. Chen, H. C. and Chen, M (1998),"Chimera RANS Simulation of a Berthing DDG-51 Ship in Translational and Rotational Motions." International Journal of Offshore and Polar Engineering, Vol. 8, No.3, pp. 182-191.
4. Chen, H. C. and Patel, V. C. (1988), " Near Wall Turbulence Models fro Complex Flows Including Separation, " AIAA Journal, Vol. 26, No. 6, pp. 641-648.
5. Chen, H. C., Patel , V. C. and Ju, S. (1990),"Solution of Reynolds-Averaged Navier-Stokes Equations for Three-Dimensional Incompressible Flows," Journal of Computational Physics, Vol. 88, No. 2, pp. 305-336.
6. Dou, X. "Numerical Simulation of Three-dimensional Flow Field and Local Scour at Bridge Crossings" Ph.D. Dissertation, 1997, The University of Mississippi.
7. Hoffmans, G. J. C. M. and Booij, R., "Two-dimensional Mathematical Modeling of Local Scour Holes", Journal of Hydraulic Research, Vol. 31, No.5, 1993.
8. Molinas, A. " Bridge Stream Tube Model for Alluvial River Simulation (BRI-STARS)", user's Manual, National Cooperative Highway Reaserch Program, Project No. HR 15-11, Transportation Research Board, 1990, Washington, D. C.
9. Olsen, N. R. B. and Melaaen, M. C. "Three-dimensional Calculation of Scour Around Cylinder", Journal of Hydraulic Engineering, Vol. 119, No. 9, 1993, pp.1048-1054.
10. Roulund, A., Sumer, B. M., Fredsoe, J., and Michelsen, J., "3D Mathematical Modeling of Scour Around A Circular Pile", ISRS 98, December 16-18, 1998, Hong Kong.
11. Wei, G., Chen, H. C., Ting, F, Briaud, J.-L, Gudavalli, S. R. and Perugu, S., "Numerical Simulation to study Scour Rate in Cohesive Soils" Research report to the Texas Department Transportation, 1997, Department of Civil Engineering, Texas A&M University, College Station, TX.