Numerical Simulation of Fluid Structure Interaction Between Wave and Pile in Deep Water

Wenyue Yan
Department of Building Engineering, College of Civil Engineering, Tongji University, 1239 Siping Road, Shanghai, P.R. China
1432043ywy@tongji.edu.cn

Fengwu Lv
Department of Building Engineering, College of Civil Engineering, Tongji University, 1239 Siping Road, Shanghai, P.R. China
fengwur@tongji.edu.cn

***Abstract*—The $k-\varepsilon$ turbulence model and the VOF method of moving interface tracking have been used to generate the wave. By two-way iterative method，the fluid structure interaction between wave and steel pipe pile platform has also been simulated numerically by using ANSYS Workbench. And a comparison among the results of wave force in one-way coupling and two-way coupling as well as code method has been made, thus a reference is provided for the calculation of wave in engineering. And four sets of influence coefficients were simulated respectively, which include the transverse coefficient of three parallel piles and the shelter coefficient of three tandem piles, and a comparison of transverse coefficient among numerical simulation and code has been made. It is found that the wave shelter effect on front and rear piles is greater than that of the middle pile, and the transverse coefficient of the parallel piles is affected by the wave superposition effect.**

Keywords—VOF method; the two-way coupling; wave; transverse coefficient of piles; shelter coefficient of piles

I. INTRODUCTION

The trend of ocean development is towards deep waters. People have been building many structures in deep water in recent years, such as oil drilling platforms, offshore wind power, ports and bridges crossing seas and rivers. The wave plays an important role in safety, cost and lifespan of the construction in the deep water [1]. In practical engineering, most of small piles are in the form of pile groups, so the interaction between pile group and wave makes the wave field more complicated. With the development of engineering in deep water, the research on the coupling effect of wave and structure has attracted more attention.

It's a tendency to study fluid structure coupling by means of numerical simulation, as a result of rapid development in calculation. Xiaoling Zhu [2] used ANSYS and CFX to simulate two-way fluid-structure interaction for tandem double piles and wave field. Integrating with the simulation results of the single pile, the maximum equivalent stress of both was compared. Chen Yang [3] simulated the interaction between waves and two vertical and two horizontal piles. Xinxin Lei [4] studied hydrodynamic characteristics of the five piles structure under the wave action, by the means of physical model test and numerical simulation. Shengrong Zuo [5] simulated the role of circular piers by combing the action of waves and uniform flow and analyzed the response of the piers. Shan Zhong [6] studied the simulation analysis of the wave surface coupling, the dorsal surface coupling and the whole coupling respectively.

Documents above mainly focus on numerical simulation of waves and fluid structure coupling between wave and one or two piles, but few researchers study the coupling effect between wave and three piles. In this paper, the interaction between water flow and steel pipe piles is taken into consideration by establishing two-way fluid solid coupling models in ANSYS Workbench platform, and a study is made about the transverse coefficient of three parallel piles and the shelter coefficient of three tandem piles, which provides a reference for future engineering practice.

II. MODEL AND CONTROL EQUATIONS

A. *Equation of Fluid Solid Coupled System Field*

Fluid is assumed to be free viscous, compressible and small perturbations, and fluid free surface is small fluctuation; and solid is assumed to be linear elastic. The finite element analysis method of the system constituted with the two ideal materials above has two types: Displacement-Displacement form and Displacement-Potentials form. As the Displacement-Potentials form is more efficient, it is highly used in engineering now. It means that the displacement u_i is used as basic unknown quantity in the equation of solid domain, while the flow field pressure p is used as basic unknown quantity in the equation of fluid domain. The corresponding finite element expression format is called displacement-pressure (u_i, p) scheme for fluid solid coupling analysis [7].

1) Fluid Field

a) Fluid Field Equation

$$p_{,ii} - \frac{1}{c_o^2}\ddot{p} = 0 \qquad (1)$$

where p is fluid pressure, c_o is sound velocity in fluid.

a) Fluid Boundary Conditions

Rigid fixed boundary:

$$\frac{\partial p}{\partial n_f}=0 \tag{2}$$

Free surface:

$$\frac{\partial p}{\partial z}+\frac{1}{g}\ddot{p}=0 \tag{3}$$

2) Solid Field

a) Solid Field Equation

$$\sigma_{ij,j}+f_i=\rho_s\ddot{u}_i \tag{4}$$

where σ_{ij} is solid stress component, u is solid displacement component, f_i is solid volume force component, ρ_s is solid mass density.

b) Solid Doundary Conditions

Force boundary:

$$\sigma_{ij}n_{sj}=\overline{T_i} \tag{5}$$

Displacement boundary:

$$u_i=\overline{u_i} \tag{6}$$

where $\overline{T_i}$ is known surface force component on solid, $\overline{u_i}$ is known displacement components on solids.

3) Compatibility Conditions on the Fluid Solid Coupling Interface

a) Kinematic Condition: The normal velocity of the fluid solid interface should be continuous, which means

$$\frac{\partial p}{\partial n_f}+\rho_f\ddot{u}\cdot n_f=0 \tag{7}$$

where u is solid displacement vector, ρ_f is fluid mass density.

b) Dynamitic Condition: Normal stress should be continuous on the interface of fluid solid interface, i.e.

$$\sigma_{ij}n_{sj}=pn_{sj} \tag{8}$$

B. Turbulence Model

For the fluid structure coupling problem in deep water, considering the fluid as turbulent fluid, the additional turbulent equation is needed. In this paper, the standard $k-\varepsilon$ turbulence model is used.

Equation of fluctuating kinetic energy k is:

$$\frac{\partial(\rho k)}{\partial t}+\nabla\cdot(\rho\boldsymbol{u}k)=\nabla\cdot\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\nabla k\right]+P_k-\rho\varepsilon+S_k \tag{9}$$

P_k is defined by

$$P_k=\mu_t\nabla\boldsymbol{u}\cdot\left(\nabla\boldsymbol{u}+\nabla\boldsymbol{u}^T\right)-\frac{2}{3}\nabla\cdot\boldsymbol{u}\left(3\mu_t\nabla\cdot\boldsymbol{u}+\rho k\right) \tag{10}$$

Where μ_t is eddy viscosity,

$$\mu_t=C_\mu\rho\frac{k^2}{\varepsilon} \tag{11}$$

Equation of turbulent dissipation rate ε is:

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\nabla\cdot(\rho\boldsymbol{u}\varepsilon)=\nabla\cdot\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\nabla\varepsilon\right]+\frac{\varepsilon}{k}\left(C_{\varepsilon1}P_k-C_{\varepsilon2}\rho\varepsilon\right)+S_\varepsilon \tag{12}$$

Here, S is custom source term.

In $k-\varepsilon$ turbulence model, C_μ, σ_k, σ_ε, $C_{\varepsilon1}$, $C_{\varepsilon2}$ are empirical constants, whose values are given by [8].

C. Wave Theory

The linear wave equation is used for the wave numerical simulation in this paper:

$$\eta=\frac{H}{2}\cos(kx-\omega t) \tag{13}$$

The velocity equation of wave motion is:

$$\begin{cases} u=\dfrac{\partial\varphi}{\partial x}=\dfrac{kHg}{2\omega}\dfrac{\mathrm{ch}k(h+z)}{\mathrm{ch}kh}\cos(kx-\omega t) \\ w=\dfrac{\partial\varphi}{\partial z}=\dfrac{kHg}{2\omega}\dfrac{\mathrm{sh}k(h+z)}{\mathrm{ch}kh}\sin(kx-\omega t) \end{cases} \tag{14}$$

The pressure of the linear wave is as follows:

$$p=-\rho gz-\rho g\frac{H}{2}\frac{\mathrm{ch}k(h+z)}{\mathrm{ch}kz}\cos(kx-\omega t) \tag{15}$$

In the above Eqs, k is the number of waves per second, and $k=\frac{2\pi}{\lambda}$, λ is the wavelength; ω is the circle frequency of wave; H is the wave height;

h is the distance of the fixed boundary to the water surface; g is the gravity acceleration.

D. VOF Method

The volume of fluid (VOF) is a free interface tracking method, and it has advantages in reducing the amount of calculation, as well as ensuring the quality of conservation. Furthermore, it can deal with large deformation of the free surface without any numerical dissipation. Therefore it is selected to trace wavefront.

In the VOF method, the volume fraction function ϕ_q is introduced, which represents the ratio of the phase q fluid volume to the whole unit volume. $\phi_q=1$ indicates that all of the units are q phase material, $\phi_q=0$ indicates that q phase material is empty in the units, $0<\phi_q<1$ indicates that the unit is an interface unit.

Wave in the deep water is gas-liquid two-phase, so the governing equations are as follows:

$$\frac{\partial\phi_q}{\partial t}+\frac{\partial(u\phi_q)}{\partial x}+\frac{\partial(v\phi_q)}{\partial y}=0 \tag{16}$$

$$\sum_{q=1}^{2}\phi_q=1 \tag{17}$$

III. NUMERICAL SOLUTION METHOD

The partial differential equations of fluid structure coupling shown above are quite complicated. It's difficult to get the analytical solution of these equations. However, it is common to use numerical analysis method by computer to solve differential equations with a given boundary condition. Commonly used numerical analysis methods are Finite Difference Method (FDM), Finite Element Method (FEM) and Finite Volume Method (FVM). In this paper FEM in ANSYS Structure is used to solve the solid equations, while FVM in ANSYS CFX is used to solve the fluid equations.

Displacement-Pressure form is used as the finite element analysis model in this paper. The solid transmits its displacement to the fluid, considering the nodal displacement as boundary condition, the equation that uses the pressure of the flow field as the unknown variable can be solved. And then the flow transmits the boundary pressure to the solid, which considers nodal displacement as the unknown quantity of equations. After solving the nodal displacement, it is transmitted to the fluid, which will become the boundary condition of fluid field, and then circulate step by step until convergence.

This paper use ANSYS Workbench to carry out numerical simulation of fluid solid coupling. Workbench use ANSYS CFX and ANSYS Structure, respectively, to solve the fluid equation and solid equation in the same step, and then use MFX procedure of ANSYS to exchange the data on the fluid solid coupling surface. The solid and fluid fields are calculated according to the updated boundary conditions. The specific implementation thought is shown as Fig. 1.

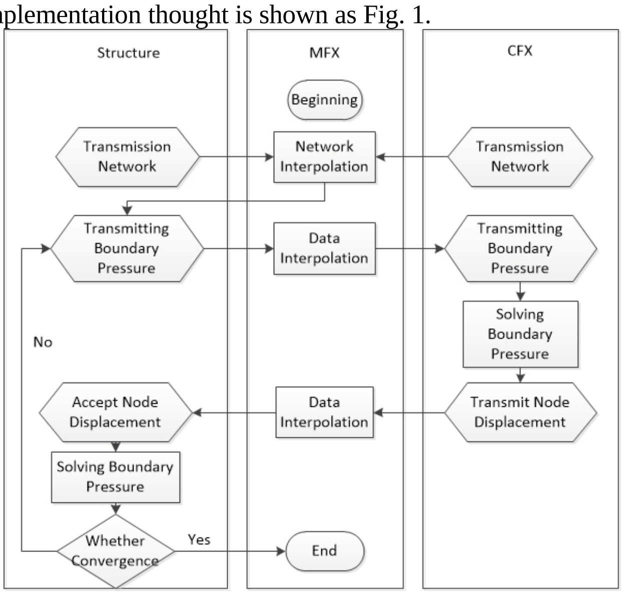

Fig. 1. The Realization of Fluid Structure Coupling

The number of iteration is controlled by the convergence accuracy. And it is taken as the convergence control criterion when RMS is less than 1e-4 in this paper.

IV. WAVE NUMERICAL SIMULATION

A. *Wave Parameter Setting*

The data for setting wave parameter is from Xu Liu Jing hydrological observation station in the Yangtze River Estuary. So the wave height is 0.7m, the wavelength is 16m, the depth is 25m. According to experience, the tank size is 100 m * 14m * 30m, and the static water depth is 25m.

The hexahedral automatic meshing is used to mesh the numerical tank. The size control is used in a wave height range from static water, as shown in Fig. 2. The number of grid cells is 822500, and the number of nodes is 3397379, and the average quality of the grid is 0.84.

The boundary setting is shown in Fig. 3. In order to avoid backflow, the opening boundary condition is used both in entrance and exit.

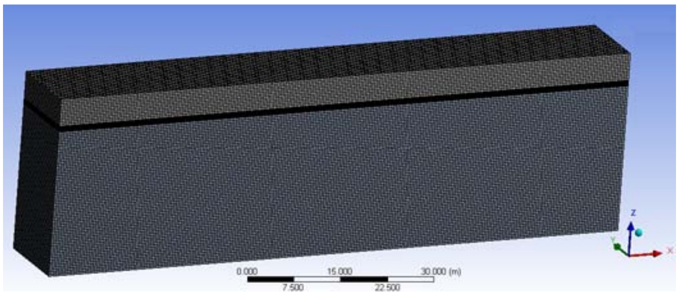

Fig. 2. Tank Meshing

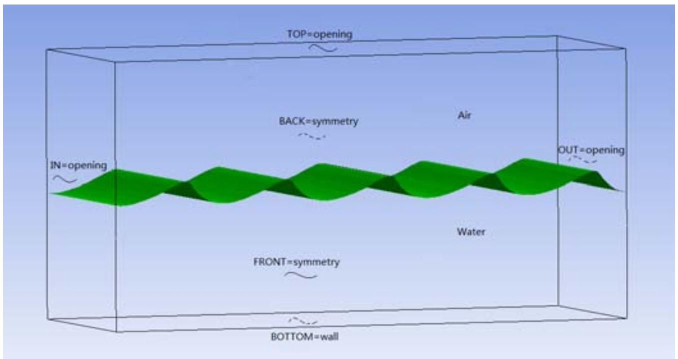

Fig. 3. Tank Boundary

B. *Wave Validation*

The wave tank should be verified before the simulation of two-way coupling between wave and steel pipe pile. The simulation value is compared with the theoretical calculation of the wave position. Fig. 4 shows the simulation wave and theoretical wave at 2.4s. The computer simulated curve depicts similar trend as that of the theoretical curve. The maximum difference is 0.28% and the average difference is 0.009%.

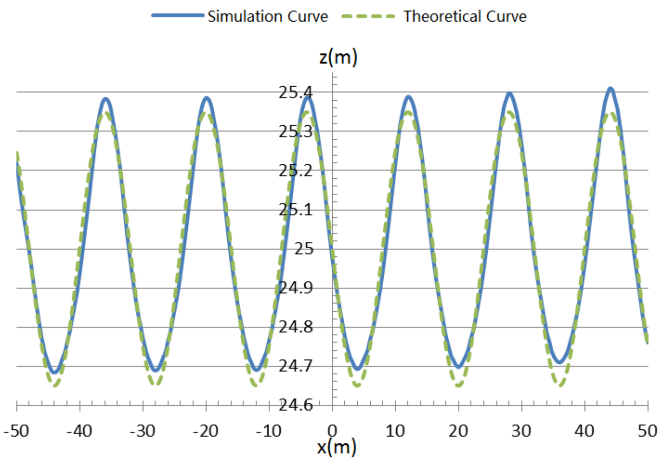

Fig. 4. Comparison between Simulation and Theory of Wavefront Position (t=2.4s)

V. CASE ANALYSIS

This paper takes a steel pipe pile foundation of a construction platform in the Yangtze River Estuary as an example to analysis fluid solid coupling. And the 30m-high steel pipe pile embedded in river bed, whose diameter is 2.54m and thickness is 0.06m. And the hydrological conditions are the same as the section IV.

A. Single Pile Analysis

1) Wave velocity distribution

The basic law of the distribution of velocity on wavefront can get from Fig. 5 and Fig. 6.

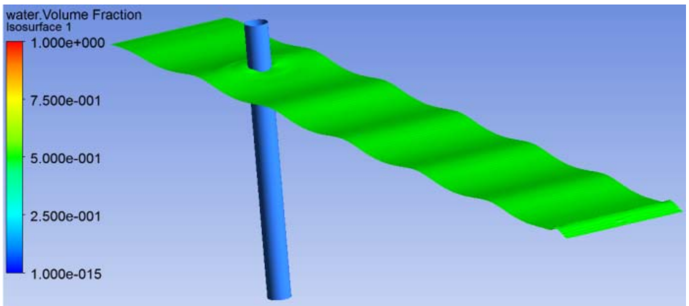

Fig. 5. The Couping Between Wave and Steel Pipe Pile

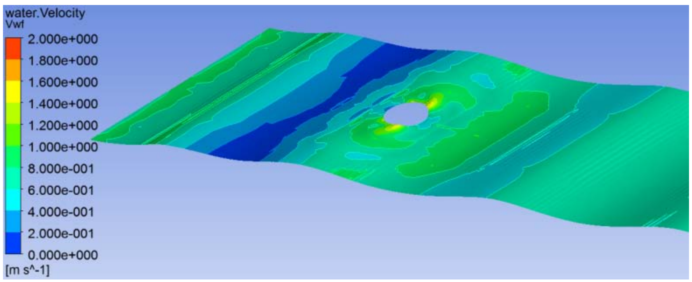

Fig. 6. Velocity on Wavefront

2) Comparison of Stress Calculation Results of Pile

Making a comparison of the stress at the bottom of the pile, the results of three calculation methods are showed as Fig. 7 and TABLE I. Moreover, the influence of the solid deformation to the water field is neglected in the one-way coupling, and the wave force is regarded as static force in the code method [9].

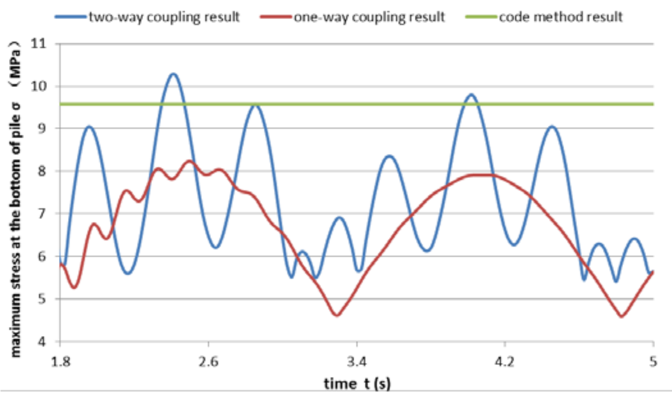

Fig. 7. Maximum Stress-Time Contrast

TABLE I. COMPARISON OF MAXIMUM STRESS

Computing Method	one-way coupling	two-way coupling	code method
Maximum Stress of Pile σ [MPa]	8.24	10.3	9.57

it can be known that the calculating results of the code method and two-way coupling are very close to, whose difference is only 7.1%; while the results of one-way coupling is smaller than that of two-way coupling by 20%.

It shows that the stiffness of steel pipe pile is large, and the deformation of the pile is not obvious, and the code method of current force is partial safety.

B. The Shelter Influence Analysis of Tandem Pile

L stands for spacing of piles and D is the diameter of the steel pipe pile. Setting the shelter coefficient of wave is :

$$m_i = \frac{F_i}{F_{si}} = \frac{\sigma_i}{\sigma_{si}} \tag{18}$$

F_i is the wave force of steel pipe pile of No.i; F_{si} is the wave force of isolated steel pipe pile; σ_i is the maximum stress of steel pipe pile of No.i; σ_{si} is the maximum stress of isolated steel pipe pile, and that is 10.3MPa.

By taking L/D=2, 3, 4, 5, the shelter coefficients of isometric three piles by numerical simulation can be seen in Fig. 8.

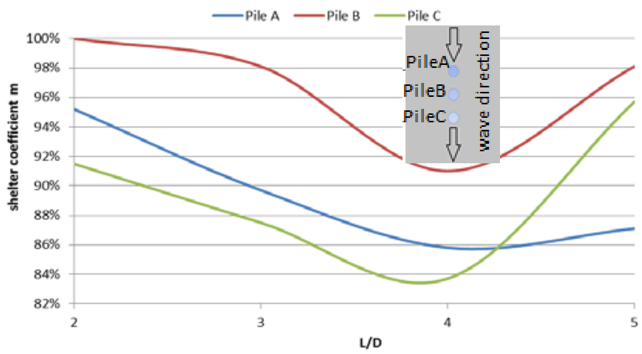

Fig. 8. The Comparison of Shelter Coefficient

The Fig. 8 shows that when the pile spacing L is equal to 4D, the shelter effect on wave force is maximal, and the shelter coefficient is the smallest; When the pile spacing L is less than 4D, the L is larger, the shelter influence is larger, the shelter coefficient is smaller; When the pile spacing L is larger than 4D, the L is larger, the shelter influence is smaller, the shelter coefficient is closer to 1, and the shelter coefficient variation of pile C is largest.

The shelter influence of parallel piles is ignored in the code method, which means shelter coefficient is equal to 1 in the code. However, the numerical simulation result shows that the shelter effect on wave force is great, which is different with the code. That is because the water depth is so large that the difference of the wave force will have a great impact on the stress of the pile.

Thus, it is suggested that the calculation factors of code should be refined, when the water depth is large, especially for the front and rear piles of tandem pile.

C. The Transverse Influence Analysis of Parallel Piles

Setting the transverse coefficient of flow is:

$$m_i' = \frac{\sigma_i}{\sigma_{si}} \qquad (19)$$

By taking L/D=2, 3, 4, 5, the transverse coefficient of isometric three piles by numerical simulation can be observed in Figure 5.

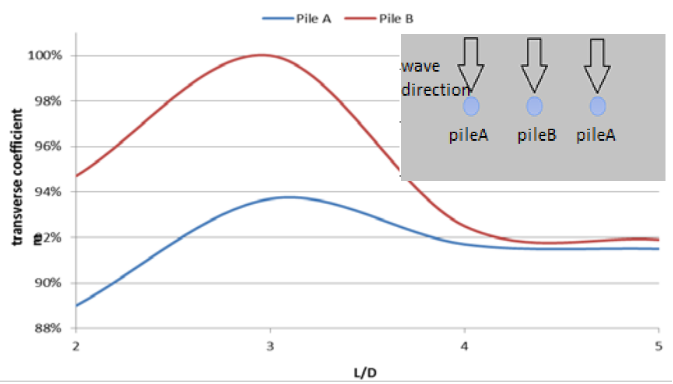

Fig. 9. The Comparison of Transverse Coefficient

It can be seen that when the pile spacing L is equal to 3D, transverse coefficient is maximal; when the pile spacing L is less than 3D, with the increase of pile spacing, transverse coefficient increases; when the pile spacing is larger than 3D, transverse coefficient decreases with the increase of pile spacing.

As the increase of pile spacing, the transverse influence of side pile and middle pile is closer to each other. When the pile spacing is larger than 4D, the transverse influence of side pile and middle pile is almost the same.

VI. CONCLUSIONS

- It's feasible to use VOF method and $k-\varepsilon$ turbulence model for numerical simulation of wave. The simulation wave curve coincides well with the theoretical curve.
- The effect of the two-way coupling of pile and wave in deep water area depends on the stiffness of pile. When the stiffness of pile is large, the results of code method and two-way coupling are close to. Considering the two-way coupling consumes great computational resources, Morison equation is recommended when calculating wave force of single pile or other simple structure.
- Wave shelter effect on front and rear piles is greater than that of the middle pile. And the shelter effect on wave force is maximal, when the pile spacing L is equal to 4D. And it is suggested that the calculation factors of code should be refined, when the water depth is large.
- Wave transverse effect on side piles is greater than that of the middle pile. And the transverse effect on wave force is minimal, when the pile spacing L is equal to 4D. With the increase of pile spacing, the lateral influence of side pile and middle pile is closer and closer. When the pile spacing is larger than 4D, the transverse influence of side pile and middle pile is almost the same.

References

[1] Yuanfang Liu, Xiufen Huang, Jia Hao Lin. Dynamic Safety Checking of Fixed Jacket Platform [J]. Computational Structural Mechanics and Applications, 1990,7 (3) :73-83.

[2] Xiaoling Zhu. A Study on Fluid-solid Coupling Simulation of Offshore Platform Pile Foundation Structure [D]. North China Electric Power University, 2014.

[3] Chen Yang. Numerical Simulation of wave forces on small-diameter cylinders using three-dimension Unsteady Reynolds Equation Model [D]. Tianjin University, 2006.

[4] Xinxin Lei. Research on Hydrodynamic Performance of Pile Array Under Wave Action[D]. Dalian University of Technology, 2013.

[5] Shengrong Zuo. Research on Wave Response of Deepwater Pier [D]. Wuhan University of Technology, 2013.

[6] Shan Zhong. The Dynamic Response Analysis of Pier Under The Action of Wave Based on Fluid-Solid Coupling [D]. Chongqing Jiaotong University, 2013

[7] Yanrong Zhu. *Ocean Engineering Wave Mechanics* [M]. The first edition. Tianjin: Tianjin University press, 1998.

[8] Zhengxian Liu, Baojun Song, Chuangang Gu. Numerical Study of High Order Anisotropic $k-\varepsilon$ Models and Its Coefficients. Journal of Engineering Thermal Physics, 1996. 17 (Suppl): 31-34.

[9] CCCC-FHDI Engineering Co., Ltd. Code of Hydrology for Sea Harbour [M]. The first edition. Beijing: China Communications Press, April 2013.