해상풍력발전용 고정식 원형 하부구조물에 작용하는 파랑 및 조류 하중 해석을 위한 CFD 기법의 적용

박연석* · 진정수** · 김우전*

*목포대학교 해양시스템공학과

**중국절강해양대학교 선박해양공학과

CFD Application to Evaluation of Wave and Current Loads on Fixed Cylindrical Substructure for Ocean Wind Turbine

Yeon-Seok Park*, Zheng-Shou Chen** and Wu-Joan Kim*

*Dept. of Ocean Engineering, Mokpo National University, Mokpo, Korea

**School of Naval Architecture and Civil Engineering, Zhejiang Ocean University, Zhoushan, China

KEY WORDS: Computational fluid dynamics, Morison's formula, High-order boundary element method, Numerical wave tank, Wave load, Current load, Circular cylinder, Ocean wind turbine, Substructure

ABSTRACT: Numerical simulations were performed for the evaluation of wave and current loads on a fixed cylindrical substructure model for an ocean wind turbine using the ANSYS-CFX package. The numerical wave tank was actualized by specifying the velocity at the inlet and applying momentum loss as a wave damper at the end of the wave tank. The Volume-Of-Fluid (VOF) scheme was adopted to capture the air-water interface. An accuracy validation of the numerical wave tank with a truncated vertical circular cylinder was accomplished by comparing the CFD results with Morison's formula, experimental results, and potential flow solutions using the higher-order boundary element method (HOBEM). A parametric study was carried out by alternately varying the length and amplitude of the wave. As a meaningful engineering application, in the present study, three kinds of conditions were considered, i.e., cases with current, waves, and a combination of current and progressive waves, passing through a cylindrical substructure model. It was found that the CFD results showed reasonable agreement with the results of the HOBEM and Morison's formula when only progressive waves were considered. However, when a current was included, CFD gave a smaller load than Morison's formula.

1. Introduction

Wind turbine foundations installed at offshore sites are subject to wave and current directly. As a main support system of wind turbine assembly, cylindrical substructures have been widely used. Flow around circular cylinder has been a research topic in fluid mechanics for decades because of its complex physical phenomena, such as separation and vortex shedding, deformation of incoming wave.

It is common that the in-line force F_x on a circular cylinder in waves can be written as the sum of wave inertial forces and viscous drag, known as Morison's formula (Gunter et al., 1981). Potential flow models are often utilized to calculate wave load on offshore structures, e.g., high-order boundary element method (HOBEM) by Choi et al. (2001). Weggel et al. (1996) offers a wave load and response model

for a cylinder in deep water, using the second-order potential theory. A weakly nonlinear diffraction on a vertical truncated cylinder was investigated by Boo (1995) in a numerical wave tank, where the body boundary condition was exact, but the linear free surface condition was imposed. Recently, fully nonlinear interactions in regular and irregular waves studied by Boo (2002). There were some CFD applications (Dong and Huang, 2001; Park et al., 2001) to establish the numerical wave tank, for which the Navier-Stokes equations are solved other than potential-based Laplace equation.

First of all, in the present study, a numerical wave tank was actualized to generate progressive regular wave acting on a fixed cylindrical cylinder. The velocity components in both horizontal and vertical directions were given at the inlet of numerical wave tank, based upon Airy's linear wave theory. In addition, the so-called isotropic momentum loss

model supported in ANSYS-CFX package was adopted to damp out residual wave energy far downstream, preventing the wave reflection at the end of numerical wave tank. The interface capturing method using the VOF method (Hirt and Nichols, 1981) was used to simulate wave propagation in the numerical wave tank for each time-step. In the next, for the efficiency validation of numerical method, wave load on a truncated cylinder was calculated and compared with Morison's formula, experiments (Sung et al. 2007) and potential flow solutions using the higher-order boundary element method (HOBEM). CFD simulation shows good agreement with experiment and potential flow solutions, when the wave steepness is relatively small. In the last, CFD calculation of wave and current loads on a fixed circular cylinder were performed with both incoming wave and current being taken into account.

2. High-Order Boundary Element Method for the Potential Flow Solution

In the present study, the potential flow solution was sought for the comparison with the CFD results of wave force acting on a fixed circular cylinder. For the solution of potential flow model, the fluid is assumed to be inviscid, incompressible and the flow is irrotational. The governing equations becomes the Laplace equation for the velocity potential as given below.

$$\nabla^2 \Phi = \frac{\partial^2 \Phi}{\partial x^2} + \frac{\partial^2 \Phi}{\partial y^2} + \frac{\partial^2 \Phi}{\partial z^2} = 0 \quad (1)$$

The MLINHYDH (Multi-body Linear and Nonlinear HYdro-Dynamic analysis system using Higher-order boundary element method) program developed by MOERI was used (Choi et al., 2001). High-order boundary element method (HOBEM) is known to provide more accurate results than constant panel method (Liu et al., 1990). Hong et al. (2002) showed that the higher-order boundary element method can suppress numerical exaggeration encountered when conventional constant panel method is used. For details, see Choi et al. (2001).

3. Morison's Formula

The so-called Morison's formula was used to provide the basic information on forces exerted on a circular cylinder, since it is still most widely adopted in engineering. The drag force on a circular cylinder due to current is often expressed by

$$F_x = \frac{1}{2} \rho V_c^2 S C_d \quad (2)$$

where ρ , V_c , S and C_d represent the water density, velocity of the current, projection area of the cylinder and drag coefficient. When regular waves were imposed on a fixed cylinder, inertial forces composed of Froude-Krylov and scattering forces are usually the main part of wave load. Generalized Morison's formula that used relative velocity and relative acceleration of fluid particles is as follows

$$dF_x = C_m \rho \frac{\pi D^2}{4} dz \frac{\partial u}{\partial t} + C_d \frac{\rho}{2} D dz |u| u \quad (3)$$

where C_m , D , u and $\frac{\partial u}{\partial t}$ represent the inertial coefficient including add mass, diameter of the cylinder, instantaneous water particle velocity and local water particle acceleration at the center line of the cylinder.

When wave and current is acting on the structure, the Morison formula for a fixed structure can be written in terms of the resultant velocity including a steady current and an oscillatory wave velocity component.

$$dF_x = C_m \rho \frac{\pi D^2}{4} dz \frac{\partial u}{\partial t} + C_d \frac{\rho}{2} D dz |u \pm V_c| (u \pm V_c) \quad (4)$$

In the above formulas, the first term represents the inertial force and the latter represents viscous force. Force on the whole cylinder is calculated by integrating the Eq. (3) and Eq. (4) along the length of the cylinder.

Generally, the inertia coefficient (C_m) is taken to be 2.0 in a uniformly accelerated flow and the value of drag coefficient (C_d) is dependent on the Reynolds number and cylinder roughness. In the present study, the drag force coefficient (C_d) is chosen to be 1.0 as cylinder roughness is smooth and Reynolds number range of $10^3 < Re < 10^5$. For the accurate evaluation of wave load using Morison's formula equation, the integration was carried out from bottom to the wavy surface of the cylinder other than the mean free surface.

4. Generation of Numerical Wave Tank

4.1 Geometry and mesh topology

Numerical wave tank was actualized to estimate the forces of progressive regular waves on a fixed circular cylinder model. The horizontal and vertical length of 2D numerical wave tank is of 7 m and 3 m, respectively. Water depth is of 2 m. Geometry and meshing were done using ICEM-CFD. Fine mesh was created near the free surface for the more accurate movement of free surface. About 55,000 grids were used in 2-D tank. Geometry and grid system of the numerical wave tank are shown in Fig. 1.

Fig. 1 Geometry and grid system of the numerical wave tank

4.2 Definition of boundary conditions

In this section, boundary conditions of numerical wave tank were defined. A special computational domain at the end of numerical wave tank has been adopted to use isotropic momentum-loss model as a wave damping device. Bottom wall of the domain were given as wall boundary condition with free slip and top as opening boundary condition with static pressure. Velocity profiles from linear wave theory were given at the flow entrance. The direction of gravitational acceleration denotes (-)z axis and the special plane $z=0$ had been regarded as the still water level in the Cartesian coordinate system. Mathematical expression of a regular sinusoidal wave that proceeds towards positive x-direction is given below.

$$\zeta = \zeta_a \cos(kx - \omega t + \epsilon_0) \quad (5)$$

$$\zeta_a = \frac{H}{2} \quad (6)$$

The velocities of fluid particle of sinusoidal wave are expressed as follows.

$$u = \zeta_a \omega \frac{\cosh k(z+d)}{\sinh kd} \cos(kx - \omega t + \epsilon_0) \quad (7)$$

$$w = \zeta_a \omega \frac{\sinh k(z+d)}{\sinh kd} \sin(kx - \omega t + \epsilon_0) \quad (8)$$

where H , k , ω , d represent wave height, wave number, circular frequency, and water depth, respectively.

Velocity components at the inlet section are defined using CFX Expression Language (CEL) in ANSYS-CFX. Isotropic momentum-loss model was adopted to reduce reflection wave effect from the end wall of numerical wave tank. The numerical wave tank is composed of two separated domains. Separated domains and boundary conditions are shown in Fig. 2.

Fig. 2 Separated domains and boundary conditions of wave tank

4.3 Numerical calculation method

The commercial software ANSYS-CFX, which is based on Finite Volume Method (FVM), has been adopted for the numerical simulation. The set of equation solved by ANSYS-CFX are the unsteady Reynolds-averaged Navier-Stokes equations in their conservation form. Multiphase simulations for free surface deformation were captured using Volume of Flow (VOF) method. The governing equations for viscous flow include the continuity equation, three momentum equations, volume fraction and turbulence equations.

In addition, the $k-\omega$ Shear-Stress-Transport (SST) model was adopted, since it was designed to give highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients by the inclusion of transport effects into the formulation of the eddy-viscosity. For the inclusion of current effect later on, the $k-\omega$ SST model was expected to accurately simulate flow separation around a cylinder with relatively low computational cost compared to higher-order turbulence models.

In the present study, the first grid cell center was located at $y^+ = 50 \sim 150$ to utilized the wall function bridging the velocity profiles to the wall shear stress. In case of the first grid point positioned inside the log-zone, the automatic wall treatment method switching to the near-wall turbulence model was adopted in ANSYS-CFX.

4.4 Test parameters for numerical wave tank

The wave length and amplitude could be regarded as primary parameters to investigate the efficiency of the numerical wave tank. The calculation conditions of parameter study are shown in Table 1.

Table 1 Calculation conditions for parameter study

T (s)	0.6		0.8		1.0	1.2
A (m)	0.03	0.05	0.04	0.03	0.03	0.03
H/L	0.11	0.1	0.08	0.06	0.04	0.03

where T , $A(=\zeta_a)$, H , L and H/L represent the period, amplitude of wave, wave height, wave length and wave steepness, respectively.

Wave elevations were displayed in particular location. In order to observe the wave elevation in particular point by time history, waves are shown at location 2 m apart from inlet. The comparison of wave elevation was in Fig. 3 and Fig. 4.

Numerical simulation was performed to check the variations of amplitude at constant time period 0.8 s. The obtained wave profiles with the specific wave amplitude of 0.03 m, 0.04 m and 0.05 m respectively, were illustrated in

Fig. 3 Time history of wave elevation at location 2 m apart from inlet with different amplitude ($T = 0.8$ s)

Fig. 4 Time history of wave elevation at location 2 m apart from inlet with different periods ($A = 0.03$ m)

Fig. 3. It has been found that the calculated wave amplitudes are closed to the estimated values.

Variation of wave elevations with wave period is shown in Fig. 4. When the amplitude of wave at the tank inlet was

fixed to 0.03 m, numerical simulation was with the period of 0.6, 0.8, 1.0 and 1.2 s. In case of long wave period and small wave steepness, generated wave elevations seemed to better coincide with the amplitude given at the inlet. However, wave elevations would be underestimated when the wave period was 0.6 s. From the comparison between wave elevations of the cases with $T=0.6$ s and 1.2 s, one can find that observed wave elevation of the former case was distinctly lower than the latter. During simulation the distortion of wave profile was found near the inlet in the case of $T=0.6$ s, which probably indicated the emergence of breaking due to relatively short wave length. In addition, the authors had also found the wave breaking phenomena taking place widely when the wave period was of 0.4s. In this paper, the attention was paid upon the wave simulation with low wave steepness ($H/L < 0.1$) to ensure sinusoidal wave forms.

To simulate the wave propagation in the present numerical wave tank, at least 50 grid points were distributed along one wave length, while more than 20 grids were positioned across the wave height to ensure the sufficient resolution of velocity and pressure variation with propagating waves.

5. Efficiency Validation of Truncated Circular Cylinder in The Numerical Wave Tank

5.1 Description of test set-up for efficiency validation

Before applying the ANSYS-CFX package to solve the forces on a fixed circular cylinder, an efficiency validation of numerical wave tank with a truncated vertical circular cylinder had been accomplished by comparing the calculation with Morison's formula and the experimental data. The prototype's diameter and draft was of 16 m and 24 m. The experiment was performed at MOERI tank with the model scale of 1/50. The diameter and draft of the truncated vertical cylinder model were of 31.8 cm and 47.7 cm, respectively.

In this paper, four cases with the wave period ranging from 7 s to 10 s were selected for the efficiency validation among 24 cases of MOERI tests, as given in Table 2.

Table 2 Test cases for truncated cylinder in waves

Test case	07130	08130	09130	10130
T (sec)	1	1.13	1.27	1.41
H (m)	0.05	0.07	0.08	0.10

The computational domain of 3D numerical wave tank was defined as follows: horizontal length being of 12 m, vertical

length of 3 m and width of 3 m, respectively. Water depth is of 2 m. Structured grid system of 3,704,192 cells was created and refined grids were distributed around the cylinder and the free surface. The geometry and mesh topology are given Fig. 5.

Fig. 5 Geometry and mesh topology of truncated vertical circular cylinder

5.2 Results for the truncated cylinder cases

In this study, numerical calculations for a truncated cylinder with different wave height and fixed wave steepness (1/30) were carried out for efficient validation of numerical wave tank. Comparison of wave load is shown in Fig. 6.

Fig. 6 Comparison of wave loads for truncated cylinder cases

Here, F_x , ρ , g , k_0 , r and A are horizontal wave force acting in incoming wave direction, water density, gravity acceleration, wave number, radius of the cylinder and wave amplitude, respectively. Potential-flow solutions using the higher-order boundary element method(HOBEM) and wave loads by Morison's formula were also plotted. The present

numerical simulation gives reasonable values when k_0r is relatively small. However, the difference is noticed when wave length decreases. In the present study, numerical simulations were performed in the condition that k_0r was below 0.4 to ensure the validity of the present numerical wave tank.

6. Numerical Simulation of Loads on The Fixed Vertical Circular Cylinder

6.1 Problem description

Hitherto, it is still difficult to directly solve the engineering problem related to full-size prototype ocean structures. Size minimization should be done according to the present calculation ability. In the present study, numerical simulations were performed with 1/50 of a prototype cylindrical structure, whose diameter is of 3.5 m and draft is of 40 m, designed for the substructure of ocean wind turbine. Model cylinder's diameter was of 0.07 m and draft was of 0.8 m. Numerical simulation conditions are shown in Table 3.

Table 3 Numerical simulation conditions

	Prototype	Model
Scale ratio (Γ)	50	
Diameter (m)	3.5	0.07
Depth (m)	40.0	0.80

As a meaningful engineering application, three kinds of conditions have considered as follows, i.e. the cases with current, waves and the combination of current and progressive wave, respectively. First of all, in case of the effect of wave load on the cylinder taking into account, various combinations of time periods and amplitudes have been adopted. The parameters of calculation cases are shown in Table 4.

Table 4 Calculation conditions for numerical simulations

	Case01	Case02	Case03	Case04	Case05
T (s)	0.9	1.0	1.1	.	1.0
A (m)	0.05	0.05	0.05	.	0.05
H/L	0.08	0.06	0.05	.	0.06
V_c (m/s)	.	.	.	0.35	0.35

In case of evaluating the effect of dynamic loads imposed by current, current velocities of 0.35 m/s has been considered. Finally, in case of wave and current existing in the same direction, the encounter frequency should be

considered due to wave-current interaction. Encounter frequency is defined as follows

$$\omega_r = \omega_a - k V_c \quad (9)$$

where, k is wave number, V_c is current velocity, ω_a and ω_r mean the circular frequencies in absolute (space-fixed) coordinate and relative (moving) coordinate respectively. If the uniform current is taken into account, the Eq. (7)~Eq. (8) in absolute coordinate are changed as follows.

$$u = \zeta_a (\omega_a - k V_c) \frac{\cosh k(z+d)}{\sinh kd} \cos\left(\frac{\pi}{2} - \omega_a t\right) + V_c \quad (10)$$

$$w = \zeta_a (\omega_a - k V_c) \frac{\sinh k(z+d)}{\sinh kd} \sin\left(\frac{\pi}{2} - \omega_a t\right) \quad (11)$$

6.2 Result comparison of forces acting on the fixed circular cylinder

In this study, numerical simulations were performed for a circular cylinder model with wave, current, and the combination of current and progressing wave, respectively. Results of numerical simulation were compared with solutions of HOBEM and Morison's formula.

In case of waves acting on the fixed circular cylinder (Case01, Case02, Case03), wave loads with different wave frequency were calculated, as shown in Fig. 7. Here, $|F_x|_{\max}$, ρ , g , r and A are the peak value of horizontal wave force, water density, gravity acceleration, radius of the cylinder and incoming wave amplitude, respectively. Simulated results showed smaller values than those by Morison's formula and results of the HOBEM. This can be attributed to the underestimation of wave amplitude in numerical simulation.

The variation of force coefficient related to the case with only current being considered (Case04) was shown in Fig. 8. As time passed by, lift and drag forces increased, for the vortex shedding had developed with Strouhal frequency. Reynolds number of the present case was about 20,000. Drag coefficient approaches to 1.2. It is not easy to determine the drag coefficient especially when the vortex-shedding occurs. It is worth noting that the turbulence transition often affects the flow separation position which is a key parameter in determining actual drag forces and is strongly dependent upon Reynolds number.

Finally the case in which wave and current acted on the fixed circular cylinder at the same time (Case05) is shown in Fig. 9. The numerical simulation using ANSYS-CFX provided smaller values than Morison's formula, as expected from Fig. 7. Furthermore, when the current existed, the present numerical wave tank generated smaller wave amplitude than when only wave is simulated. A reasonable explanation to

this phenomenon is that wave deformation is sensitive to the current effect, when the incoming wave and current progress in the same direction. The wave length has been extended and wave amplitude declined. It is not clear yet how the numerical wave tank includes the current effect without declination of wave amplitude.

Fig. 7 Wave force by the wave frequency (without current)

Fig. 8 Time history of drag and by the current

Fig. 9 Comparison of the simulation and Morison's formula

Fig. 10 Comparison of the forces in Morison's formula

Fig. 11 Comparison of the forces in numerical simulation

Fig. 10 and Fig. 11 show that forces acting on a circular cylinder with wave, current, and the combination of wave and current in the same direction by Morison's formula and the CFD solution. The force coefficient for the case of the combination of wave and current seems to be a little bigger than the sum of force coefficients calculated for the case with current and wave separately. It is mainly due to the fact that the force value is proportional to the square of the local velocity and small increment of velocity due to progressive wave would result in the increase of resultant force.

Fig. 12 shows the variation of pressure on the cylinder at different phases when only wave is considered. When the wave passing by the circular cylinder, maximum pressure was appeared at the wave crest and trough. Because velocities of the fluid particles were gotten into maximum at the crest and trough.

Fig. 13 shows variation of wave profile when the combination of wave and current is considered. The free surface is elevated at the front face of cylinder due to stagnation pressure increment, while the opposite happens at the rear side of the cylinder. The vortex shedding at the lee side reduces pressure resulting in hollow free surface profile.

Fig. 12 Variation of the pressure coefficient by wave elevation

Fig. 13 The snapshot of incoming waves profile

7. Summary

In the present paper numerical simulations were performed for the evaluation of wave and current loads on a fixed cylinder model for ocean wind turbine substructure using ANSYS-CFX package. The Reynolds-averaged Navier-Stokes equations with $k-\omega$ SST turbulence model were solved. The wave and free surface profiles were captured using the VOF scheme. At first, the numerical wave tank was actualized by giving velocity at the inlet and momentum loss as wave damper at the end part of the wave tank. In the next, wave flow around a truncated cylinder was simulated and compared with the potential flow solution using HOBEM and the experiment carried out in MOERI tank for the efficiency validation. It was found that the present numerical method could give the reasonable estimation of wave load when wave steepness is relatively low. In the last, a fixed vertical circular cylinder as a support substructure of offshore wind

turbine assembly was considered. The simulated results were compared with Morison's formula and results of HOBEM. It is likely that there are some rooms for the improvement of the numerical wave tank especially when the current is overlapped. The present numerical wave tank seems to give the reasonable wave load in the case of wave exerting on a cylinder. However, when the current is overlapped with progressive wave, the numerical results predicted much smaller forces compared to Morison's formula.

Acknowledgements

This research work was supported by Basic Science Research Program through the National Research Foundation of Korea (Grant No. 2010-0002897) and Human Resource Development Center for Economic Region Leading Industry Project of Mokpo National University, funded by the Ministry of Education, Science and Technology, Korea.

References

- Boo, S.Y. (1995). "Weakly Nonlinear Diffraction Due to Vertical Cylinder in a 3-D Numerical Wave Tank", Proceedings of the Fifth International Offshore and Polar Engineering Conference, Hague, Netherlands, Vol 3, pp 19-25.
- Boo, S.Y. (2002). "Linear and Nonlinear Irregular Waves and Forces in a Numerical Wave Tank", Ocean Engineering, Vol 29, No 5, pp 475-493.
- Choi, Y.R., Hong, S.Y. and Choi, H.S. (2001). "An Analysis of Second-order Wave Forces on Floating Bodies by using a Higher-order Boundary Element Method", Ocean Eng, Vol 13, No 5, pp 117-138.
- Dong, C.M. and Huang, C.J. (2001). "2-Dimensional Wave Tank in Viscous Fluid", Proceedings of the 11th International Offshore and Polar Engineering Conference, Stavanger, Norway.
- Guenter, C., Ike, L. and Ostergaard. (1988). Offshore Structures Engineering, Springer-Verlag Berlin Heidelberg.
- Hirt, C.W. and Nichols, B.D. (1981). "Volume of Fluid (VOF) Method for The Dynamics of Free Boundaries", Journal of Computational physics, Vol 39, No 201.
- Hong, S.Y., Kim, H.J. and Choi, Y.R. (2002). "Experimental Study on Behavior of Tandem and Side-by-side Moored Vessels", Proceedings of 12th ISOPE, Kita-Kyushu 3, 841-847.
- Liu, Y.H., Kim, C.H. and Lu, X.S. (1990). "Comparison of Higher-order Boundary Element and Constant Panel Methods for Hydrodynamic Loadings", Journal of Offshore and Polar Engineering, ISOPE, Vol 1, No 1, pp 8-17.
- Park, J.C., Uno, Y., Matsuo, H. and Miyata, H. (2001). "Reproduction of Fully Nonlinear Multi-directional Waves by a 3-D Viscous Numerical Wave Tank", Proceedings of 11th International offshore and polar Engineering Conference, Stavanger, Norway.
- Sung, H.G., Kim, Y.S., Nam, B.W. and Hong, S.Y. (2007). "Experimental Investigation of Wave Loads on a Truncated Vertical Circular Cylinder", Proceedings of 2007 Fall Meeting of The Korean Society of Ocean Engineers.
- Weggel, D.C., Roesset, J.M. and Kim, M.H. (1996). "Second Order Vertical Diffraction Forces on Truncated Cylinders", Journal of Offshore Mechanics and Arctic Engineering 118, pp 259-266.

2011년 3월 8일 원고 접수

2011년 4월 7일 심사 완료

2011년 4월 22일 게재 확정