Wave Forces on Large Cylindrical Structure using CFD Simulation

Chanatha Netrasiri[1,a], Somchai Chucheepsakul [1,b] and Duangrudee Kositgittiwong[1,c]

[1] Department of Civil Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi,
126 Pracha Uthit Rd., Bang Mod, Thung Khru, Bangkok 10140,Thailand
[a]chanatha.net@gmail.com, [b]somchai.chu@kmutt.ac.th, [c]duangrudee.kos@kmutt.ac.th

Abstract

In this paper, a simulation using the commercial computational fluid dynamic software is performed to investigation of wave propagation and wave forces on a large cylindrical structure. The regular waves were generated according to the linear wave theory based on the three-dimensional unsteady Navier-Stokes equations. The free surface was described by using the method of Volume of Fluid (VOF). A wave absorbing method was employed by the porous media model at the downstream to prevent the wave reflection.

The simulation models demonstrated the results of inertia force caused by the wave on fixed cylinder. The total horizontal forces acting on the cylinder obtained by using CFD were compared with the results from the Morison equation and the diffraction theory, and they are found to be in close agreement. It also can be concluded that the CFD can be used to predict the wave forces on large cylindrical structures with the limitation of D/L less than 0.7.

Keywords*:* wave force, porous media, Morison equation, the diffraction theory

1. Introduction

The large cylindrical structures are commonly employed in many coastal and offshore engineering applications, to name a few, caisson concrete structures and spar offshore platforms. These structures are sustained under severe environmental loadings. Furthermore, the interaction between wave and structure is a complex phenomenon. Therefore, it is essential for design engineers to have good understanding on wave-structure interaction behavior. In the analysis of wave forces on large cylindrical structures, it requires an accurate mathematical model in order to have a good prediction of wave force.

Wave disturbing force is an important component of the stability of structure. There are many research studies on numerical simulation using commercial software to examine the hydrodynamic loading acting on the offshore structures, especially with the slender bodies. For example, Teixeira [1] discussed the acting of waves on a vertical pile with a square section using the commercial software, FLUENT [2011]. The represent problems associated with the interaction among wave and three-dimensional structures, and a user defined function (UDF) is introduced to maintain the elevation of wave along the wave tank. Repalle [2] described the boundary treatments device in FLUENT. It is shown that the out flow boundary condition should be set to allow the continuity through the boundary with a minimum of reflection. Iftelhar [3] had conducted the simulation model on slender body and compared the results between the wave theory and numerical mode.

The Morison's equation is widely used to approximate the wave force for the cylindrical slender bodies with D/L (diameter /wave length) ratio less than 0.2 in which the effect of diffraction is insignificant. However, when the ratio D/L larger than 0.2, the diffraction effect is significant and it has to be considered in the analysis. MacCamy and Fuchs [4].

As mentioned above, this research highlights the simulation of the wave force on large cylindrical structure based on the Computation Fluid Dynamic. The results from CFD were compared with Morison's equation and diffraction theory.

2. Wave Force from Morison's Equation

The force exerted on a fixed vertical stationary cylinder with small diameter or slender body can be calculated by using the Morison's equation. The force per unit length,F, is expressed as the inertia force and the drag force, and it can be written as

$$F=\frac{1}{2}\rho C_D D u/u/ + \rho C_M A\dot{u} \tag{1}$$

Where ρ is the density of fluid, C_D is the drag coefficient and u is the velocity far from the cylinder. The absolute sign in Eq.(1) is used in order to ensure that the drag force is always in the direction of velocity. This equation was first developed by Morison et al. [5]. The horizontal water particle velocity,u, and the horizontal water acceleration, according to linear wave theory, are determined by using Eqs.(2) and (3), respectively.

$$u = \frac{\pi H}{T}\frac{\cosh ky}{\sinh kh}\cos\theta \tag{2}$$

$$\dot{u} = \frac{2\pi^2 H}{T}\frac{\cosh ks}{\sinh kh}\sin\theta \tag{3}$$

Here y is distance between free surface level and bottom sea level, H is the wave height, T is wave period, k is the wave number, h is the water depth, and θ is $-\omega t$, ω is the angular frequency and t is the times at the maximum force

It is noticed here that the last term in Eq.(1) is called the inertia force and the coefficient C_M is called the inertia coefficient. For a circular cylinder, the value of C_M equal to 2 is normally used for the un-separated flow with the small Keulegan-Carpenter number. These values are rough estimates.

3. Wave Force from Diffraction Theory

When the diameter of cylinder becomes relatively large, the incident wave is disturbed by the cylinder, and it generated the reflected and diffracted waves. The reflected wave moves outward while the diffracted wave bent around the cylinder on the sheltered size of the cylinder.

The combination of reflected and diffracted waves is usually called scattered wave. This phenomenon causes the inconstancy of pressure around the cylinder. Hence, The effect of diffraction becomes important when the ratio D/L is larger than 0.2.

Basically, the flow around a circular cylindrical body in the diffraction flow regime is un-separated therefore the sinusoidal wave theory can be used to determine the wave force.

The total force on the cylinder can be calculated by [6]

$$F = \frac{\pi}{8}\rho g H D^2 C_M \tanh(kh)\cos(\omega t - \delta) \quad (4)$$

where g is acceleration of gravity and δ is the phase difference.

The results from the linear diffraction theory have been tested against the experiments by several researchers, in general they are in good agreement [7,8].

4. CFD simulations

The computational fluid dynamic (CFD) is the method that can be used to create the complex of wave phenomenon and helps to find the results of hydrodynamic such as hydrodynamic force or characteristic of wave. In the simulation model the finite volume method (FVM) [9,10] is used to solve the governing equations.

The CFD calculations shown here are based on the simulated solution to the Reynolds Averaged Navier-Stokes equations for incompressible fluid in a three-dimensional geometry. The prediction of the free surface is based on the volume tracking method (VOF, volume of fluid) [11]. This method was developed to simulate highly nonlinear effects such as breaking wave at interface. In this study, two phases of fluids, water and air, were used.

VOF model is used to simulate the free surface between water and air. The method is based on the idea of so called fraction function $\propto_g$. It is defined as the integral of fluid's characteristic function in the control volume. Basically, when the cell is empty, $\propto_g = 0$; if the cell full, $\propto_g = 1$; If $0 < \propto_g < 1$, then the volume is the interface between the two phases.

CFD simulations including all geometric details of the model test would be costly and so the model has to be simplified depending on theory of diffraction. The present domain is depicted in Fig.1. A numerical wave tank has been modelled with the following dimensions: 10L m length, 2.5L m width, 3H m the water depth layer and the domain depth is 6H m.

A circular cylindrical size were varied by the ratio of dimension and wave length according by the diffraction theory, the cylindrical structure seated on the bottom of the plane at 20 m from the inlet areas to the centre of the cylinder and at a distance of 1.25L from the side walls as shown in Fig 1.

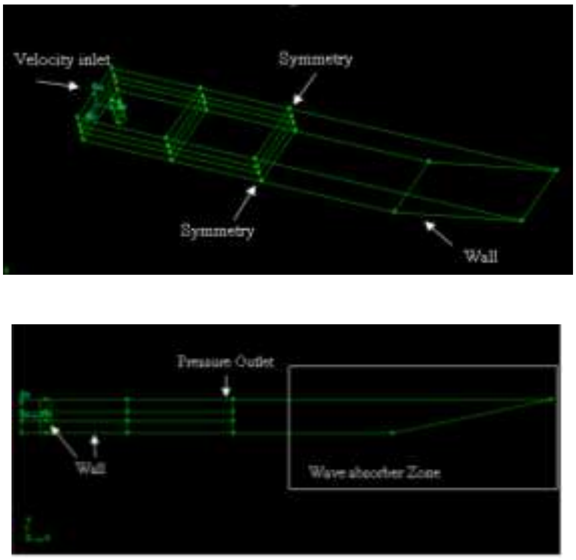

Fig.1 Wave tank domain

Waves are generated at the left boundary (inlet) and propagated to the right based on the linear theory. The fluid employed in this simulation is seawater, with specific mass of 1025 kilograms per cubic meters. A wave period, T, was 6 seconds and wave height, H, is 6 meter and water depth is 18 meters.

The inlet boundary conditions employed for investigation of water (Fig.1).The outflow is modeled as the sloped wall boundaries to create the breaker wave and to decrease reflection at downstream. For the lateral surfaces of the wave tank, as well as the cylinder surface, the symmetry condition is employed.

A user defined function (UDF) is used to reduce the effect of reflection at the downstream areas or porous zone in order to maintain the wave elevation profile corresponding to the first order free surface elevation (Eq.2). It is given by

$$\eta = \frac{H}{2}cos(kx - \omega t) \quad (5)$$

where x is the distance along the direction of wave propagation.

Momentum equations for porous media are modeled by the addition of a momentum source term to the standard fluid flow equations. The source term is composed of two parts: a viscous loss term and an inertial loss term. The source term for momentum tank is $\boldsymbol{S_i}$ which contributes the pressure gradient in the porous zone, creating a pressure drop that is proportional to the fluid velocity (or velocity squared) in the cell. It is written as

$$\mathbf{S_i} = \left(\frac{\mu}{\alpha}v_i + C\frac{1}{2}\rho v_{mag} v_i\right) \quad (6)$$

where α is the permeability, v_{mag} is a velocities of magnitude and C is the inertial resistance factor.

5. Results

5.1. Verification of test tank

Fig. 2 shows a comparison between the water elevation at x = 20 m along 60 sec which is about 10 wave periods. The solid line represents the linear wave theoretical results, and the dotted line represents the numerical results. Although there is a slight shift in the phase at the through of wave, but at the crest the numerical results matched very well with the theoretical results. This verification thus indicates that the test tank and underling mesh scheme is suitable for the cylinder simulation.

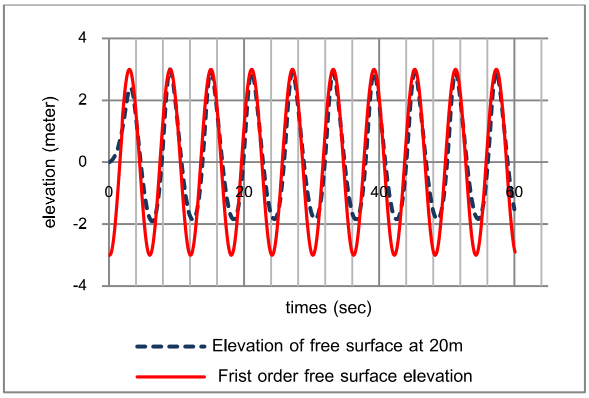

Fig. 2 surface elevation history
At point x=20 m

Figure 3 shows the water free surface at 60 sec, the water surface is sinusoidal which matched with the theoretical analysis

Fig. 3 Wave surface at t = 60 s

For the wave absorbing model, the waves are absorbed completely by the porous media in the right part. Otherwise, the waves will reflect since the right boundary condition is wall

5.2 Results and Discussions

In the numerical simulations, the total horizontal force was calculated by the integration of pressure in the areas which is parallel to the direction of wave propagation. Figure 4 presents the total force in the wave propagation direction obtained by FLUENT [12,13] compared with the total force calculated by Morison's equation and the total forces from diffraction theory for various values of D/L.

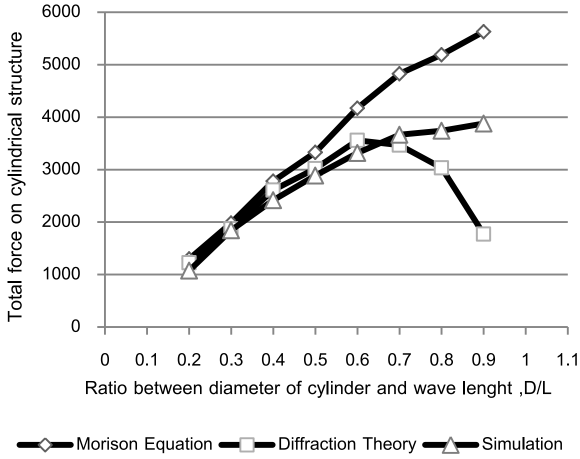

Fig. 4 numerical simulation results of wave force compared to the Morison's equation and the diffraction theory

According to the diffraction theory, the effect of diffraction will be significant, if the ratio of D/L is larger than 0.2. It was found that the results obtained by CFD are in a good agreement with those from the diffraction theory for the range of $0.2 < D/L < 0.7$. In comparison with the Morison equation, the simulation results yielded close agreement when the ratio of D/L is less than 0.4.

Since the cylindrical diameter is small, incident waves around the cylindrical structure do not provide much scattered. But when the diameter of cylinder exceeds 0.2 of the wave length, then the flow will be separated and bent around the surface of cylindrical structure. Thus the resultant drag force (viscous effects) has a small value. This demonstrates that the effect of scattered wave has controlled the force on the large cylindrical structure.

And, it can be concluded that the CFD can be used to predict the wave forces on large cylindrical structures with the limitation of D/L less than 0.7

6. Conclusions

From the simulation model of wave forces on large cylindrical structures in this study, it can concluded that for the range of D/L ratio of 0.2-0.4 the response from the Morison equation yielded a very good prediction of the wave force on the slender structure. For the ratio is greater 0.4, the Morison equation is unable to predict the wave force. From the results of the diffraction theory, it provides the reliable and accurate results for wave force on large cylinders for the ratio of D/L up to 0.7, over this ratio the results are inaccurate.

7. References

[1] Teixeira, Gomes, Santos, Isoldi and Rocha (2010). Numerical Analysis of A Regular Wave over A Vertical Pile with A Square Section, paper presented in *Brazilian congress of thermal Sciences and Engineering 2010*, Uberlandia, MG, Brazil

[2] Repalle, Thiagarajan and Thomas (2007). CFD Simulation of Wave Run-up on A Spar Cylinder, paper presented in *Australasian Fluid Mechanics conference 2007*, Crow Plaza, Gold Coast, Australia

[3] Iftekhar and Jose (2004). Slender-body Approximations of Hydrodynamic Forces For Spar Platfrorms, *Applied Offshore and Polar Engineer*, vol.14(2), June 2004,ISSN 1053-5381.

[4] MacCamy, R.C. and Fuchs, R.A. (1954): wave force on piles: A diffraction theory. U.S. Army Corps of Engineers, *Beach Erosion Board*, Tech. Memo No.69, 17p

[5] Morison, J.R., O'Brien, M.P., Johnson, J.W. and Schaaf, S.A., 1950. The forces exerted by surface waves on piles. Petr. Trans., 189, 149-156.

[6] Mogridge, G. R.and Jamieson, W. W. (1975) wave force on a Circular Caisson: Theory and Experiment. *Can. J. Civil Eng.*, Vol.2, pp.540-548

[7] Chakrabarti, S.K. (1987) Hydrodynamics of Offshore Structure. *Computational Mechnics Plublication*

[8] B.Multlu and Jorgen (1999). *Hydrodynamics Around Cylindrical Structures*, Revise edition, volume 26, ISBN:981-270-039-0, Fulsland offset printing , Singapore.

[9] LeVeque, Randall (2002) Finite Vulome Methods for Hyperbolic Problems, Cambridge University Press.

[10] Toro, E. F. (1999), Riemann Solvers and Numerical Method for Fluid Dynamics, Spronger-Verlag

[11]Hirt, C. W. ,nichols, B. D. (1981) Volime of fluid (VOF) method for the dynamics of free boundaries, *Journal of computational Physics39*, 201-225p.

[12] ANSYS FLUENT 12.0[online] 2009[cited 2011 feb 10]. Avialable from URL: https://www.sharcnet.ca/software/Fluent12/html/ug/node1.htm.

[13] ANSYS,Inc. ANSYS FLUENT 12.0 Manual pennsylvania, USA: ANSYS, Inc.,2009