Journal of Taiwan Energy
Volume 4, No. 3, September 2017, pp. 249-266

Scour Behavior Analysis for the Mono-Pile Substructure of the Offshore Wind Turbine

Ming-Hong Chen[1*] Chun-Wei Lin[2] Tso-Ren Wu[3] Chin-Cheng Huang[4]

ABSTRACT

In the present study, the development of the scour around the support structure for an offshore wind turbine (OWT) has been investigated. The behavior of sand on the seabed is calculated by the developed Bi-Viscous Rheological Model (BVRM). The comparison of the developed model with the experimental study has been conducted. Results by the numerical model consistent well with compared case. The combined hydrodynamic load, including current and regular wave, are incorporated into the developed numerical model with the validated BVRM and the variation of scour distribution is presented and discussed. The maximum scour depth/diameter of mono-pile (S/D) by the observed data is compared with the estimation by the proposed numerical model. The empirical formula in literature is also included into the comparison. Empirical formulation from the surveyed studies leads to the larger S/D value (1.24 ~ 1.5) comparing to the measured data among the surveyed studies (0.125 ~ 1.66). Although this paper predicts the smaller value of S/D (0.33), it is still within the range of the measured data of surveyed studies. The developed numerical model is applicable to predict the variation of scour distribution and the maximum scour depth around the support structure of the OWT can be evaluated.

Keywords: offshore, wind turbine, scour, numerical analysis

1. Introduction

Wind is an important renewable energy source. This is especially true for countries without natural resources (such as Taiwan). A challenging goal has been made by Taiwan government that there will be 20% of energy coming from the renewable energy. The possible and practical choices are solar and wind energy. Therefore, it is paramount to develop the technique for the wind energy system. Comparing to the onshore wind farm, the offshore wind farm has higher quality of wind power source, including higher average wind speed and lower turbulence. However, harsh environmental impacts, such as typhoon, wave, current, and earthquake, must be taken into the consideration for the development of a reliable OWT system. In order to provide the safe operation and reliability for the construction of the OWT, site specific conditions should be employed to the analysis and evaluation for the verification of the designed wind turbine and its support structure.

[1] Research Assistant, Mechanical and System Engineering Program, Institute of Nuclear Energy Research
[2] Ph. D. Candidate, Graduate Institute of Hydrological and Oceanic Sciences, National Central University
[3] Professor, Graduate Institute of Hydrological and Oceanic Sciences, NCU
[4] Researcher, Mechanical and System Engineering Program, INER
* Corresponding Author, Phone: +886-3-4711400#3351, E-mail: minghongchen@iner.gov.tw

Received Date: May 26, 2017
Revised Date: July 6, 2017
Accepted Date: July 26, 2017

There are 2,488 wind turbines installed in 74 wind farms among 11 countries across Europe. Total installed capacity is 8,045 MW at the end of 2014, and it is enough to supply 1% of total electricity consumption in Europe (EWEA, 2015). Unlike the onshore wind turbine, the offshore turbine is installed in the ocean. Therefore, a reliable substructure that supports the turbine system and withstands the impact of current and wave is crucial for the development of OWT. There are two kinds of substructure design, i.e. fixed and floating. Comparing to the developing floating design, the mono-pile of fixed design is the most commonly used substructure of the installed OWT system. In Europe, 78.8% of the installed OWT used the mono-pile substructure (EWEA, 2015).

The availability of the design data about the OWT is limited due to the confidentiality. A more effective and efficient method to develop the verification technique would be the international corporation project, i.e., the Offshore Code Comparison Collaboration (OC3), the Offshore Code Comparison Collaboration Continuation (OC4), and the Offshore Code Comparison Collaboration Continuation, with Correlation project (OC5) by International Energy Agency (IEA) (Robertson *et al*., 2015). These international corporation projects, leading by the National Renewable Energy Laboratory (NREL), conducted the integrated structure load calculation and results were compared by participated countries and units. Complicated and dynamic loads on the OWT were considered and structure analyses are compared by programs via different sources to verity the reliability and correctness of the verification process.

The development of the scour on the structure of OWT is inevitable, and it also affects the structure integrity. Therefore, the protection to reduce the extent of scour is crucial to ensure the structure stability in the marine environment. In order to propose the effective scour protection mechanism, the scour developed near the solid structure induced by wave and current should be predicted using validated model. Until nowadays, however, the scour prediction formula is still in large inaccuracy under the combined wave and current for the OWT (Matutano *et al*., 2013). The effectiveness of the scour protection mechanism can only be verified with a reliable description of scour and sediment around the support structure under the combined marine loads.

In the present study, the numerical model has been developed based on the reference 5 MW OWT from the report of NREL (Jonkman *et al*., 2009). The subject of the present study is on the evaluation of the hydrodynamic effect on the development of scour. Specifically, the effects of the site-specific current and wave on the support structure of the OWT are investigated. Thereafter, the newly developed scour prediction model is proposed and verified with experimental results from reference studies. With the validated scour predicting model, the dynamic development of the scour around the support structure of the 5 MW OWT is calculated and analyzed. Results were also compared with literature data.

In recent studies, simplified assumptions were employed in the simulation of the oceanic flow. For instance, it was commonly assumed that the flow is inviscid and irrotational in solving the potential equation. The semi-empirical Morison's formula was employed to describe the wave load. Those assumptions make the proposed model unable to evaluate the unsteady and highly turbulent flow (Sebastian & Lackner, 2012). Zanke *et al*. developed the equilibrium model to evaluated the scour depth in non-cohesive sediments under current and short waves (Zanke *et al*., 2011). The proposed empirical formula showed better results than previous calculation in the evaluation of the equilibrium scour depth. The dynamic variation of the flow and scour, however, can't be investigated by the simplified empirical formula. Prendergast *et al*. investigated the effect of scour depth on the natural frequency of the support structure of the OWT (Prendergast *et al*., 2015) by using an extreme value of scour depth rather than to evaluate based on the realistic flow conditions. Sørensen and Ibsen (Peder Hyldal Sørensen & Bo Ibsen, 2013) employed the empirical formula to evaluate the scour depth for the mono-pile OWT. Huang *et al*. (Huang *et al*., 2009) provided the computational fluid dynamics (CFD) model for the calculation of the scour around the bridge piers. Using the commercial code-FLUENT, flow patterns around the pile with good quality were obtained. However, they also point out that further detailed and rigorous investigation should be conducted for validation. Nielson *et al*. proposed some protection mechanisms against the scour (Nielsen *et al*., 2013) in a current. The variation of the velocity distribution with/without the protection mechanism was investigated. Nevertheless, the scour was not directly predicted and the effect of wave was not considered.

The wave model should be able to predict the effect of the nonlinear and breaking wave to simulate the interaction among large waves, strong current, structure and scour. Wienke and Oumeraci (Wienke & Oumeraci, 2005) coupled the breaking wave theory and the parameter model to evaluate the wave force on the wind turbine due to the plunging breaking waves. Yang and Shen solved the equation of the potential flow to calculate the behavior of the wave (Yang & Shen, 2011). Calderer *et al*. included the effect of wave by the simplified linear theory (Calderer *et al*., 2014). Marino *et al*. (Marino *et al*., 2011; Marino, 2010) coupled the aero-elastic model with the non-linear wave theory to solve the dynamic force on the structure. It was found by the studies of Marino and Lugni (Marino *et al*., 2011; Marino, 2010; Lugni *et al*., 2006) that the wave force and local pressure increase in a short period of time when wave breaking near the support structure of the wind turbine. Marino *et al*. (Marino *et al*., 2013) employed the nonlinear wave theory and statistical method to analyze the force on the wind turbine for a long time. Luengo *et al*. (Martinez-luengo *et al*., 2017) investigated the effect of scour on the service life of the mono-pile supported OWT. Results showed that the service life of wind turbine decreased from 33.1 year of no scour to 11.4 year with 3 m scour. This indicates that the development of scour around the mono-pile damages the reliability of the wind turbine. Baykal *et al*. (Baykal *et al*., 2017) developed the numerical model for the simulation of scour/backfilling around a pile with k-ω model for turbulence flow. Results were compared with experimental data in literature.

Since the traditional wave theories, whether linear or non-linear, are unable to appropriately describe the combined effect of typhoon induced wave and current, the present study using the internal wave maker module to produce the target wave. The behavior of the surface breaking wave is described by the volume of fluid (VOF) method. For the turbulence effect, the large-eddy simulation (LES) method is employed to describe its behavior. By the LES model, the small scale turbulence, which is unable to be resolved by the computational grid, can be predicted with reasonable accuracy. The LES equation is solved by the projection method. The model of the wind turbine structure is developed by the partial-cell treatment. In our previous study, the numerical model for the jacket-type support structure was proposed with the modified internal wave maker module. The effects of regular, irregular wave and current on the force of the support structure were compared with different azimuthal position of the support structure (Chen *et al*., 2016).

For the simulation of scour, the Discontinuous BVRM is proposed in the present study. For the evaluation, the maximum scour depth is generally compared with the diameter of the mono-pile (D) as a characteristic parameter to evaluate the severity of scour. From the available observed data, the maximum scour depth ranged from 0.125D to 1.66D (Matutano *et al*., 2013; Gerdes *et al*., 2008; Whitehouse *et al*., 2008; Carroll *et al*., 2010; Whitehouse *et al*., 2011). The large deviation on the maximum scour depth is mainly due to the different properties on the sand and seabed among the observed locations. The maximum scour depth was also estimated by the standard or the empirical formula. The offshore standard Det Norske Veritas suggests the value of 1.3D (Veritas, 2011), while it was between 1.24D to 1.5D from the empirical formulation by Breusers (Breusers *et al*., 1977).

It was shown from the above review that the scour developed around the support structure of the OWT will affect the natural frequency and stability of the structure. Numerical model has been proposed for the evaluation of the velocity distribution without considering the sand and scour (Nielsen *et al*., 2013). For the study evaluated the scour variation using commercial code needed further validation (Huang *et al*., 2009). Therefore, the numerical model will be proposed and validated with literature data in the present study. Furthermore, the maximum scour depth will be evaluated and compared with those from observation and empirical formula.

2. Numerical Model

The computational fluid dynamics (CFD) model is capable to predict the interaction between the wind turbine and various kinds of load. The complicated effects of turbulence, wave, current, structure, and scour variation should be considered comprehensively in the analysis for the development of the OWT. Solving the Navier-Stokes equation with the method of the fluid-structure interaction (FSI) is an effective way to evaluate the hydrodynamic load on the OWT. Another critical issue for the simulation is that the investigated environment is a multi-scale problem. The scale of the oceanic effect is usually measured by kilometer, while it is meter for the OWT. This scale difference results in the difficulty for the simulation. In the present study, the numerical model is developed by the open source code-Splash3D, and the LES model is incorporated into the simulation to solve the Navier-Stokes equation. The Splash3D model was modified from the Truchas developed by the Los Alamos National Laboratory (Liu *et al*., 2005; Wu, 2004). With the access of the source code, one can modify or add necessary function. The developed model is capable to solve the three-dimensional, incompressible Navier-Stokes (N-S) equation. It also can track the position of the fluid surface by the VOF model. Parallel computation is available by using the Message Passing Interface with Chameleon portability system to increase the grid resolution and reduce the required time. The developed code is able to predict the breaking wave and the turbulence around the solid structure. The Discontinuous BVRM is proposed for the simulation of scour around the support structure of the OWT. Governing equations and numerical models are described in the follow sections.

2.1 Governing equations

The N-S equation is solved by the LES-VOF model. Specifically, the turbulent flow is described by the LES model, while the VOF model is employed to preciously predict the distribution of the fluid flow and the position of the free surface. With the assumption of the incompressible flow, the continuity equation (mass conservation) can be described as:

$$\nabla \bullet u = 0 \tag{1}$$

Momentum equation (N-S) is:

$$\frac{\partial(\vec{u})}{\partial t} + \nabla \cdot (\rho \vec{u}) = -\frac{1}{\rho}\nabla P + \frac{1}{\rho}\nabla \cdot \tilde{\tau} + \vec{g} + \vec{F}_0 \tag{2}$$

$$\tilde{\tau} = \mu\left(\nabla \vec{u} + \nabla^T \vec{u}\right) \tag{3}$$

where u is the velocity vector, ρ is the effective density, g is the gravity, F_0 is the external force on solid, t is time, P is pressure, and μ is the coefficient of dynamic viscosity. The first term on the RHS in Eq. (2) represents the effect of pressure, and the second term is the effect of viscosity. The vortex and boundary layer as flow passing through the structure are calculated by solving the N-S equation. The conservation equations are discretized by the Finite Volume (FV) method. The volumetric integration of each discretized computational cell would be transformed into surface integration by the Gaussian divergence theorem as:

$$\frac{\phi_i^{n+1} - \phi_i^n}{\Delta t} + \frac{1}{V_i}\sum\nolimits_f \left[A\vec{u}^n\right]_f [\phi]_f^n = S_i^n \tag{4}$$

2.2 Volume of Fraction

Equation (1) and (2) describes the phenomenon of a single-phase flow. In the simulation of the wave and structure, two kinds of fluid (water and air) should be considered simultaneously. Furthermore, the third fluid (sand) will also be incorporated in the scour prediction. It is complicated to solve the N-S equation for different kinds of fluid with different fluid properties. The VOF (Hirt & Nichols, 1981) method is considered to solve such complicated problem. With modification by Kothe *et al.*, (Kothe *et al.*, 1996) this method is capable to solve the complicated free surface level for multi-phase flow with single set of N-S equation. With the VOF method, physical properties within the computational cells are evaluated based on its proportion as an averaged fluid.

2.3 Large Eddy simulation

For a simulation incorporated the effect of the breaking wave, the turbulent model should be considered. There are three kinds of method available, i.e., Direct Numerical Simulation (DNS), Reynolds-Average Navier-Stokes Equation (RANS), and LES. For the method of the DNS, results with the highest precision could be obtained since there is no assumption considered within this method. However, extremely high precision of numerical model, time step size and grid resolution are necessary. These requirements lead to long calculation time and huge computational resource. In general, the DNS method would be practical for the condition with low Reynold Number and small scale of turbulence (Kim *et al.*, 1987). For the method of the RANS, the concept of time-average is incorporated. The turbulence effect is described as the Reynold Stress in the momentum equation. For instance, the k-ε turbulence model using the additional term of the turbulence kinetic energy (k) and the turbulence dissipation rate (ε) to solve the Reynold stress and describe the turbulent flow (Lemos, 1992).

Besides the DNS and RANS method, the LES method is also capable to describe the turbulent flow (Deardorff, 1970). Without evaluating the motion of all vortexes, the LES method employed the Spatial Filtered method to classify the physical properties into the grid-scale and subgrid scale catalog. Effects of the geometry and the flow condition on grid-scale properties are significant, while these are minor on the subgrid scale properties. Thus, vortexes of grid-scale are solved by the N-S equation, while those of the subgrid scale are evaluated by the turbulence closure model. If the size of a grid is smaller than the minimum vortex, results by the LES model would be consistent with the DNS method. If not (due to limited computational resources), vortexes with different scale are described by different algorithm. As a result, the trade-off is reached on the precision and practicality by the LES method. In the present study, the LES method is employed to describe the complicated turbulent flow as the wave and current go through the support structure of the OWT.

2.4 Discontinuous Bi-Viscous Rheological Model

Fluid can be classified into the Newtonian and the Non-Newtonian fluid based on its viscosity. For the Newtonian fluid, such as air and water, the relationship between the strain rate and the applied shear stress is linear. On the other hand, the non-linear viscosity and the critical shear stress are the properties for the Non-Newtonian fluid, such as sediment and lava. There would be a critical shear stress known as the Bingham yield stress. Below the critical value, fluid acts like solid due to the strong bonding force between molecules. When the applied shear force is larger than the critical value, yield and the Newtonian-fluid behavior would be observed. Such behavior can be described by many proposed Rheological models. One of the well-known models is the Bingham model (Assier-Rzadkiewicz *et al*., 1997). As described by the Bingham model, there is no deformation observed for a material within the region of subcritical shear force. Otherwise, the fluid will deform and move accordingly. The behavior of such material can be described by two parameters: Bingham yield stress (τ_0) and Plastic dynamic viscosity (μ_B) (Liu &Mei, 1989) as illustrated in Figure 1. The employed parameters are listed in Table 1.

2.5 Wave model and input data

In the present simulation, the considered wave is generated from the observed data by the Central Weather Bureau of Taiwan from 2013 to 2014 at the investigated region (Hsinchu). Sample was taken every 10 minutes, and data was collected every hour. The measured range is from 0.03 to 0.4 Hz with 41 frequencies and resolution of 0.01 Hz. In order to select the representative event, the case with relatively large significant wave height and single peak spectrum would be preferred. Selecting the event with higher significant wave height is based on the engineering consideration. The Joint North Sea Wave Project is preferred since it is one crest spectrum. Furthermore, events with two or more crests spectrum usually are not representative and can only be observed in special conditions. Based on these two selection consideration, the event observed at July-13, 2013 would be employed as the input data to generate the required wave for the simulation. After the rearrangement and statistical treatment, the wave parameters for the 3D simulation can be obtained. The significant wave height is 5.78 m and its period is 9.8 seconds. Based on the dispersion relationship, the modified internal wave maker module is implemented to generate the required wave as a site-specific load for the OWT. Besides the generated wave, the

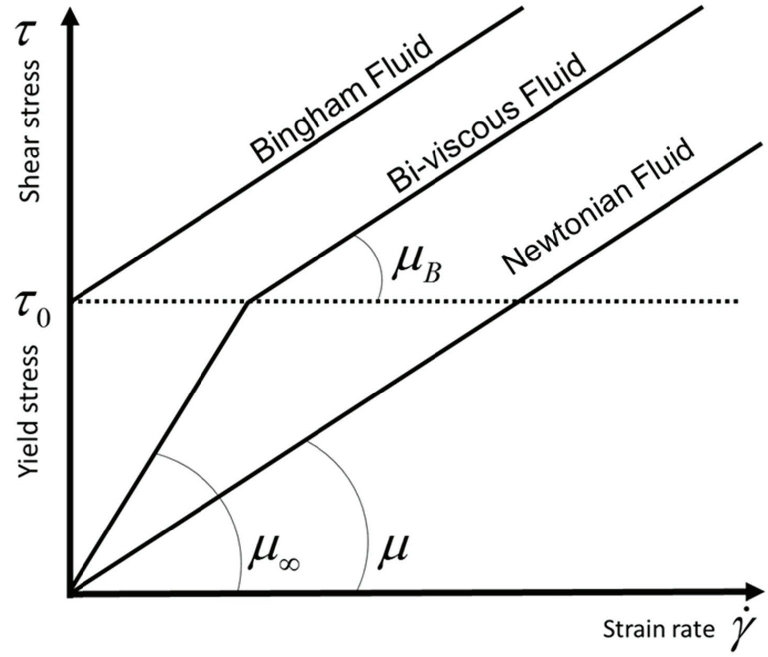


Fig. 1. Schematic diagram of the relation between the shear stress and strain rate for different types of fluid (Liu & Mei, 1989).

Table 1. Parameters for the case with constant speed current (by authors)

Bottom Friction Effect	Not considered
Flow Velocity (m/s)	1.41
Finest Grid Resolution (m)	1.0
Residual of Convergence Condition	1.0e-6
Mean Water Depth (m)	25.0
Courant Number	0.85

effect of the current is also considered as a coupled load. Numerical simulation will be conducted with this coupled load for a referenced 5 MW OWT. With the proposed BVRM, the development of the scour on the seabed of the support structure will be investigated.

The development of scour around the mono-pile substructure (with diameter of 6 m) is investigated by the validated BVRM. The regular wave with constant speed current is considered as the hydrodynamic load. The employed parameters for the investigated case are listed in Table 1. The current speed of 50 years return period of typhoon (1.41 m/s) is selected in the calculation.

Parameters for the BVRM are adopted from the experimental study mentioned in the previous section as listed in Table 2.

The Dirichlet boundary condition is employed in the calculation to generate the desired current at the upstream region. At the downstream region, the sponge layer is considered to reduce the momentum of flow, avoiding the undesired numerical reflection from the outlet boundary. For the lateral sides, the free-slip boundary is considered. The hydrostatic outflow is implemented

Table 2. Sediment Rheology Parameters Assigned in the case with constant speed current (by authors)

Yield Stress τ_0 (Pa)	Plastic Viscosity μ_b (Pa-s)	Penalty parameter μ_∞ (Pa-s)	Density (kg/m^3)	Thickness (m)
1600.0	10.0	1.0 × E + 11	1500.0	40.0

at the outlet boundary. The computational domain is shown in Figure 2. The flow direction aligns with the x-coordinate, and the considered length is 800 m. The range of width (y-coordinate) is from -100 m to 100 m, and the range of height (z-coordinate) is from -40 m to 60 m. The size of the computational domain is determined by the sensitivity study and the employed wave condition. The Internal Wave Maker Module (IWMM) is installed at inlet boundary of the upstream region. The suitable distance to make the desired wave would be 1/20 to 1/5 of the wave length. The region of the sponge layer should be larger than 1.5 times of the wave length. Thus, a distance of 300 m as the wave chopping region is considered in the present simulation. Water depth is 25.0 m, and the thickness of the sand is 40.0 m in the present study. The initial interface between sand and water is at the position of z = 0.0 m. Numerical mesh for the structure is built by the method of the partial-cell treatment (PCT). By examining the case without the structure, the desired wave is obtained appropriately by the IWMM at the position of x = 350 m from the inlet region. Therefore, the substructure is installed at the location of x = 350 m from the inlet region as shown in Figure 2.

The orthogonal grid system is employed to develop the computational cells with the refinement near the region of the structure. The grid size around the structure is 1.0 m. The refinement is made around the structure with ratio of 1.1849 in the x-direction, and the ratio is 1.14833 in the y-direction. In the z-direction, the uniform grid size is employed. The employed grid size is selected based on the verification of the BVRM with experimental data as shown in the previous section. The grid resolution in the verified case (half-cylinder experiment) is applied to the present case of 5 MW OWT. The time step size is selected between 1.0 to 0.001 sec. per step by the program

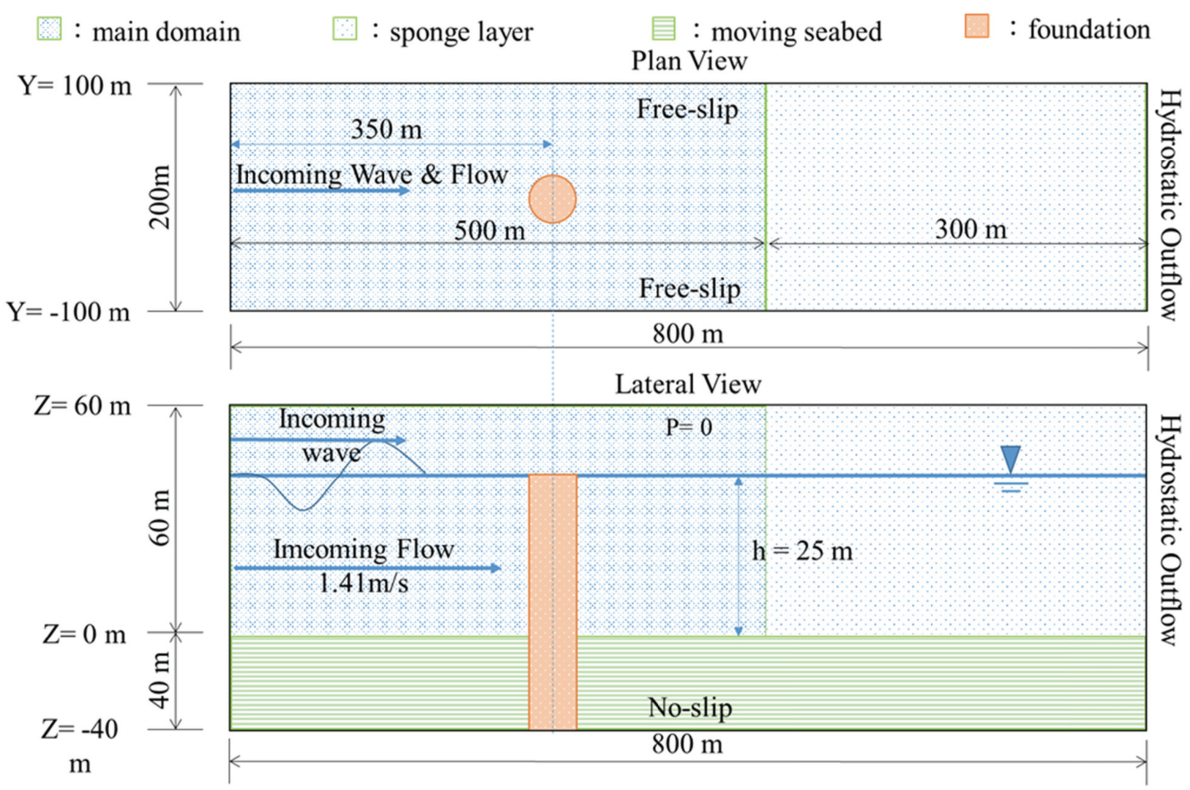


Fig. 2. Schematic diagram for the computational domain (by authors).

based on the stability criterion for the transient calculation. As the generated wave reaches the solid structure, smaller time step size would be employed to capture the behavior of the fluid-solid interaction with precision and stability. The numerical model without considering the effect of scour has been presented in our previously study (Chen *et al.*, 2016), and the domain size, grid system and the generated wave were verified. The finalized grid number is 1,036,800 after the independency test.

3. Results and discussion

3.1 Comparison with experimental investigation

The proposed BVRM on the prediction of the scour hole is verified by applying to the experimental study by Dey and Barbhuiya (Dey & Barbhuiya, 2005). The geometry of the experiment is illustrated in Figure 3. A half cylinder was installed at the center of the channel wall. The radius of the cylinder is 0.13 m. The thickness of the sand deployed within the channel is 0.3 m with the mean diameter (D_{50}) as 0.52 mm. The initial water depth is 0.2 m, and the inflow velocity is 2.094 m/s. The flow direction is indicated by the red arrow within Figure 3.

Numerical model is developed accordingly as shown in Figure 4. Geometries are the same with experimental study. The uniform computational grid is employed with the grid resolution of 0.02 m.

Three parameters should be provided to describe the behavior of the sand on the seabed using the BVRM: yield stress, coefficient of viscosity before and after yield criterion. The yield stress was measured by Raikar and Dey (Raikar & Dey, 2008). It is 0.408 Pa for the sand with D_{50} = 0.81 mm. The coefficient of viscosity after the yield criterion (μ_b) is adjusted in the present study since it's rarely mentioned in literature and difficult to measure by instrument. The finalized value of μ_b is 0.0005 Pa-s. For the coefficient of viscosity before the yield criterion, however, an extremely large value would be appropriate to describe the solid-like behavior. In the present study, it is set as 1.0E+11 Pa-s. The density of the

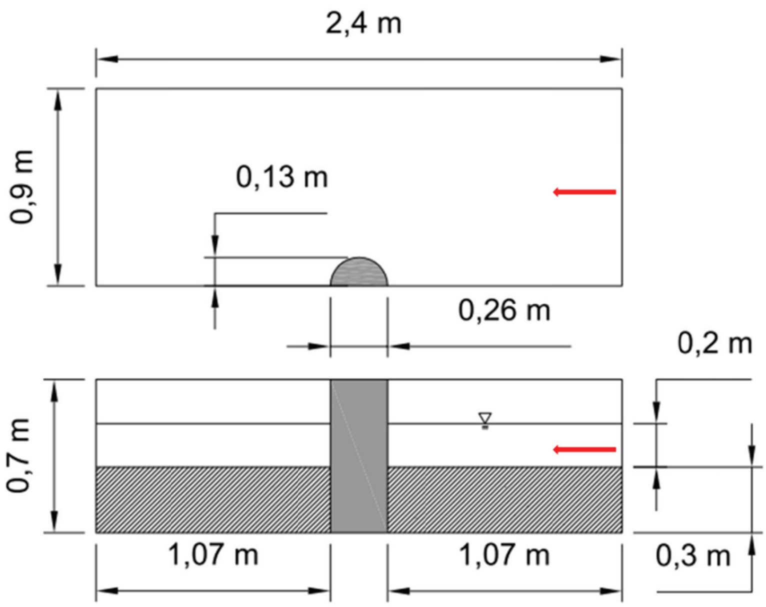


Fig. 3. Experimental setup by Dey and Barbhuiya (Dey & Barbhuiya, 2005).

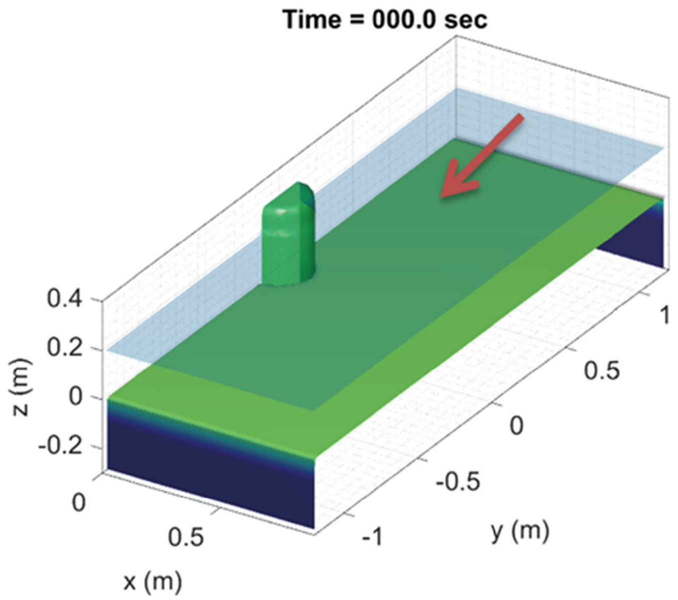


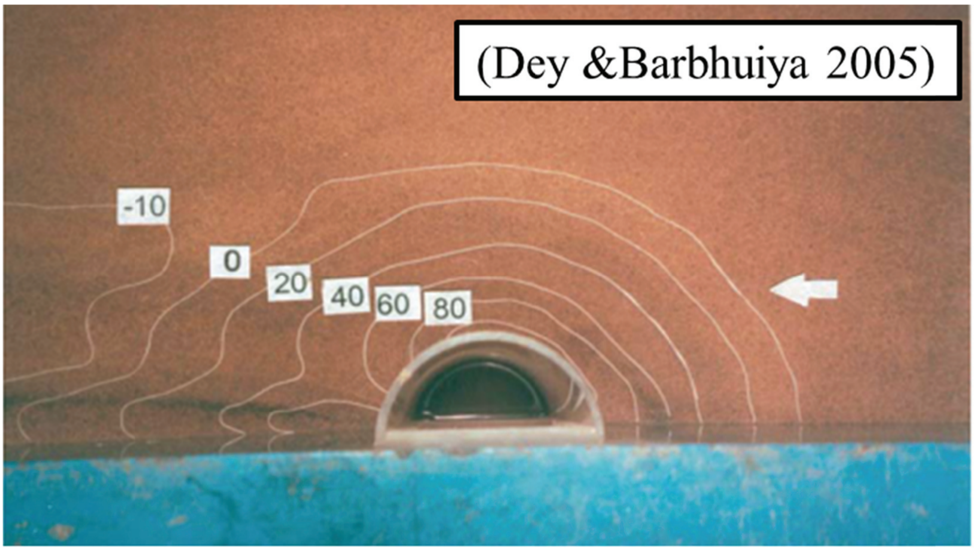
Fig. 4. Geometry of the numerical model for the simulation of scour prediction (by authors).

sand is 1,400 kg/m^3. The distribution of the scour depth by the numerical model is compared with experimental results (Dey & Barbhuiya, 2005) in Figure 5. Agreement is observed on the size of the upstream scour in experimental and numerical results. The range of the upstream scour hole is about the size of the diameter of the installed cylinder. The overall distributions of the scour hole are consistent pretty well. The range of the ultimate depth of scour hole is 0.08 to 0.1 m for simulation and experiment. Deviation is observed on the height of the downstream sediment. The height of the downstream sediment is 0.04 m for the numerical simulation, while it is 0.01 m in the experimental data. The over prediction would be alleviated by adjusting the employed physical and numerical parameters. The maximum scour depth, which is the focus of this study, can be appropriately calculated using the proposed model as indicated in this comparison.

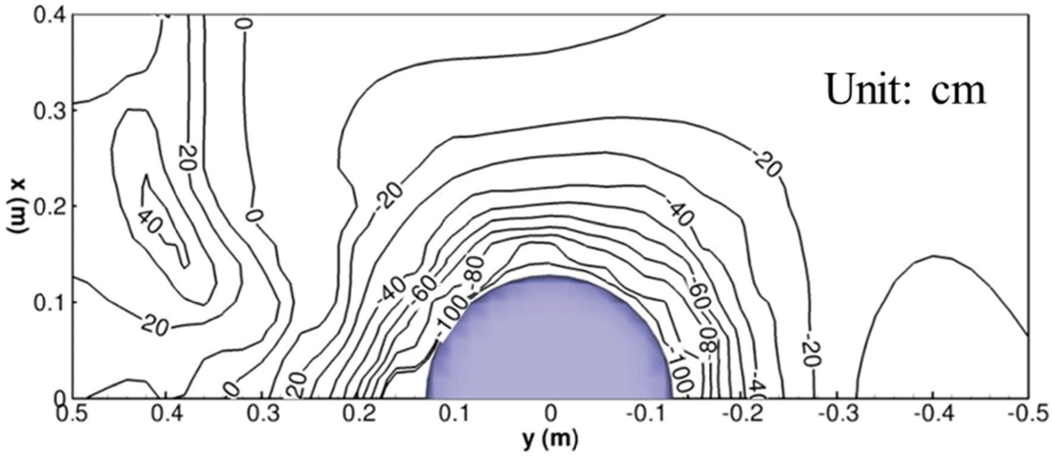
3.2 Scour development for a mono-pile substructure under constant speed current and regular wave

The effect of the regular wave is included with the current to investigate the development of the velocity distribution and scour depth around the mono-pile structure. The development of the horizontal velocity distribution within one period on the surface of seabed (z = 0.0 m) for the case of constant speed current and regular wave is shown in Figure 6. Positive value indicates that the direction of the velocity is the same with the flow (from left to right). Periodic variation of the flow distribution near the structure is observed, and the frequency is consistent with the incoming wave. The peak of the regular wave leads to positive velocity and the trough results in negative velocity on the seabed near the structure. Positive velocity is observed as the wave peak goes through the structure, while it is negative as the trough goes through the structure. The scour hole and its depth are developed as the peak and the trough go through the structure and sand nearby is taken to downstream as the sediment.

The contour for the scour distribution for the case of constant speed current and regular wave is shown in Figure 7. As the wave and



(a)



(b)

Fig. 5. Comparison for the distribution of scour depth by (a) experimental and (b) BVRM (Dey & Barbhuiya, 2005).

current go through the structure, the sediment moves to downstream with height about 2 m, and the deepest scour hole is observed on the circumference of the structure. From the variation of the scour depth within a period (111 sec to 120 sec) as shown in Figure 7, it is observed that the top of the sediment moves with the variation of the wave peak. It is also shown that the distribution of the scour hole varies with the phase of the regular wave, and the dynamic equilibrium is also observed with synchronized period. With the combined hydrodynamic load (current and wave), the maximum scour depth among the periodic variation is also about 2 m.

3.3 Comparison of the scour depth with observed data

The maximum scour depth/diameter of mono-pile (S/D) by the observed data (Matutano *et al*., 2013; Gerdes *et al*., 2008; Whitehouse *et al*., 2008;

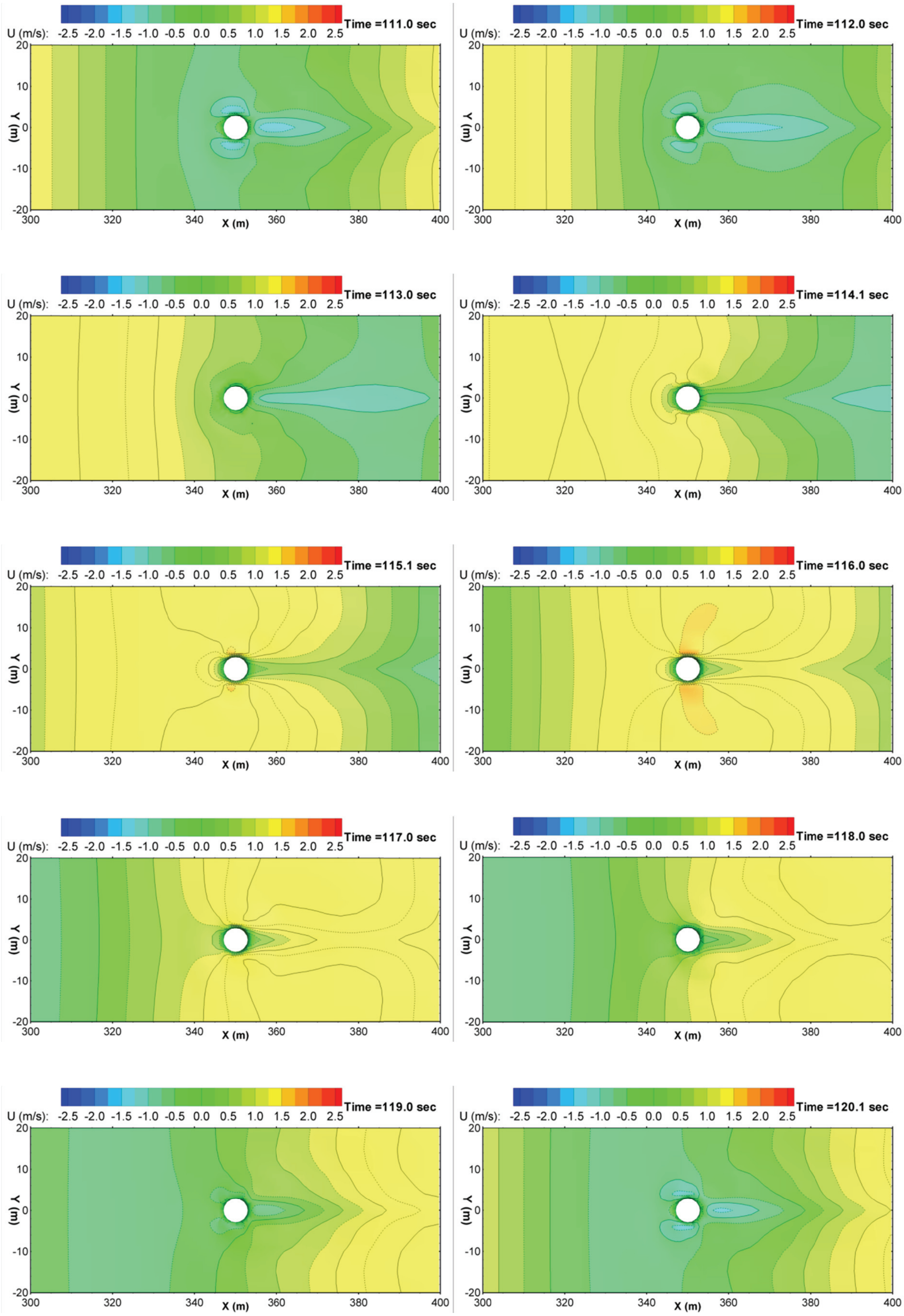


Fig. 6. Contour distribution of the horizontal velocity on the sand surface (z = 0.0 m) for the case of constant speed current and regular wave (by authors).

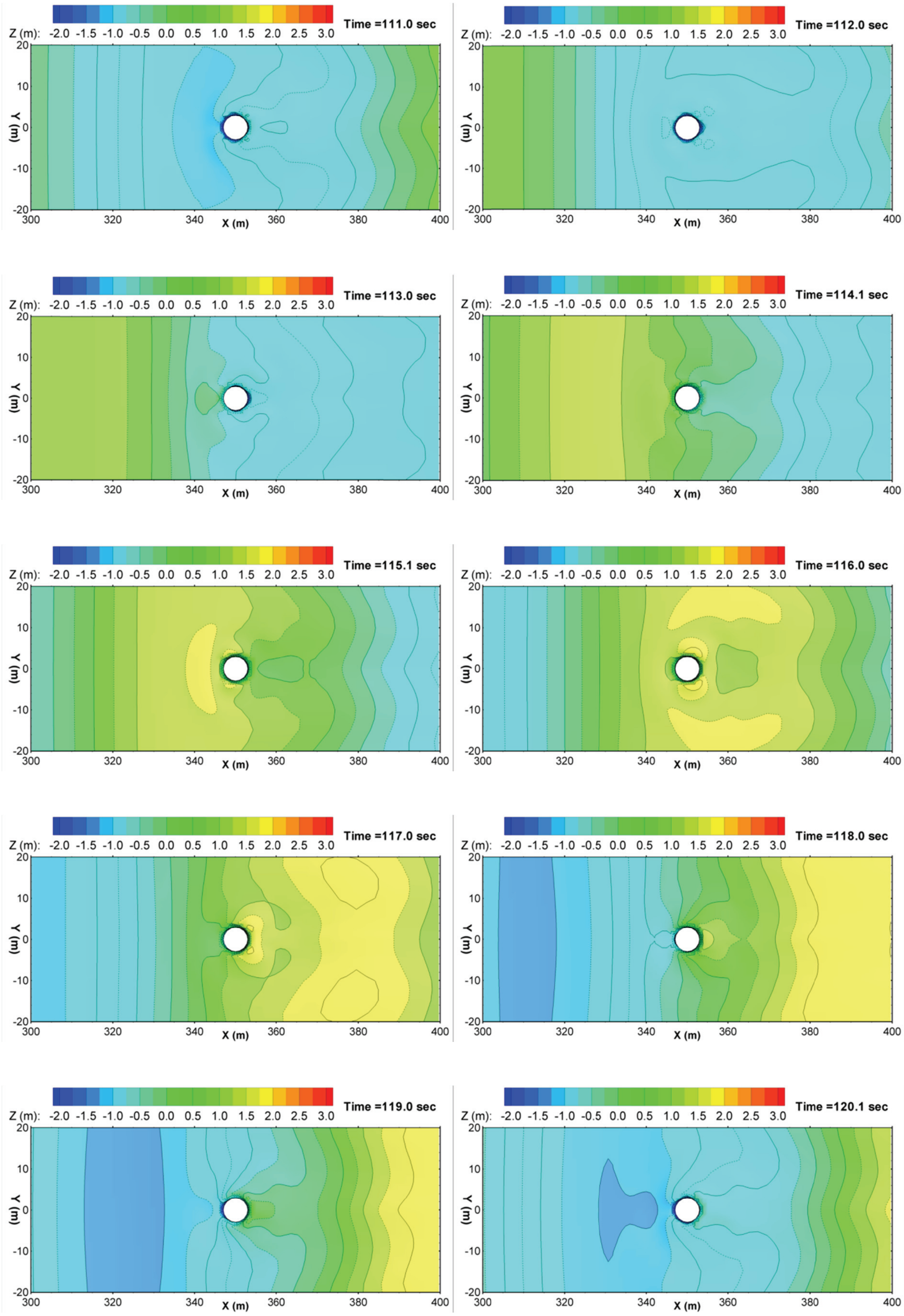


Fig. 7. Contour distribution of the scour depth for the case of constant speed current and regular wave (by authors).

Carroll *et al*., 2010; Whitehouse *et al*., 2011) is compared with the estimation by the proposed numerical model and the empirical formula (Veritas, 2011; Breusers *et al*., 1977) in literature as shown in Figure 8. The observed S/D is from 0.125 to 1.66, with the average value of 0.892. It is 1/3 by the present study, which is smaller than the average value of the observed data. The standard of DNV suggests the value of 1.3 (Veritas, 2011). Using the empirical formula proposed by Breusers (Breusers *et al*., 1977) leads to the value of 1.24 to 1.5. The soil properties and seabed composition will affect the maximum scour depth for the mono-pile structure. Prediction from the surveyed studies leads to the larger S/D value (Veritas, 2011; Breusers *et al*., 1977) comparing to the observed data as indicated in Figure 8. Although this paper predicts the smaller value of S/D, it is still within the range of the observed data.

4. Conclusion

In the present study, the development of the scour around the support structure for an OWT has been investigated. The interaction of the current, wave and structure are simulated by the developed code, Splash3D. The behavior of sand on the seabed is calculated by the developed BVRM. The comparison of the developed model with experimental case has been conducted. Results by the numerical model consistent well with compared case. In the comparison of scour distribution, the distribution of scour depth by the numerical model is comparable with experimental results. The size of upstream scour hole is comparable with the diameter of the cylinder for the experimental and numerical results.

The combined hydrodynamic load, including current and regular wave, are incorporated into

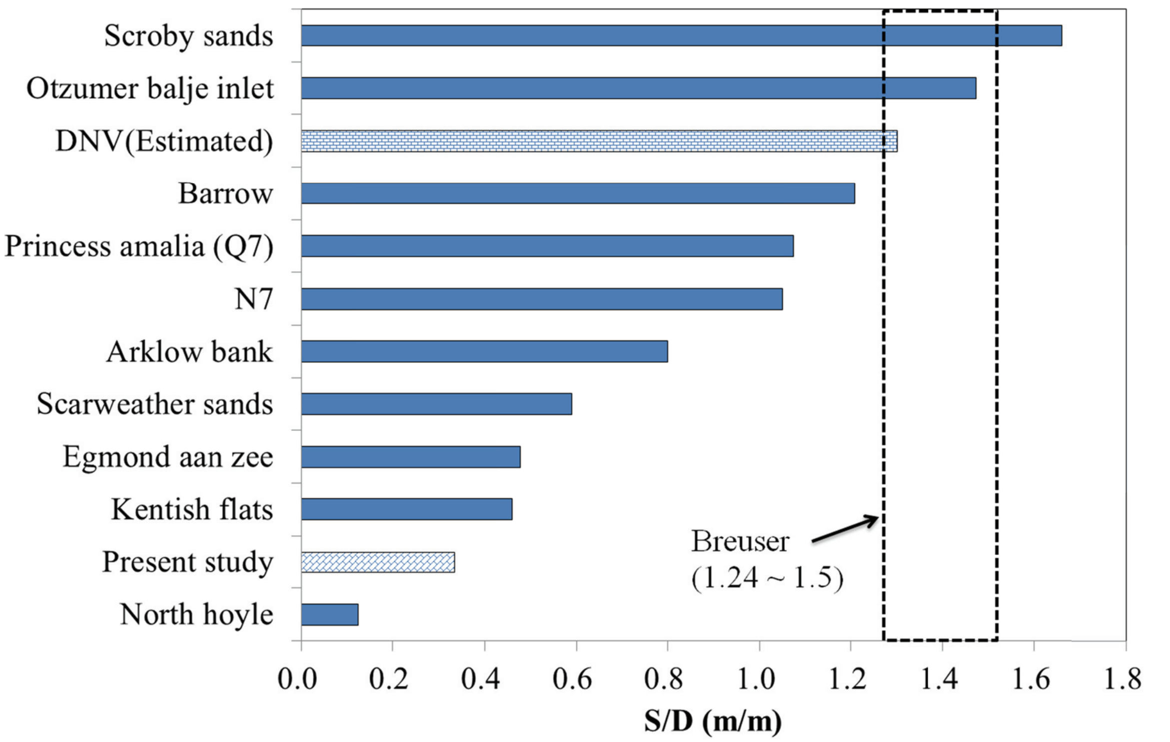


Fig. 8. Comparison of the maximum scour depth with observed data (Matutano *et al*., 2013; Gerdes *et al*., 2008; Whitehouse *et al*., 2008; Carroll *et al*., 2010; Whitehouse *et al*., 2011) and estimation (Veritas, 2011; Breusers *et al*., 1977).

the developed numerical model with the validated BVRM and the variation of scour distribution is presented and discussed. The maximum scour depth/diameter of mono-pile by the observed data is compared with the estimation by the proposed numerical model. The empirical formula in literature is also included into the comparison. Prediction from the surveyed studies leads to the larger S/D value comparing to the observed data. Although this paper predicts the smaller value of S/D, it is still within the range of the observed data.

The employed parameters of sand in the BVRM are taken from the experimental results of reference as the first step to develop the analyzing tool. When the data for the investigated region of the OWT is comprehensively available, it will be included into the BVRM. The evaluation based on the site-specific properties will be provided. The developed numerical model is applicable to predict the variation of scour distribution and the maximum scour depth around the support structure of the OWT can be evaluated. The effectiveness of the scour protection mechanism could be evaluated with the proposed scour prediction model in the future.

References

Assier-Rzadkiewicz, S., Mariotti, C. & Heinrich, P., 1997. Numerical Simulation of Submarine Landslides and Their Hydraulic Effects. *Journal of Waterway, Port, Coastal and Ocean Engineering*, 123(4), pp.149-157.

Baykal, C., Sumer, B. M., Fuhrman, D. R., Jacobsen, N. G., & Fredsøe, J., 2017. Numerical simulation of scour and back fi lling processes around a circular pile in waves. *Coastal Engineering*, 122(May 2016), pp.87-107.

Breusers, H. N. C., Nicollet, G. & Shen, H. W., 1977. Local scour around cylindrical piers. *Journal of Hydraulic Research*, 15(3), pp.211-252.

Calderer, A., Kang, S. & Sotiropoulos, F., 2014. Level set immersed boundary method for coupled simulation of air/water interaction with complex floating structures. *Journal of Computational Physics*, 277, pp.201-227.

Carroll, B., Cooper, B., Dewey, N., Whitehead, P., Dolphin, T., Rees, J., Judd, A., Whitehouse, R., & Harris, J., 2010. *A further review of sediment monitoring data*, UK: Cowrie Scour Sed-09 eBook.

Chen, M., Lin, C., Wu, T., & Huang, C., 2016. Hydrodynamic Analysis for a Jacket-type support structure of the offshore wind turbine. In *ICSREE 2016*. South Korea.

Deardorff, J. W., 1970. A numerical study of three-dimensional turbulent channel flow at large Reynolds numbers. *Journal of Fluid Mechanics*, 41(2), pp.453-480.

Dey, S. & Barbhuiya, A. K., 2005. Turbulent flow field in a scour hole at a semicircular abutment. *Canadian Journal of Civil Engineering*, 32(1), pp.213-232.

EWEA, 2015. *The European offshore wind industry - key trends and statistics 2014*.

Gerdes, G., Tiedemann, A. & Zeelenberg, S., 2008. *Case Study: European Offshore Wind Farms - A Survey for the Analysis of the Experiences and Lessons Learnt by Developers of Offshore Wind Farms*, Deutsche WindGuard, German Energy Agency (DENA), University of Groningen.

Hirt, C. W. & Nichols, B. D., 1981. Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics*, 39, pp.201-225.

Huang, W., Yang, Q. & Xiao, H., 2009. CFD modeling of scale effects on turbulence flow and scour around bridge piers. *Computers and Fluids*, 38(5), pp.1050-1058.

Jonkman, J., Butterfield, S., Musial, W., & Scott, G., 2009. Definition of a 5-MW reference wind turbine for offshore system development. *Contract*, (February), pp.1-75.

Kim, J., Moin, P. & Moser, R., 1987. Turbulence statistics in fully developed channel flow at low Reynolds number. *Journal of Fluid Mechanics*, 177, pp.133-166.

Kothe, D. B., J., R. W., Mosso, S. J., Brock, J. S., & Hochstein, J. I., 1996. Volume tracking of interfaces having surface tension in two and three dimensions. *AIAA Meeting Papers*, pp. 96-859.

Lemos, C. M., 1992. A simple numerical technique for turbulent flows with free surfaces. *International Journal for Numerical Methods in Fluids*, 15, pp.127-146.

Liu, K. F. & Mei, C. C., 1989. Slow spreading of a sheet of Bingham fluid on an inclined plane. *Journal of Fluid Mechanics*, 207, pp.505-529.

Liu, P. L.-F., Wu, T.-R., Raichlen, F., Synolakis, C. E., & Borrero, J. C., 2005. Runup and rundown generated by three-dimensional sliding masses. *Journal of Fluid Mechanics*, 536, pp.107-144.

Lugni, C., Brocchini, M. & Faltinsen, O. M., 2006. Wave impact loads: The role of the flipthrough. *Physics of Fluids*, 18, p.122101.

Marino, E., 2010. *An Integrated Nonlinear Wind-Waves Model for Offshore Wind Turbines*. TU, Braunschweig: University of Florence.

Marino, E., Borri, C. & Peil, U., 2011. A fully nonlinear wave model to account for breaking wave impact loads on offshore wind turbines. *Journal of Wind Engineering and Industrial Aerodynamics*, 99, pp.483-490.

Marino, E., Lugni, C. & Borria, C., 2013. A novel numerical strategy for the simulation of irregular nonlinear waves and their effects on the dynamic response of offshore wind turbines. *Computer Methods in Applied Mechanics and Engineering*, 255, pp.275-288.

Martinez-luengo, M., Kolios, A. & Wang, L., 2017. International Journal of Marine Energy Parametric FEA modelling of offshore wind turbine support structures : Towards scaling-up and CAPEX reduction. *International Journal of Marine Energy*, 19, pp.16-31.

Matutano, C., Negro, V., López-Gutiérrez, J.-S., & Esteban, M. D., 2013. Scour prediction and scour protections in offshore wind farms. *Renewable Energy*, 57, pp.358-365.

Nielsen, A. W., Liu, X., Sumer, B. M., & Fredsøe, J., 2013. Flow and bed shear stresses in scour protections around a pile in a current. *Coastal Engineering*, 72, pp.20-38.

Peder Hyldal Sørensen, S. & Bo Ibsen, L., 2013. Assessment of foundation design for offshore monopiles unprotected against scour. *Ocean Engineering*, 63, pp.17-25.

Prendergast, L. J., Gavin, K. & Doherty, P., 2015. An investigation into the effect of scour on the natural frequency of an offshore wind turbine. *Ocean Engineering*, 101, pp.1-11.

Raikar, R. V & Dey, S., 2008. Kinematics of horseshoe vortex development in an evolving scour hole at a square cylinder. *Journal of Hydraulic Research*, 46(2), pp.247-264.

Robertson, A. N., Wendt, F. F., Jonkman, J. M., Popko, W., Vorpahl, F., Stansberg, C. T., Bachynski, Erin E., Bayati, I., Beyer, F., Vaal, J. B. de, Harries, R., Yamaguchi, A., Shin, H., Kim, B., Zee, T. van der, Bozonnet, P., Aguilo, B., Bergua, R., Qvist, J., Qijun, W., Chen, X., Guerinel, M., Tu, Y., Yutong, H., Li, R., & Bouy, L., 2015. OC5 Project Phase I: Validation of Hydrodynamic Loading on a Fixed Cylinder. *Proceedings of the Twenty-fifth (2015) International Ocean and Polar Engineering Conference*, (April), pp.471-480.

Sebastian, T. & Lackner, M. A., 2012. Characterization of the unsteady aerodynamics of offshore floating wind turbines. *Wind Energy*.

Veritas, D. N., 2011. Design of offshore wind turbine structures. *offshore standard DNV-OS-J101*.

Whitehouse, R., Harris, J. & Rees, J., 2008. *Dynamics of scour pits and scour protection-synthesis report and recommendations (milestones 2 and 3)*, Department of Energy and Climate Change.

Whitehouse, R. J. S., Harris, J. M., Sutherland, J., & Rees, J., 2011. The nature of scour development and scour protection at offshore windfarm foundations. *Marine Pollution Bulletin*, 62(1), pp.73-88.

Wienke, J. & Oumeraci, H., 2005. Breaking wave impact force on a vertical and inclined slender pile-theoretical and large-scale model investigations. *Coastal Engineering*, 52(5), pp.435-462.

Wu, T. R., 2004. *A Numerical Study of Three-Dimensional Breaking Waves and Turbulence Effects*. Cornell University.

Yang, D. & Shen, L., 2011. simulation of viscous flows with undulatory boundaries. part i: Basic solver. *Journal of Computational Physics*, 230, pp.5488-5509.

Zanke, U. C. E., Hsu, T.-W., Roland, A., Link, O., & Diab, R., 2011. Equilibrium scour depths around piles in noncohesive sediments under currents and waves. *Coastal Engineering*, 58(10), pp.986-991.

離岸風機單樁式支撐結構之水動力與沖刷分析

陳銘宏[1*] 林君蔚[2] 吳祚任[3] 黃金城[4]

摘 要

本文主要探討離岸風機單樁式支撐結構之沖刷坑發展過程。海床底部之泥沙移動現象主要由BVRM模型所描述。BVRM模型計算結果與所比對之文獻結果趨勢一致。接著採用包括洋流與規則波等水動力負載進行離岸風機支撐結構沖刷坑發展現象分析，並將無因次化參數-最大沖刷坑之深度/樁徑與文獻結果進行比對。文獻當中採用簡易模型之結果(1.24 ~ 1.5)高於實際觀測值(0.125 ~ 1.66)。而本研究之數值模型計算出之結果數值較低(0.33)，但還在所蒐集之文獻數值範圍當中。本研究建立之數值模型可用於評估離岸風機支撐結構周圍之沖刷坑最大深度與範圍。

關鍵詞：離岸、風機、沖刷、數值分析

[1]核能研究所機械及系統工程專案計畫研究助理
[2]中央大學水文與海洋科學研究所博士候選人
[3]中央大學水文與海洋科學研究所教授
[4]核研所機械及系統工程專案計畫研究員
*通訊作者, 電話: 03-4711400#3351, E-mail: minghongchen@iner.gov.tw

收到日期: 2017年05月26日
修正日期: 2017年07月06日
接受日期: 2017年07月26日