 doi:10.20944/preprints202007.0723.v1

REVIEW OF FLUID-STRUCTURE INTERACTION MODEL IN A NUMERICAL WAVE TANK WITH OFFSHORE STRUCTURES NEAR THE FREE SURFACE

MUSHTAQ AHMED[1*], ZAFARULLAH NIZAMANI[1], AKIHIKO NAKAYAMA[1], MONTASIR OSMAN[2]

[1] Department of Environmental Engineering, Faculty of Engineering and Green Technology, Universiti Tunku Abdul Rehman, 31900, Kampar, Perak, Malaysia.

[2] Department of Civil Engineering, Faculty of Engineering, Universiti Techologi PETRONAS, Persiaran UTP, 32610, Seri Iskandar, Perak, Malaysia.

ABSTRACT

The effects of the surface waves generated by the wind have a significant effect on the currents. A wave current coupled model plays an important role in the design of offshore structures. The interaction between fluids such as incompressible ocean waves and current and offshore structures is significant with many real-time applications in offshore engineering. These coupled models can be applied to Offshore Floating Production Operating and offloading (FPSO), Wind or current turbines and offshore pipelines. The complex issues related to the design are analyzed by using Computational Fluid Dynamics, which requires an investigation of the multiphase flow between wave and current and the structure which is considered restrictive due to the computational cost. If viscous effects are neglected then the single-phase flow models have been recommended, where wave-current interaction have been modelled successfully. Models have been developed where velocities and pressure are computed and the results can be verified with the experimental results available in the literature. In this study the existing numerical methods, mesh types are discussed along with their coupling methods. Here single-phase and multiphase models with small and medium movement are reviewed and their applications are highlighted.

Keywords: Offshore fixed floating structures, wave current coupling, numerical modelling

1. Introduction

The offshore structures serve not only for Oil and gas exploration but they are also used for the degeneration of energy from non-renewable energy sources like ocean winds, waves and currents such as windmills, current energy generator. The deepwater offshore structures are more vulnerable to environmental loadings such as wind-induced water waves and currents than shallow water. To make them operational throughout their design life; it is utmost necessary to ensure the safety of such structures. In past; due to their location and dealing with sensitive material, and environmental hazards associated with the fossil fuels these structures had been designed with conservative loadings. Recently, due to advance in shell technologies the hydrocarbon fuel prices have decreased. It is time to design offshore structures efficiently to reduce the extra costs associated with them. For proper structural design, it is necessary to consider the real environmental loads on the structure. The coupled and non-coupled wave-current

[*]Corresponding Author
Mushtaq Nizamani (*nizmani.mushtaq@gmail.com*)

 doi:10.20944/preprints202007.0723.v1

interaction has been observed in many cases on the structure [1]. The analysis of such interactions needs different mathematical approaches, observations and experiments. This interaction has been considered as the most complex phenomenon, where current is superimposed with waves.

Initially, studies were performed in laboratory-based wave tanks but after the computational evolution, the experimental tanks have been replaced by Computational Fluid Dynamics (CFD) numerical wave tanks involving computational costs only. The combined wave-current interaction based on Navier-Stokes equations has been studied by various researchers [2, 3, 4, 5, 6, 7, and 8]. Most of these studies are done under Numerical wave tank simulations with different CFD solver techniques. Various open-source and commercial codes are available for the simulation of flow dynamics. The accuracy of such simulation depends upon the comparison of model validation results with available experimental works. The experimental works regarding wave and current interaction have not been much studied due to limitations of cost, space and measurements. Under such circumstances, the studies of one CFD techniques could be compared with the results of other CFD technique as done in the study of [7 & 9].

The involvement of current with waves make flow characteristics complex. The superimposition of current onto the wave in a numerical wave tank can be achieved either by introducing pressure inlet boundary condition [4 & 6] or by velocity inlet boundary condition [3 & 11]. The velocity of waves remains the same throughout the depth with velocity wavemaker while waves generated with pressure wavemaker have varying current velocities with the depth. The FLUENT is a CFD commercial software of ANSYS incorporation based on Finite volume method. The software is very effective in modelling the physical phenomenon of flow and turbulence. The world supercomputing record has been set by this software with scaling to 172,000 cores. It is very effective for RANS simulations. It has been successfully used in the studies of [3, 4, 7, 8, 11, and 12].

2. Previous Studies

The simultaneous occurrence of wave and current in ocean environments make their interaction as the most prominent factor for the analysis of hydrodynamics of oceans [5]. The major environmental forces for this region are considered to be waves and currents [6]. Every offshore structure before its design requires the analysis of such hydrodynamic forces. Offshore structures have been studied by many authors using frequency domain analysis [13, 14, 15] and Time domain analysis [13, 15, 16, 17]. The forces have been determined by using the Morison equation and linear diffraction wave theory. The main determination of the study involved capturing random wave responses in a surge, heave and pitch [18]. Such a response is called Response Amplitude Operator (RAO).

The wave-current interaction with structures involved the numerical approaches in these studies. This numerical approach is based on the potential flow theory, linear waves and small current velocity approximation. The coexisting field of the wave with current involves the separation of the velocity potential. This separation is done into the unsteady potential of wave and current. It becomes easier to determine the elevation of water surface around an offshore structure under the coexisting field of wave and current. The unsteady wave potential and current velocity are substituted in first-order dynamic surface boundary condition to obtain the water surface elevation [19]. One of the studies involved the time-domain method [13]. The regular waves diffraction and radiation effects were determined due to current with a 2-D body. The observation proved the current effects on offshore structures and a non-linear relation with currents was shown with first and second-order results.

Surface wind stress, bottom friction, wave climate, wave field, depth and current refraction and modulation of the absolute and relative wave period are the most common wave-current interaction mechanisms reviewed [16,17,18]. The change is slow drift motions, wave run-up and wave forces have been observed due to the combination of waves and currents [19]. It has been observed that wave and current occur at the same time in the oceans. A resonance is normally created when wave and current interact with the structure. This resonance causes slow drift motions. The prediction of such motions is also very important from the engineering perspective. The pure wave interaction with the offshore structures has been observed differently as compared to wave current coupled interaction. When time-domain results were compared with the frequency domain, no difference was found in these studies.

The long-crested wave along with current produce more forces as compared to short crested wave involving current. The experimental study of [20] made comparisons between short crested wave with current and long-crested wave with currents. It was found in the study that long-crested wave with the current was able to produce 45% more forces as compared to short crested wave with the current. The author of the study suggested that further study was required to be conducted regarding short crested waves involving six-degree-of-freedom. It has been observed that in intermediate and shallow waters, the short crested wave remains dominant while for the deep waters long-crested waves are dominant. Due to lack of proper data, the offshore structures are designed for the long-crested waves.

The short crested waves along with current produce different results than the long-crested waves with the current. In the analytical study of [2], the effects of the current having uniform velocity were studied on the short crested wave. The wave parameters on which the effect was studied were wave frequency, wave run-up, wave force and inertia. The water depth was found to be in inverse relation with the short crested wave having uniform current and total wave number was having a direct relation. The wave frequency was influenced by the current velocity and current incident angle. Details of various numerous studies involving wave, current and structure have been briefly described with the purpose, observations and suggestions of the study in Table 1.

3. Wave and Current Fields in offshore environments

The dimensionless quantities are the key to the numerical analysis of flow so non-dimensional parameters are preferred. The most used non-dimensionless parameters are, wave height (H/gT^2), wave steepness (*H/L)* and water depth (d/gT^2). In a numerical wave tank, the waves are generated at the inlet to simulate the wave parameters as shown in Figure 1. Without the presence of current, the free surface displacement Eq. 1 and velocity potential Eq. 2 can be written as;

$$\eta = \frac{H}{2}\sin(kx - \omega t) \tag{1}$$

$$\phi = \frac{g\,H\,\cosh\,k(h+z)}{2\,\omega\,\cosh\,k\,h}\cos(kx - \omega t) \tag{2}$$

Where, η is free surface displacement, H is wave height, $(kx - \omega t)$ is wave phase angle, $\emptyset$ is velocity potential, g is the acceleration due to gravity, h is water depth, k is wave number, and t is time.

Preprints (www.preprints.org) | NOT PEER-REVIEWED | Posted: 30 July 2020 doi:10.20944/preprints202007.0723.v1

Table 1: Some recent studies on Wave-current interaction with structures

Author & Year of Study	Nature of Study			Purpose of the study	Observations and Suggestions	Ref. No
	Exp.	Anal.	Num.			
Wang (2019)	-	Anal.	-	Short crested wave-current interactions with composite bucket foundation for the windmill. Effects of uniform current on wave frequency, wave run-up, wave force and inertia were investigated. Drag coefficients on composite bucket foundation	Total wave number has a direct relation with the short crested wave-current interaction and inverse relation with the water depth. The inertia and drag coefficient increases under increased velocity only if the relative angle is smaller than 90. The current velocity and current incident angle also influence the wave frequency. Main incident forces occur under the condition of current incident forces being parallel to the direction of wave propagation.	2
C.Y.Ng (2017)	Exp.	-	-	The dynamic responses of Truss spar were determined under the conditions of short crested wave and long-crested wave with currents. A comparison of both waves with the current was made.	The long-crested waves with current were found to produce (45%) more responses than short crested waves with the current. Suggestion 1: Further study needs to be conducted by considering wind loads also. Suggestion 2: The study should include surge, sway, heave, roll, pitch and yaw (Six degree-of-freedom).	20
Kim (2016)	-	-	Num.	To Investigate the combined wave and current loads on a cylinder under the regular wave environment.	Under combined wave and current situations, an increase in current velocity increases the wave height and wavelength. Suggestion 1: numerical simulations for actual jacket structure because complex flow interactions occur at the multiple legs of the platform. Suggestion 2: Vortex induced motion (VIM) and vortex-induced vibrations (VIV) around the structure.	3
C.Y.Ng (2016)	Exp.	-	-	Wave-current interaction with truss spar	Dynamic responses of the truss spar model due to wave with current are greater as compared to wave without current. Maybe because current velocity provides additional lateral force in the water body which increased the structure resonance. Dynamic responses were found to be greater under wave with the current. One-unit increase in the current velocity was found to increase surge by 24.8%, heave by 31.1% and pitch by 32.2%.	21

Preprints (www.preprints.org) | NOT PEER-REVIEWED | Posted: 30 July 2020 doi:10.20944/preprints202007.0723.v1

Windt (2018)	-	-	Num.	Development of an impulse source based WCI model with simultaneous generation of current velocity profile (CVP) and Free Surface Wave (FSW) in a numerical wave tank (NWT).	Impulse source wave maker is able to accurately create the desired wave field. The tank layout effectively shields the current generation area from wave action. Generated waves can pass through the numerical beaches. Suggestions 1: Further work is required to recreate flow profiles accurately. Suggestions 2: the tendency of decreasing wave height and phase shift for waves travelling in current direction is captured correctly.	6
Zhang (2016)	-	-	Num.	WCI in the bottom boundary layer to obtain the periodic velocities in the bottom boundary layer to know the degree of agreement with the experimental work. Total mesh cells were fixed 200000 due to limited computational power so no more refined mesh.	Turbulent current was generated by a pressure-inlet boundary condition and a pressure-outlet condition at the other end. Wave was generated by giving a velocity-inlet boundary condition. Reflected waves were absorbed the numerical beach which is based on adding a sink term to the momentum equation. The convergence test was carried out and basic test took 60 hours (Normal for numerical studies). For smooth best, results show good agreement with the experimental results for mean velocities and periodic velocities. Results could be helpful for wind farms designers and sediment transport studies. Suggestion: Further research need to be carried out to investigate the equivalent phenomenon under real wave environments either by numerical experiments or by field tests.	7
Zhang (2016)	Exp.	-	-	Wave-currents effects in free surface turbulent flow where waves of different frequencies were superimposed on current field. Changes in mean velocities, turbulence intensities, and Reynolds shear stress were investigated.	Changes in velocity profile were observed. Wave presence causes the reduction in mean velocity near free surface. Near bed changes are not much significant for intermediate and deep waters. When waves propagate along with current. The resultant Reynolds stress increases rapidly near the bed decreases until mid and finally changes sign in the upper region. Suggestion 1: The investigation of interaction between irregular wave with current be carried out. Suggestion 2: In present study, wave frequency was studied mainly. Studies involving wavelength need to be carried out.	5

 doi:10.20944/preprints202007.0723.v1

The free surface displacement has two parameters associated with it i.e. Kinematic and dynamic both of which need to be satisfied with velocity potential. If pressure p_η is assumed to be consistent at free surface $z = \eta(x,t)$ than dynamic boundary condition for the free surface can be represented with the Bernoulli equation as eq. (3):

$$-\frac{\partial\phi}{\partial x} + \frac{1}{2}|\nabla\phi^2| + gz = C(t)z = \eta(x,t) \tag{3}$$

The description of water particle in the horizontal direction can be made as:

$$u = -\frac{\partial\phi}{\partial x} = Ak\ \cos h\, k(h + z)\sin(kx - \omega t) + U \tag{4}$$

$$u^2 = U^2 + 2AkU\ \cos h\,(h + z)\sin(kx - \omega t) + (Ak)^2\ \cos h^2\, k(h+z)\sin^2(kx - \sigma t) \tag{5}$$

Where Ak is the velocity component for the horizontal direction and it is small in magnitude so $(Ak)^2$ is just negligible. So now the Eq. (3) for *z*=0 can be written as Eq. (6) or eq. (7):

$$C(t) = \frac{1}{2}[U^2 + 2AkU\ \cos h\,(h + z)\sin(kx - \omega t)] - A\omega\ \cos h\, kh\, sin(kx - \omega t) + g\eta \tag{6}$$

Or

$$\eta(x,t) = -\frac{U^2}{2g} + \frac{A\omega}{g}\left(1 - \frac{kU}{\omega}\right)\cos h\ kh\sin(kx - \omega t) + C(t) \tag{7}$$

The water particle velocity should be within specified physical conditions to satisfy the boundary condition so if satisfied, water particle motion will be considered as the kinematic boundary condition. So under this situation, $z - \eta(x,t) = 0$ at any arbitrary moment in the boundary. Now kinematic boundary condition at the free surface will be as shown in Eq. (8);

$$\left(\frac{\partial\eta}{\partial t} - \frac{\partial\phi}{\partial x}\frac{\partial\eta}{\partial x}\right) + \eta\,\frac{\partial}{\partial z}\left(\frac{\partial\eta}{\partial t} - \frac{\partial\phi}{\partial x}\frac{\partial\eta}{\partial x}\right) + \cdots = -\frac{\partial\phi}{\partial z} - \eta\,\frac{\partial}{\partial z}\left(\frac{\partial\phi}{\partial z}\right) + \cdots, \qquad z = 0 \tag{8}$$

When we arrange Eq. (8) by linear terms, it can be rewritten as Eq. (9):

$$\frac{\partial\eta}{\partial t} + U\,\frac{\partial\eta}{\partial x} = -\frac{\partial\phi}{\partial z}, \qquad z = 0 \tag{9}$$

Preprints (www.preprints.org) | NOT PEER-REVIEWED | Posted: 30 July 2020 doi:10.20944/preprints202007.0723.v1

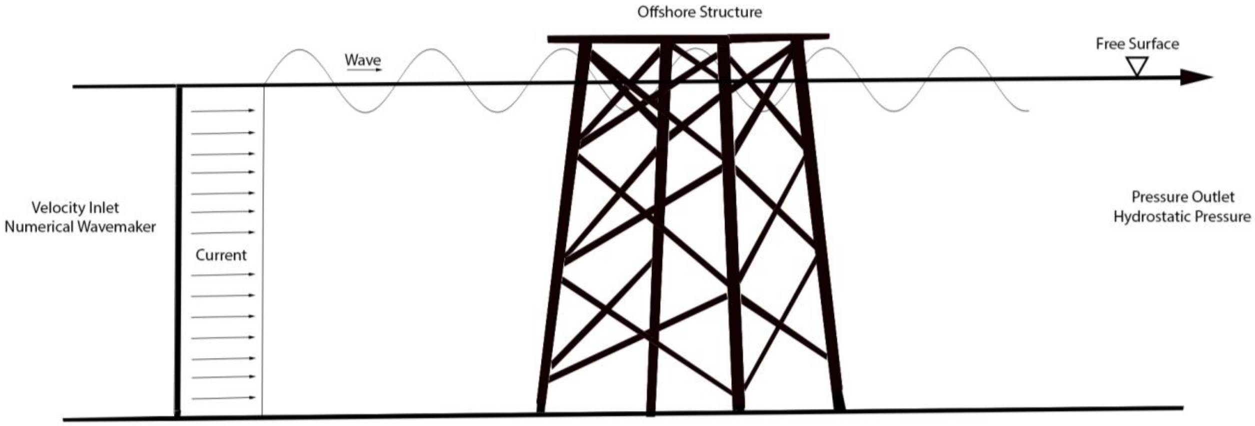

Figure 1. Sketch of wave and current fields near offshore structure in the offshore environment

The currents are directed motions of seawater generated by the action of earth rotation, wind, temperature, water salinity and gravity of sun and moon. They are not like waves and flows at greater distances in the oceans and play a vital role in the earth's climate. They can be divided into deep water currents and surface water currents. The deepwater currents move under the influence of temperature and density of water while the surface currents move under the influence of wind.

By assuming a uniform current for the whole depth of the tank, the velocity potential of Eq. (2) can be rewritten as Eq. (10):

$$\phi = A \cosh k(h+z)\cos(kx - \omega t) - Ux \tag{10}$$

Where U is current velocity, H is wave height, k is wave number, h is water depth, ω is frequency, and $A = \frac{g H}{2 \omega \cosh k h}$.

The amplitude of wave with the consideration of current can be written as Eq. (11):

$$A = \frac{g H}{2 \omega \left(1 - \frac{kU}{\omega}\right) \cosh k h} \tag{11}$$

By putting, η and ϕ in equation (6), we can obtain the equation for the dispersion relation with current velocity (U) as shown in Eq. (12);

$$\omega^2 = \frac{g k \tanh kh}{\left(1 - \frac{kU}{\omega}\right)^2} \tag{12}$$

$$\omega = kU + \sqrt{g k \tanh kh} \tag{13}$$

$$\omega_e = \omega - kU \tag{14}$$

Where ω is the absolute frequency and ω_e is encounter frequency.

Preprints (www.preprints.org) | NOT PEER-REVIEWED | Posted: 30 July 2020 doi:10.20944/preprints202007.0723.v1

4. Numerical Wave Tank Approach

The wave, current and their interaction can be measured by different technologies currently available. Previously, the ocean environments have been studied on the laboratory basis only. But due to limitations involved with the experimental works, the wave-current interaction has not extensively been studied under field conditions. The advancement in the computational field has created new opportunities for fluid dynamics. The most cost-effective way to study such an environment is by the use of computational fluid dynamics (CFD) techniques. These techniques involve the modelling of ocean waves and currents in open source and commercial computer codes.

The numerical simulation with the help of CFD has enabled us to properly create the ocean like environments in numerical tanks. The promising results have been obtained to various researchers who simulated ocean flows in numerical tanks using Reynold's Average Navier-stokes (RANS) equations. Numerical wave tank flumes are developed in commercial and open-source CFD codes. These flumes are mostly based on the finite volume method (FVM). The Stokes and continuity equations are used as the governing equations. Depending on the flow conditions, either Navier-stokes (NS) or Reynolds Average Navier Stokes (RANS) equations are used to model the incompressible flow. The volume of fluid is the most suitable method for tracking the free surface. Table 2 shows some of the recent studies of numerical wave tank simulation with RANS equations. Most of the Numerical tanks mentioned in Table 2 have applied the turbulent model. All the numerical wave tank use the artificial damping zone to reduce the reflection effects of the waves generated in the tank. Every model involves initial and boundary conditions. The waves and currents are generated at the inlet of the tank while the reflected waves are absorbed at the outlet of the tank. The numerical wave-current flume shown in Figure 2 gives the full details of initials and boundaries.

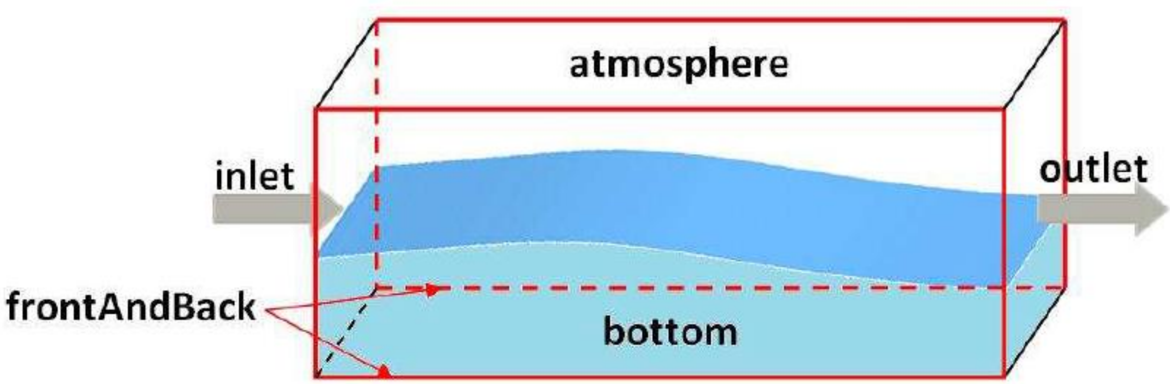

Figure 2. A numerical wave tank defined with each of its boundary [30]

Providing the most favourable initial and boundary conditions is the key to towards the successful simulations. Some of the studies under review involved the pressure based waver maker at the inlet boundary of the wave tank [4, 6]. While on the contrary, the studies of [3, 11] involved velocity based wavemaker. When the authors compared their obtained results with the results of experimental works, the studies involving velocity wavemaker gave the best comparisons. The pressure-based studies provided the best results for the wave condition only. While for current and wave-current interaction, their simulation was not promising enough. The wave tank based on the pressure as a current producing source provided

Preprints (www.preprints.org) | NOT PEER-REVIEWED | Posted: 30 July 2020 doi:10.20944/preprints202007.0723.v1

fluctuating current values concerning the depth of the water while in case of velocity, the value of current remained the same throughout the depth. The author of study [6] concluded that the location of the current source in the model should be further studied to get the best results for the current and wave-current interaction cases too.

The RANS equations are most common among the researchers involving less computational costs. The simulations based on RANS equations provide the best simulation results. The accuracy of simulations mainly depends upon the boundary conditions chosen. Most of the studies mentioned in this paper involve the same source region for waves and current except the study of [6] where two separate sources were chosen. This study also involved two separate beaches too, to avoid the reflection effect of these sources. Most of these studies were performed in the two-dimensional domain. Involvement of the Three-dimensional domain consumes an extra computer memory and requires some special computers. The accuracy of such simulation depends upon the comparison of model validation results with available experimental works. The experimental works regarding wave and current interaction have not been much studied due to limitations of cost, space and measurements. Under such circumstances, the studies of one CFD techniques could be compared with the results of other CFD technique as done in the study of [7, 9].

Most of the studies mentioned in Table 2 were carried out with commercial CFD solver "FLUENT" which is the most popular tool for the simulations of turbulence. Some studies have involved the use of open source tools too. The most popular CFD opensource software is OpenFOAM. The commercial software is intended to provide ease to the user while open source software involves not many favourable user-friendly interfaces.

Kim et al [3], studied wave-current interaction and load analysis using finite volume method (FVM) based numerical wave tank. CFD studies were done in FLUENT software, where governing equations were discretized by the finite volume method. He used 3rd order Monotone upstream centred scheme (MUSCL) for convection term. The velocity and pressure coupled with Pressure implicit with the splitting of operator (PISO) method. The k-ε model was used for the turbulent stress term in the governing equation. He used Volume of the fluid method with implicit High-resolution interface capturing (HRIC) for the air-water interface.

Zhang et al. [4], developed a numerical wave tank in FLUENT software by using Navier Stokes equations to study the wave and current interaction in the bottom boundary layer. The flow was modelled by RANS equations and the Volume of the fluid method was used to track the free surface. K-ε model was used and at the bottom boundary layer, a low Reynolds number scheme was used instead of high Reynolds's Number scheme. The waves generated were second-order Stokes waves by using velocity inlet boundary. The validation of the wave tank was made by using the experimental data of Kemp, [23]. The implicit high-resolution interface capturing method was used.

Christine et al. [6], studied wave-current interaction without involving any offshore structure by using impulse-based wave-current interaction (WCI) model. Waves and currents were simultaneously generated through the inclusion of source terms, added to the Reynolds Averaged Navier-Stokes (RANS) equations. He used depth varying current velocities but his model was unsuccessful to produce the best results for the wave-current interaction. It gave the best results for the waves only.

Preprints (www.preprints.org) | NOT PEER-REVIEWED | Posted: 30 July 2020 doi:10.20944/preprints202007.0723.v1

Table 2 Wave tank simulations with different CFD techniques involving wave, current and structures.

Author	Year	CFD Solver	Turbulence Model	CFD code	Free surface	NWT TYPE	Dimensions (m)			Wave Type	Wave Parameters		Ref.
							L	W	D		Height	Period	
Hans	2019	RANS	k-w	REEF3D	LSM	2D,3D	38	5	7	Regular	1.6-1.7	4.0-6.0	10
Christine	2019	RANS	-	OpenFOAM	VOF	2D	-	-	0.74	Regular	0.037	1.1	6
Salwa	2017	RANS	-	Python	VOF	2D,3D	20	10	4	Regular	-	5.3	31
Dagli	2017	RANS	k-e	FLUENT	VOF	-	-	-	-	Regular	-	-	32
Paci	2016	RANS	k-e	OpenFOAM	VOF	3D	-	-	-	Regular	0.25-3	1.5-2.0	33
Kim	2016	RANS	k-e	FLUENT	VOF	2D	45	-	2	Regular	0.1	1.41	3
Zhang	2016	RANS	K-e	FLUENT	VOF	2D	12.86	-	0.2	Regular	0.02	1	4
Silva	2016	RANS	k-e	CFX	-	2D	30	-	2.1	Regular	0.08	1.80	8
Zarruk	2015	LES	-	-	VOF	3D	1	0.3	0.26	Solitary	0.2	-	34
Sonia	2015	RANS	k-e	FLUENT	VOF	3D	10	1.8	1.25	Regular	-	-	11
Rodriguez	2015	LES	-	-	-	-	-	-	-	-	-	-	36
Persic	2014	LES	-	OpenFOAM	VOF	-	-	-	-	-	-	-	22
Zhang	2014	RANS	k-e	-	VOF	3D	25	7	20	Solitary	4.0	-	5
Chen	2014	RANS	k-w	OpenFoam	VOF	-	-	-	-	-	-	-	35
Anton	2014	RANS & LES	k-e	FLUENT	VOF	-	-	-	-	-	-	-	7
Leonardo	2014	RANS	k-w SST	OpenFOAM	VOF	-	-	-	-	-	-	-	37
Markus	2013	URANS	k-w SST	OpenFOAM	VOF	-	-	-	-	-	-	-	24

Hans et al. [10], used the Finite volume method (FVM) by using OpenFOAM. He used the 5th order Finite difference weighted Essentially Non-Oscillatory (WENO) scheme for the field equations. He used the projection method to calculate the pressure in the Navier-stokes equation. The K-ω model was used along with Reynolds Averaged Navier-Stokes (RANS) equations for turbulence closure. Cartesian grid approach was utilized for high order discretization scheme. Ghost cell immersed boundary method (GCIBM) was employed for wave structure analysis.

Markus et al. [24 & 25], also used the same approach as [3] but he used Fenton model [26] to create the depth varying currents The results were found to be more successful but he used nonlinear waves with unsteady flow conditions using Unsteady Reynold's Averaged Navier-Stokes (URANS) equations. He used stream function wave theory to validate his waves generated in the wave tank and compared the results of non-uniform current with uniform currents. His studies mostly involved wave kinematics and no offshore structure was used in the model.

5. Summary and Conclusion:

For the safety of offshore structures, it is important to account all the environmental loads acting on the structure. Previously, these forces were assessed and analysed using experimental and analytical approaches. The advancement in computational power has replaced the method of analysis from the traditional one to computational fluid dynamics (CFD). Various CFD Tools including open source and commercial software are available now a day. Open-foam and ANSYS FLUENT software has most widely been used for the application of CFD in offshore engineering. Due to the user-friendly interface, the fluent software has been the choice of researchers for the applications of CFD in offshore engineering. These studies have taken the simplest form of offshore structure but due to the complex interaction of wave-current with offshore structures, it is necessary to use the geometry of offshore structure close to the actual one.

Acknowledgement

This research work is supported by the Universiti Abdul Rahman (UTAR) under the research funded by an FRGS grant from the Ministry of Higher Education of Malaysia (FRGS/1/2018/TK01/UTAR/02/4).

References

[1] Olabarrieta, M., Medina, R., & Castanedo, S. (2010). Effects of Wave- Current Interaction on the Current Profile. *Coastal Engineering*, 643-654.

[2] Wang, C. H., Wai, W. H., Li, Y. S., & Chen, Y. (2006). Modelling of the Wave-Current Interaction in the Pearl River Estuary. *Conference of Global Chinese Scholars on Hydrodynamics*, (pp. 159-165). Hong Kong, China.

[3] Kim, S.Y., (2016). Numerical simulation of wave and current interaction with a fixed offshore substructure.

[4] Zhang, X., Simons, R., & Buldakov, E., (2016). A Numerical Study of wave-current interaction in the bottom boundary layer. *Coastal Engineering*

[5] Zhang, J., S., Zhang, Y., Jeng, D., S., Liu, P., L., & Zhang, C., (2014). Numerical Simulation of wave-current interaction using a RANS solver. *Ocean Engineering*, 157-164.

[6] Christine. W., Pal. S., John. R., Josh. D., (2018). Development of an impulse source-based wave-current interaction (WCI) model.
[7] Anton, K., Olga, S., & Micheal, M., (2014). Numerical Simulation of Droplet impingement and flow around a cylinder using RANS and LES Models. *Journal of fluids and structures.*
[8] Silva, M. C., Vitola, M. A., Esperanca, P.T.T., Sphaier, S.H., Levi, C.A., (2016). Numerical simulations of wave-current flow in an ocean basin. Journal of Applied Ocean Research, vol. 61, pp. 32-41.
[9] Catalano, P., Wang, M., Iaccarino, G., & Moin, P., (2003). Numerical simulation of the flow around a circular cylinder at high Reynold's Number. *International journal of heat and fluid flows* 463-469.
[10] Hans.B., Ankit. A., Shrinov. S., & Dag. M., (2019). Estimation of breaking wave properties and their interaction with a jacket structure.
[11] Sonia, B., H., Sabra, H., & Said, N., M., (2015). Numerical Simulation of wave-structure interaction around an obstacle.
[12] Clauss, G.F., Schmittner, C.E., Stuck, R., (2005). Numerical wave tank Simulation for the investigation of structural responses. 24th Int. conf. offs. Mech. & Arc. Eng. OMAE 2005-67048.
[13] Kurian, V. J., Gasim, M. A., & Narayanan, S. P. (2009). Frequency and Time Domain Analyses for TLP Responses. *7th Asia Pacific Structural Engineering & Construction Conference 2009*, (pp. 555-560). Pulau Langkawi.
[14] Kurian, V. J., Wong, B. S., & Montasir, O. A. (2008). Frequency Domain Analysis of Truss Spar Platform. *International Conference of Construction and Building Technology*, (pp. 235244). Kuala Lumpur.
[15] Roitman, N., Andrade, R. F., & Batista, R. C. (1992). Dynamic Responses Analysis of a small-Scale Model Tension Leg Platform. *Marine Structures*, 491-513.
[16] Isaacson, M., & Cheung, K. F. (1993). Time Domain Solution for Wave-Current Interactions with a Two Dimensional Body. *Applied Ocean Research*, 39-52.
[17] Anam, I., Roesset, J. M., & Niedzwecki, J. M. (2003). Time Domain and Frequency Domain Analysis of Spar Platform. *The Thirteenth International Offshore and Polar Engineering Conference*, (pp. 240-247). Honolulu, Hawaii, USA.
[18] Montasir, O. A., Kurian, V. J., & Shafiq, N. (2009). Numerical and Model Test Results for Truss Spar Platform Subjected to Random Waves. *2nd International Conference on Engineering Technology*, (pp.1-6). Kuala Lumpur.
[19] Lin, M. C., & Hsiao, S. S. (1994). Boundary Element Analysis of Wave-Current Interaction Around A Large Structure. *Engineering Analysis with Boundary Elements*, 14, 325-334.
[20] Ng, C.Y., Sazali, M.S., (2015) Dynamic Response of Truss Spar: A comparison of a long-crested and short-crested wave with the current. Conf. paper.
[21] Ng, C.Y., Kurian, V.J., Tuhaijan, S.N.A, (2016) The Effect of current on the Dynamic Responses of Truss Spar in Malaysia Water. 35th Int. conf. Offs. & Arc. Eng. OMAE2016-54399.
[22] Prsic, M.A., Ong, M.C., Pettersen, B., Myrhaug, D., (2014). Large-eddy simulations of the flow around a smooth circular cylinder in a uniform current in the subcritical flow regime. Ocean Eng. 77, 61-73.
[23] Kemp, P. H., and Simons, R. R. (1982). The interaction between waves and a turbulent current: waves propagating with the current, Journal of Fluid Mechanics, vol. 116, pp. 227-250.
[24] Markus, D., Hojjat, M., Wuchner, R., Bletzinger, K. U. (2013). A CFD approach to modelling wave-current interaction, International journal of offshore and polar engineering.
[25] Markus, D., Hojjat, M., Wuchner, R., Bletzinger, K. U. (2012). A numerical wave channel for the design of offshore structures with consideration of wave-current interaction, International offshore and polar engineering conference.
[26] Fenton, J. D. (1990). Nonlinear wave theories, journal of Ocean engineering science.
[27] ANSYS Inc. (2017, January). ANSYS Fluent Tutorial Guide R 18.0. Canonsburg.
[28] ANSYS Inc. (2014, November). ANSYS Fluent Theory Guide R 15.0. Canonsburg.

[29] ANSYS Inc. (2013, November). ANSYS Fluent User Guide R 15.0. Canonsburg.
[30] Rosebud, J. L., (2012). Development of a numerical wave tank using OpenFOAM (Masters Dissertation). Universidade de coimbra, Coimbra, Janeiro.
[31] Salwa, T., Bokhove, O., & Kelmanson, M. A. (2017). Variational modelling of wave–structure interactions with an offshore wind-turbine mast. Journal of engineering mathematics, 107(1), 61-85.
[32] Dagli, B. Y., Yigiy, M. E., Gokkus, U., (2017). The behaviour of Large Cylindrical Offshore Structures Subjected to Wave Loads. TEM Journal, Vol 6.
[33] Paci, A., Gaeta, M. G., Antonini, A., & Archetti, R. (2016). 3D-Numerical Analysis of Wave-Floating Structure Interaction with OpenFOAM. Paper presented at the 26th International Ocean and Polar Engineering Conference.
[34] Zarruk, G., Cowen, E., Wu, T.-R., & Liu, P.-F. (2015). Vortex shedding and evolution induced by a solitary wave propagating over a submerged cylindrical structure. Journal of Fluids and Structures, 52, 181-198.
[35] Chen, L., Zang, J., Hillis, A. J., Morgan, G. C., & Plummer, A. R. (2014). Numerical investigation of wave–structure interaction using OpenFOAM. Ocean Engineering, 88, 91-109.
[36] Chiva, J., Rodriguez, I., & Oliva, A. (2015). On the flow past a circular cylinder from critical to super-critical Reynolds numbers: wake topology and vortex shedding. International journal of heat and fluid flow, 2015, vol. 55, p. 91-103.
[37] Palemón-Arcos, L., Torres-Freyermuth, A., Chang, K.-A., Pastrana-Maldonado, D., & Salles, P. (2014). Modeling wave-structure interaction and its implications in offshore structure stability. Paper presented at the Twenty-fourth International Ocean and Polar Engineering Conference.