Oscillatory loads on circular cylinder with secondary structures

Petersen, Thor Ugelvig; Mandviwalla, Xerxes; Christensen, Erik Damgaard; Tarp-Johansen, Niels Jacob; Rüdinger, Finn

Published in:
Journal of Fluids and Structures

Link to article, DOI:
[10.1016/j.jfluidstructs.2020.102935](https://doi.org/10.1016/j.jfluidstructs.2020.102935)

Publication date:
2020

Document Version
Peer reviewed version

[Link back to DTU Orbit](#)

Citation (APA):
Petersen, T. U., Mandviwalla, X., Christensen, E. D., Tarp-Johansen, N. J., & Rüdinger, F. (2020). Oscillatory loads on circular cylinder with secondary structures. *Journal of Fluids and Structures*, 94, [102935]. <https://doi.org/10.1016/j.jfluidstructs.2020.102935>

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal

If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.

Oscillatory loads on circular cylinder with secondary structures.

Thor Ugelvig Petersen¹, Xerxes Mandviwalla¹, Erik Damgaard Christensen², Niels Jacob Tarp-Johansen³, and Finn Rüdinger³

1) DHI, Port and Offshore Department, Agern Alle', DK-2970 Hørsholm, Denmark.

2) Technical University of Denmark, DTU-MEK, Nils Koppels Alle', Building 403, DK-2800 Kgs. Lyngby, Denmark

3) Ørsted, Wind Power, Nesa Allé 1, DK-2820 Gentofte, Gentofte.

Abstract

This study deals with a numerical and experimental study of the secondary structures effect on the load on the main circular cylinder. Its objective is to address the increasing amount of offshore structures installed as part of offshore wind farms. The typical support structure for a wind turbine is a monopile foundation. Boat-landings are attached to the monopile and these structures act as secondary structures. Forces on a cylinder with and without secondary structures attached in unsteady flows have been measured experimentally and simulated numerically. The results of the tests were compared to each other and to existing work in the present report for validation of the CFD model. The secondary structures were represented by circular cylinders placed at different orientations around the main circular cylinder. All the analyses assumed uniform conditions transverse to the flow direction (i.e. 2D set-up). The numerical simulations was based on the OpenFOAM CFD tools. The validation tests with a circular cylinder without secondary structures showed good agreement between the physical and numerical results, as well as to the results available in the open literature. The orientation of the secondary structures had a clear influence on the drag related part of the force as the drag coefficient C_D when the secondary structure was orientated towards $\theta = 90^\circ$, i.e. perpendicular to the flow direction. The inertia part of the force, i.e. the inertia force coefficient C_M , was only affected by the orientation to small extend.

1 Introduction

For the last couple of decades European oceans have been subject to massive development of offshore wind farms. The development is foreseen to be supplemented by a more general implementation of marine infrastructure in the near future, see. (Christensen et al., 2015). The development includes energy facilities, e.g. offshore wind farms, exploitation of wave energy, and development and implementation of marine aquaculture.

Support structures for offshore wind farms are often monopiles. Other support structures used have been jacket and gravity based structures. Boat-landings are attached to the support structure, e.g. a monopile, and these structures act as secondary structures. Several studies have investigated the total load on wind turbine support structures. (Christensen et al., 2005) studied wave forces and wave run-up on a vertical, cylinder in non-breaking and breaking waves with computational fluid dynamics (CFD). (Bredmose et al., 2006) studied the wave loads on a gravity based foundation with CFD and compared this with experimental results. (Christensen et al., 2011) studied irregular breaking wave loads on a gravity based structure. (Paulsen et al., 2014) studied the forces from steep waves on a vertical cylinder with CFD. Several other studies have also studied the forces on the large main structure. The literature is weak on the studies of forces on secondary structures, but some literature exists on wave forces on for instance a service platform in (Damsgaard et al., 2007) and (Bredmose and Jacobsen, 2011). Studies on and the effect on the main structure of other secondary structures, such as boat-landings, are scarce. However, the boat-landings are an important structural element as they provide access to the offshore wind turbine from the seaside. In the following we address some fundamental studies that is related to the analyses presented in this paper.

The forces on a circular cylinder have been intensively investigated. A general introduction to hydrodynamics around and forces on cylindrical structures are given in (Sumer and Fredsøe, 2006). (Sarpkaya, 1976), and (Sarpkaya, 1986) experimentally analysed the forces on a circular cylinder in oscillatory flow generated in a u-tube. The experiments that included smooth as well as sand-roughed cylinder surfaces showed some general trends; a minima of the inertia coefficient C_M and a maxima of the drag coefficient C_D for the Keulegan-Carpenter number (KC) around 10 -15. Increasing

roughness results in larger drag coefficients and smaller inertia coefficients for higher Reynolds numbers. (Sarpkaya, 1977) extended the analyses to include a circular cylinder near a wall. Other examples of research on oscillatory loads on a circular cylinder are (Bearman et al., 1986) who studied theoretical and experimentally the oscillatory load on a circular cylinder, (Bearman et al., 1984) set up a model equation for the transverse forces in oscillatory flow, (Justesen, 1989) studied oscillatory loads using an oscillating carriage instead of an oscillating fluid and (Justesen, 1991) carried out a numerical study of the flow around a cylinder in oscillatory flow that included estimation of force coefficients. These studies are in line with the studies of (Sarpkaya, 1976), and (Sarpkaya, 1986), and are extensions and supplementary investigations of oscillatory loads with different experimental set-up, numerical methodologies, and extension of the parameter space.

There exists other studies that examined the forces on different configurations on smaller and larger cylinders. In connection with pipelines an often used layout is the piggy back layout of a smaller pipe line on top of a larger pipeline. For instance a piggy-back line system comprise a primary line employed for export of the hydrocarbon flow away from a field with a secondary smaller diameter flow line carrying injection water, injection chemicals etc. to the field. (Kalghatgi and Sayer, 1997) studied forces on piggy-back configurations in steady incident flows. (Cheng et al., 2012c) studied the forces on a large and small cylinder in piggy-back position near a seabed experimentally. Experiments were carried out in a wave flume and both regular waves as well as irregular waves were used in the investigations. In all cases the small cylinder was on top of the large cylinder. (Zhao et al., 2007) used numerical modelling to study the forces on a piggy-back system in steady current. (Cheng et al., 2012a) and (Cheng et al., 2012b) studied the wave forces on a piggy-back system, where they used a VOF-method (Volume of Fluid) to track the free surface.

Several authors have studied tandem arrangements of two circular cylinders, such as (Alam et al., 2005). Other studies of open structures is for instance jacket structures modelled as a porous medium, (Santo et al., 2018a), (Santo et al., 2018b), and fish cages (Chen and Christensen, 2016).

In this study we focused on modelling the secondary structures directly in the experimental and numerical modelling. The main objective of our study was to assess

the effect of the secondary structures on the total load on the support structure. The motivation behind the study was to determine with improved accuracy the forces from wave interaction with the wind turbine foundations and the secondary structures. The future use is expected to be exploited with engineering tools that estimates forces with the Morison force model rather than a direct simulation of the forces in CFD. The methodology was based on a combination of physical experiments supplemented with CFD-calculations. The CFD-calculations should support the findings of the physical experiments and to clarify some uncertainties of the experiments. This involved assessing the possible scale effects.

The outline of the paper is as follows. First we describe the experimental set-up in section 2. This includes the set-up of the flume, carriage and the physical models that were tested, under different oscillating flow condition. The section also describes how basic analyses were carried out of for instance the method to assess the force coefficients that was the same in the physical experiment and the numerical calculations. The section on the experiments is followed with section 3 that describes the numerical method that was based on the OpenFOAM CFD library, (“OpenFOAM, The open source CFD toolbox,” 2018). In this section special attention is given to the description of the oscillating body force that modelled the effect of an oscillating flow. Section 4 presents a comparison between the experimental and numerical results to well-known reference works in the literature for a single circular cylinder. Section 5 presents the results for a cylinder with secondary structures. The focus was on the orientation of the secondary structure and the influence of the KC-number. Finally, section 6 summarises and concludes the work.

2 Experimental set-up and test conditions

The experiments were carried out in the towing tank at the Technical University of Denmark (DTU). The measurement section of the tank was 35m long, 3.0m wide and 1.0m high. A carriage running on rails on top of the flume can operate at up to about 1.5m/s back and forth in an otherwise calm water. The flume is sketched in Figure 1.

Figure 1 Schematic description of the test flume.

The relatively large size of the experimental facility was important as the side walls of the experimental facility induce a confining effect on the model, which normally is termed *blockage*. Generally, one may assume that the blockage effect is small and may be ignored when the ratio of the cylinder diameter to the width of the experimental facility, D/B , is less than 0.1. In the present situation, blockage effects were undesirable and the value was maintained so $D/B \leq 0.1$.

2.1 Test conditions

The study of the effect of secondary structures on the hydrodynamic forces was related to two common designs in offshore wind engineering. A very common support structure is the monopile, which also include as boat-landing for assessing the wind turbine for inspection purposes. Another common support structure is the jacket structure. A jacket structures can be used for support structures for wind turbines, but is also often used for supporting facilities such as transformer stations, and accommodation. The boat-landing is in these cases typically attached to just one of the

jacket legs. On this background two typical two-dimensional cross-sections for support structures with secondary structures were established.

The monopile with secondary structures consisted of a monopile with a diameter of 6.0 metres, and four smaller circular cylinders, where the two smallest had a diameter of 0.177 m and the remaining two a diameter of 0.419 m. The smaller diameter cylinders represent ladders and the larger boat landings. The jacket leg had a diameter of 1.80 m, and the secondary structure were represented by 6 smaller cylinders, where three had the diameter 0.146 m and the other three had a diameter of 0.386m. All measures relate to full scale. Figure 2 shows a sketch of the two cross-sections. For the monopile the total cross-sectional area, including the secondary structures, was $A_{tot} = 28.6 \text{ m}^2$, which was approximately 1 % more the main structure. For the jacket structure the total area of the cross section was $A_{tot} = 2.935 \text{ m}^2$, which was approximately 15 % of the main jacket leg alone.

Figure 2 The two layouts, Left: Monopile with secondary structures, Right: Jacket leg with secondary structures.

Table 1 Full scale data used in the tests. The data reflect typical dimensions on wind turbine support structures.

Property	Variable name / Unit	Monopile	Jacket leg
		Full scale	Full scale
Diameter	D ₁ (m)	6	1.8
Boat landing	D ₂ (m)	0.419	0.386
Ladder	D ₃ (m)	0.177	0.146
Wave period	T (s)	15	15
Amplitude	a (m)	12.5, 10, 7.5, 5.0, 2.5	12.5, 10, 7.5, 5.0, 2.5
Angular frequency	$\omega = 2\pi/T$ (rad/s)	0.42	0.42

2.2 Model and Model Scale

The flow around the monopile and the jacket leg lies within the classic topic of hydrodynamics; flow around a circular cylinder in waves. The flow around a circular cylinder in waves exerts a resultant in-line force (drag and inertia) and cross-flow force (lift) on the cylinder; a thorough account on the topic is provided in (Justesen, 1991). Cylinder diameter and surface roughness both influence the resultant force as well as the velocity, period and amplitude of the wave motion. Generally, the effect of cylinder diameter, D , maximum horizontal velocity, U_m , and the amplitude of the wave motion, a , is expressed in terms of two non-dimensional numbers: the Reynolds number, $Re = DU_m/\nu$, and the Keulegan-Carpenter number, $KC = U_m T/D$. Surface roughness is expressed in terms of the relative roughness, k_s/D , where k_s is the equivalent sand roughness (or k/D where k is the roughness height).

A vertical cylinder with secondary structures under exposure to waves experiences alterations in the drag and the lift force compared to a single plain cylinder. These effects are attributed to:

1. The deviation in the deflecting flow lines around the cylinder with the secondary structures attached
2. The change of the plan view separation point of the lee-wake vortices

3. The increase in the hydrodynamic field enforced by the secondary cylinders. Obviously, the plan view orientation, θ as shown in Figure 3 of the secondary structures relative to the KC-number and the Reynolds number will be the governing parameters.

Due to mutually exclusive scaling laws, dynamic similarity between physical model experiments and full scale is only attainable at full scale. The physical model experiments were scaled using Froude scaling. Froude scaling together with geometrical similarity allowed reproduction of the Keulegan-Carpenter number, KC, orientation of the secondary structure, θ , and the relative roughness, k_s/D , in the model. The orientation of the secondary structure, θ , is defined in Figure 3

Henvisningskilde ikke fundet.. This, however, meant that the Reynolds number was not scaled correctly in the physical model experiments. The consequences of this, however, were limited because of the surface roughness and the application of trip-wires inside the developing boundary layer around the cylinder, which will be detailed later.

Figure 3 Definition sketch (schematic) of the monopile set-up with the orientation of the secondary structures relative to the direction of oscillatory motion defined by θ .

2.2.1 Model scale

The physical model experiments were scaled according to Froude number scaling:

$$Fr_{full-scale} = Fr_{model}$$

$$\left(\frac{u}{\sqrt{gh}} \right)_{full-scale} = \left(\frac{u}{\sqrt{gh}} \right)_{model} \quad (1)$$

in which u is the velocity and h the water depth. Table 2 gives factors for scaling of full scale to model scale in the monopile and the jacket leg set-up. The monopile structure was tested at a scale of 1:21.4 and the jacket leg at scale 1: 12.9.

Table 2 Factors for scaling of full scale to model scale. Monopile with $\lambda = 21.4$ and jacket leg with $\lambda = 12.9$

Parameter	Scale law	Scale factor for monopile ($\lambda = 21.4$)	Scale factor jacket leg ($\lambda = 12.9$)
Length	λ	21.4	12.9
Area	λ^2	458	166
Volume	λ^3	9800	2147
Time	$\lambda^{1/2}$	4.62	3.59
Velocity	$\lambda^{1/2}$	4.62	3.59
Mass	λ^3	9800	2147
Force	λ^3	9800	2147
Moment	λ^4	209,727	27,692
Pressure	λ	21.4	12.9

The diameter and length of the measurement sections for the monopile and the jacket leg were:

Monopile: $D_{model} = 0.280 \text{ m}$, $L = 0.300 \text{ m}$

Jacket leg: $D_{model} = 0.140 \text{ m}$, $L = 0.300 \text{ m}$

2.2.2 Submerged cylinder with and without secondary structures

Figure 4 shows the submerged circular cylinder with and without secondary structures attached. The cylinder was suspended from an adjustable support frame, which allowed the vertical position of the cylinder in the water column to be adjusted. A close-up of the support frame is shown in Figure 5. The support frame consisted of two box profiles (placed on top of the carriage), four M20 threaded steel rods and two 200mm x 400mm x 15mm steel plates assembled with nuts and washers. The set-up of the support frame, cantilever block and the connection to the steel liner, shown in Figure 5, enabled rotation of the model relative to wave direction by simply rotating the steel liner around the cantilever block.

Figure 4 (a) Monopile without secondary structures (b) Monopile with secondary structures.

Figure 5 Left) Support frame cantilevering the monopile model. (Right) Close-up of connection between support frame and the suspended monopile model

The models were split into three parts Figure 4. The middle part was the measurement section, while the bottom and top parts were dummy sections. The dummy sections were rigidly fixed to the inner steel liner, while the measuring section was suspended in two (2-component) force gauges capable of measuring the two horizontal forces, in-line and lift (F_x , F_y). The force gauges were attached asymmetrically to the inside of the steel liner. One force gauge at the top (between dummy and measuring sections) and one force gauge at the bottom of the measuring section. One end of each of the two force gauges was rigidly fixed to the steel liner, while the other end of the force gauges held the measurement section. A small gap

between the outer PVC cylinder and the inner steel liner allowed for slight movement of the measuring section relative to the steel liner so that the displacement could be measured by the strain gauges on the force gauges.

The measurement section was mounted on two 2-component force gauges. The two force gauges of type DHI-205/150 (jacket leg set-up) and DHI-205/150 (and DHI-205/100) (monopile set-up), were calibrated independently and checked after assembly of the model. They were connected to strain amplifiers of type DHI-106E. Figure 6 illustrates the load cells attachment to the physical model.

Figure 6 Sketch of arrangement of the load cells.

Figure 7 sketches how the physical model attached to the carriage. The flow around an object placed in a channel with a free surface induces a local set-up and set-down of the water table due to the pressure variations around the model. These disturbances are increased fast with increasing velocities. There seems to be two effective ways of damping the effect of this set-down: (1) to place a plate/screen below the surface shielding the model against the set-down (i.e. the screen shall be

placed beneath the set-down level of the surface); or (2) place the measurement section sufficiently low beneath the set-down level for the free surface effects not to interfere with the flow around the measurement section.

In the present case, the second option was chosen, where the top of the measurement section was below 0.23 m below the free water surface. The maximum set-down effects were observed to penetrate the free surface water level down to about 7 cm.

Besides the direct surface deformation, a wave field is induced to the rear/downstream of the model with the same characteristics as the wave patterns around a ship hull. The wave field can give rise to additional forces on the models, but due to the rather short wavelengths, the force is concentrated at and just below the surface.

The effects of surface deformations and wake waves are in spite of their magnitude in the high-speed tests considered low in the present experiments mainly due to the distance from the free surface level to the top of the measurement section.

Figure 7 Illustration of the model attached to the carriage. The measuring section was 30 cm high. The total water depth was 0.75 m. The top of the measuring section was 0.23 m below the free water surface.

2.2.3 Trip wires

For the selected range of the model tests, the Re numbers were in the critical, super-critical or post-critical range. To ensure the transition from a laminar to a turbulent boundary layer around the main cylinder, trip wires were applied, as previously mentioned and visible in Figure 4. The trip wires ensured an external

generation of turbulence in the developing boundary layer around the surface of the main cylinder, and in turn this should render a measured force response similar to the post-critical range of the Re-number, even though at model scale the Re-number corresponded to the critical or super-critical range. The use of trip wires is a well known technique, see for instance (James and Truong, 1972), (Pearcey et al., 1982), (Igarashi, 1986) and (Hover et al., 2001).

The trip wires were placed at a plan view angle $\theta = \pm 45^\circ$ to the oscillating direction (x-direction) on both sides of the main cylinder. This plan view position of the trip wires was maintained as the cylinder with the secondary structures was re-orientated. No trip wires were applied in the case where the location of the secondary structures was in conflict with placing the trip wires (i.e. directly on top of each other); in these cases the added hydrodynamic field enforced by the secondary structures was assumed to make up for the missing trip wires. The trip wires applied to the models had a diameter of 1.6 mm and were located with a plan view radial distance of 2-3 mm from the surface of the main cylinder for the most part of the test programme, while some specific tests were run with smaller diameter trip wires or the trip wires were removed. In both cases, the purpose was to measure the effect of the trip wires on the vortex shedding and in turn on the recorded in-line forces.

The CFD calculations used a full-scale set-up. This include fully rough boundary conditions for the for instance the turbulence model. Therefore, no trip wires were used in the CFD model that is described in detail later. In this way the CFD model tests the assumption of the use of trip wires. Using a combination of physical model tests at model scale and CFD modelling at full scale has previous been used to study complex flow phenomenon. See for instance (Jensen et al., 2018).

2.2.4 Data acquisition

Data acquisition was performed using the DHI Wave Synthesizer (WS) data acquisition system. The measurements were logged and data stored at a sampling frequency of 160 Hz at model scale. The measurements contained the time series of force records (F_x , F_y), displacement and velocity of carriage. The carriage position, x , along the length of the flume was measured with a SICK DT50 (P2113) laser distance measuring device. The measurement range was 0.2 ... 20m, with a resolution less than 1

mm, and with an accuracy of ± 1 mm with our use. For longer distances and less remission the accuracy is less, e.g. ± 7 mm. The post-processing will typically narrow the accuracy further as the accuracy above refers to the instantaneous accuracy. During post-processing, the position signal was low-pass filtered with a cut-off frequency of 1 Hz and differentiated with time to give the velocity of the carriage, $u = dx/dt$, and the acceleration du/dt .

The subsequent analysis of the time series of load measurements was post-processed and analysed through the following procedure:

- The time series of load measurements were filtered by a once-through band pass filter removing network noise at a low pass filter level of 10 Hz. This is similar to previous research that used the carriage system, e.g. (Jensen et al., 2018), (Bryndum et al., 1992).
- The loads records of the individual wave periods of oscillations were assembled to give the mean force record, mean velocity and acceleration over the period of oscillatory motion.

2.2.5 Accuracy of the tests

The uncertainty of the physical model test was assessed where the effects of free surface, blockage effect, secondary motions from for instance horse shoe vortices', correlation length of vortex shedding, and workmanship of the physical model were considered. Some of these quantities have also been discussed elsewhere. All of the above-mentioned phenomena contribute to an inaccuracy of the test results. The below list gives possible sources of inaccuracies with a best estimate of their magnitude:

1. Workmanship of model 1%
2. Blockage effects 0.5%
3. The force measuring system 0.5%
4. Free surface effects low
5. Correlation length KC number dependent
6. Re-number scaling (trip-wires) Re number dependent
7. Secondary motions 5.0%
8. Velocity of travelling carriage 0.3%

The summation of the inaccuracies listed above to a total inaccuracy for the experimental results of the force measurements are estimated based on previous experience with similar model set-ups to be 5-8%. Note that this estimate does not consider the correlation length and the lacking Re number scaling.

2.3 Data analyses

For a cylinder accelerated and moved in still water, the Morison formulation for the total in-line force reads:

$$F_{inline} = \frac{1}{2} \rho C_D D h U |U| + \rho C_M A h \frac{dU}{dt} \quad (2)$$

The first term on the right-hand side is the drag force and the second term is the hydrodynamic mass force (inertia force), ρ is the water density, h is the cylinder height, D is the cylinder diameter, A is the cross-sectional area, U is the undisturbed velocity relative to the centre line of the position of structure but without structure in place. The velocity U could in our case is also the velocity far away from the structure relative to the position of the structure. C_D is the drag coefficient and C_M is the hydrodynamic mass coefficient.

The drag coefficient, C_D , and the hydrodynamic mass coefficient, C_M , were determined for each of the experimental and numerical tests. The technique used for determining the force coefficients was the method of least squares fit, see for instance (Sumer and Fredsøe, 2006). In all cases the diameter and cross-sectional area of the main structure was used. For instance for the monopile with secondary structures $D = 6.0 \text{ m}$ and $A = 28.27 \text{ m}^2$ eqs. (2). The total frontal area increases as the position of the secondary structure is turned perpendicular to the flow direction. This effect was not accounted for by adjusting the diameter and cross sectional area in the analyses. Instead the effect will be seen in modified force coefficients.

The measurement section consisted of a cylinder of PVC, fixation units of aluminium, force gauges and screw bolts as mentioned previously. Further to this, the compartments within the model (between PVC shell and steel liner; and inside steel liner) were filled with water during testing. Therefore, the measured inertia force included a contribution from the entire mass of the measurement system and the water inside.

The procedure to determine the contribution of the mass of the measurement section and the water inside is described in the following. Firstly, the assembled models were tested in the flume without water. Here the model was oscillated to achieve $KC \approx 1$, and the measured maximum force response comprised only the accelerated mass of the measurement section. It should be mentioned that there was a good agreement between the measured force response and the calculated force response in this procedure. Secondly, the model was submerged in still water and oscillated at the same test conditions (i.e. $KC \approx 1$) where $C_m = 1$. Hence, the measured inertia force could be described by:

$$\rho C'_m Ah \frac{dU}{dt} = \rho C_m Ah \frac{dU}{dt} + \rho n Ah \frac{dU}{dt} \quad (3)$$

$$C'_m = C_m + n$$

where the last term, n , on the right-hand side represents the mass of the measurement section and the water inside (not to be confused with the Froude-Krylov force). The hydrodynamic force coefficient, C_m was therefore determined from the actual measured inertia coefficient, C'_m , using the above relationship: $C_m = C'_m - n = 1$ at the given flow conditions ($KC \approx 1$). n remained constant for a specific model setup with changing wave flow climates (with specific weight of model and water inside), and this procedure was remade for each model setup tested in the present experimental campaign. In the present cases, n assumed values $O(0.9 - 1.1)$ and subtraction was made in the measured force time series accordingly.

Lastly, to obtain the inertia coefficient C_M :

$$C_M = C_m + 1 \quad (4)$$

Note that the present treatment of the force coefficients only considered the main pile geometry, and therefore the effect of the secondary cylinders were inherent in the force coefficients, drag and added mass, obtained through the present analysis. For the practical application of the available force coefficients one should re-evaluate the Froude-Krylov effect of the secondary structures relative to the main pile geometry.

In the case with secondary structures to the main cylinder, the mass of the secondary structures times the acceleration were also removed from the measured force response.

The focus in the study was on the inline forces, but as a supplement the transverse force (or lift force in the following) was addressed in the analyses of the single circular cylinder for comparison with literature. The maximum value of the lift force was considered for both the physical and CFD models.

$$F_{L,\max} = \frac{1}{2} \rho C_{L,\max} D h U_m^2 \quad (5)$$

3 Numerical Methodology

The model meshing and numerical calculations were performed using the C++ CFD toolbox OpenFOAM® (OF) (“OpenFOAM, The open source CFD toolbox,” 2018). OF is open source and freely distributed, thus allowing and enabling researchers to modify the standard code to suit the needs of a specific project or task.

3.1 The numerical model

Eqs. (6) and (7) are the incompressible Unsteady Reynolds averaged Navier-Stokes equations (URANS) with the body force term:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (6)$$

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\nu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \overline{u_i' u_j'} \right] + F_{Bi} \quad (7)$$

where $\bar{u}_i$, are the time-averaged velocities (phase-averaged for oscillatory flow), p is the pressure, ν is the kinematic viscosity of water, ρ is the density of water, x_i are Cartesian coordinates and t is time.

The body force term in eqs. (8) to model the oscillating wave is denoted by F_{Bi} and is set according to eqs. (8) below:

$$\begin{aligned} F_{B_1} &= \omega U_m \cos(\omega t + \varphi) \\ F_{B_2} &= 0 \end{aligned} \quad (8)$$

where ω is the angular wave frequency, U_m is the maximum wave velocity in the free stream, and φ is a phase shift. F_{B_1} and F_{B_2} are the components of the body force in the x and y directions, respectively.

Alternatively to body force terms you could specify the flow velocity at the inlet. This would create a pressure gradient as for instance under waves. This pressure gradient will influence on the cylinder as the Froude-Krylov force does under waves. We used the body force term and therefor did not include a pressure gradient from the ambient water. The inertia coefficient had to be compensated for that when the results is used in waves, just as we have to compensate the inertia coefficients estimated from our physical model tests with a moving carriage.

Model scenarios, where the angle of the incident was tested, were modelled by rotating the mesh geometry as presented in the next section. The body force remained fixed in the horizontal direction.

The Reynolds stress tensor term, $-\overline{u_i' u_j'}$, in (7) is equated to the Boussinesq eddy viscosity assumption as follows in eqs. (9):

$$\overline{u_i' u_j'} = \nu_T \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} k_T \delta_{ij} \quad (9)$$

where ν_T is the turbulent eddy viscosity, k_T is the turbulent kinetic energy and δ_{ij} is the Kronecker delta.

In order to close the system of equations, the eddy viscosity term was modelled by solving the two-equation k-Omega SST (Shear Stress transport) turbulence model. This turbulence model was selected for its merit in modelling flows where adverse pressure gradients and flow separation are important. It is a blended model which utilises the strength of the k-epsilon model in the outer regions of free shear flow and the k-omega model in the near wall regions (Menter, 1994).

$$\rho \frac{\partial k}{\partial t} + \frac{\partial \rho k \bar{u}_i}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \mu_t / \sigma_k \right) \frac{\partial k}{\partial x_j} \right] + \tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j} - \rho \beta^* k \omega \quad (10)$$

$$\begin{aligned} \rho \frac{\partial \omega}{\partial t} + \frac{\partial \rho \omega \bar{u}_i}{\partial x_j} = & \frac{\partial}{\partial x_j} \left[\left(\mu + \mu_t / \sigma_\omega \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\gamma}{\nu_t} \tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j} \\ & - \beta \rho \omega^2 + 2(1 - F_1) \frac{\rho \sigma_{\omega 2}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \end{aligned} \quad (11)$$

The eddy viscosity is therefore defined as follows in eqs. (12).

$$\nu_T = \frac{a_1 k}{\max(a_1 \omega, SF_1)} \quad (12)$$

where a_1 is a constant, ω_T is the turbulent dissipation, S is the strain and F_2 is a blending function which is set to 1 in the near wall and 0 in the free shear region.

The blending function F_I is a function of the distance to the wall:

$$F_I = \tanh(\arg_1^4)$$

where

$$\arg_1 = \min \left[\max \left(2 \frac{\sqrt{k}}{0.09 \omega z}; \frac{500 \mu}{z^2 \omega} \right); \frac{4 \rho k}{(CD_{k\omega}) z^2 \sigma_{\omega 2}} \right] \quad (13)$$

with

$$(CD_{k\omega}) = \max \left(2 \rho \frac{1}{\sigma_{\omega 2} \omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}; 10^{-20} \right)$$

Table 3 Model constants for inner ‘wall’ region

β_1	β^*	γ_1	σ_{k1}	$\sigma_{\omega 1}$
.075	0.09	0.553	2.0	2.0

Table 4 Model constants for outer ‘surface’ region

β_2	β^*	γ_2	σ_{k2}	$\sigma_{\omega 2}$
0.0828	0.09	0.4404	1.0	1.17

3.2 Boundary Conditions

The following section outlines the boundary conditions used in the model.

3.2.1 Pressure

For the cylinder walls, a normal zero gradient ($\frac{\partial p}{\partial n} = 0$) was applied. For the

domain boundaries, a total pressure condition was imposed ($p + \frac{1}{2} \rho |\vec{u}|^2 = 0$) allowing

for the pressure, p , to be computed from the velocity.

3.2.2 Velocity

No-slip zero velocity condition was applied to the cylinder walls ($u=0$), while for

the domain boundaries, a normal zero gradient ($\frac{\partial u}{\partial n} = 0$) was applied.

3.2.3 k , ω and ν_t

At the cylinder walls, zero gradient was applied for the turbulent kinetic energy ($\frac{\partial k}{\partial n} = 0$) and for the specific turbulent dissipation, ($\omega = \frac{U_f}{\sqrt{\beta^*} \kappa y_p}$) was applied at the wall cells. The surface roughness of the cylinder walls was handled with eddy viscosity term ($\nu_t = \frac{\sqrt{\beta^*} \sqrt{k} y_p}{\log(Ey^+)}$) which was derived from the wall function ($\frac{u}{u_f} = \frac{1}{\kappa} \log(Ey^+)$).

where $E = 9.8$ for a smooth wall or $E = \frac{9.8}{\left[\frac{k_s^+ - 2.25}{87.75} + C_s k_s^+ \right]^{\sin(0.4258 \cdot \log(k_s^+) - 0.811)}}$ when

$2.25 < K_s^+ < 90$ transitional roughness, or $E = 9.8 / (1 + C_s k_s^+)$ for $k_s^+ > 90$ fully rough wall. Parameter definitions are as follows; $k_s^+ = k_s U_f / \nu$ is the non-dimensional roughness, C_μ turbulence model constant, y_p distance to first calculation point from wall, C_s roughness constant depending on the type of roughness, (set to 0.5 here) and y^+ non-dimensional wall distance.

3.3 Solution method

The solution method is based on the open source library OpenFOAM (Open Field Operation and Manipulation), (“OpenFOAM, The open source CFD toolbox,” 2018). This is an open source, object-orientated C++ library for numerical simulations of fluid dynamics.

The numerical model applied was a modified version of the standard transient, incompressible PIMPLE algorithm that is a combination of the PISO-algorithm, (Issa, 1986) and the SIMPLE-algorithm (Caretto et al., 1973) as part of the solver pimpleFoam in OpenFOAM, which includes an oscillating body force term representative of the wave flow field. This body force term was used to model periodic oscillations. The numerical integration in time was based on the PIMPLE-algorithm using an Euler backward time integration. The Gauss linear Upwind (LUD) resolved the convective terms, while central differences was used for diffusive terms. This leads to a spatial resolution close to second order. The pressure equation that can be derived from

the continuity equation was solved using Generalised geometric/algebraic multi-grid solver (GAMG), while the momentum equations and the turbulence equations were solved using preconditioned bi-conjugate gradient solver (PBICG). As preconditioner the incomplete LU (DILU) was used for the momentum and turbulence equations.

3.4 Meshing strategy

Model geometries for the jacket and monopile foundations with secondary boat landing structures were created as presented in the previous sections. Rotated geometries to model the effect of varying wave incidences (typically in 30° intervals) were also created, along with jacket and monopile structures without secondary structures.

Model meshes were created using the OpenFOAM automated meshing library snappyHexMesh (sHM). This produced 2D meshes with a majority of hexahedral elements. Boundary layer elements were added in order to apply rough wall boundary conditions. The panel a) in Figure 8 presents the overall mesh for the monopile foundation with secondary structures rotated at 60° , while b) and c) are close-ups of the cylindrical structures.

Figure 8 Mesh for 6 m diameter monopile with secondary structures at 60° rotation, $ks=0.0045$ m. a) View of full domain. b) Close-up of the monopile and secondary structures with graded refinement. c) Close-up of secondary structures with partial view of monopile with boundary layer elements visible.

4 Test with a monopile

In order to test and validate the experimental and numerical methods several test cases were carried out for a circular cylinder alone, i.e. without secondary structures.

This included qualitative assessment of the inline force time series and comprehensive comparison of obtained force coefficients to well-known results in the literature.

Figure 10, Figure 11 and Figure 12 shows force time series from the three numerical calculations that can be compared to results presented in (Sarpkaya, 1976). The shape of the force times series can also be seen to have qualitatively resemblance with results presented in (Saghafian et al., 2003). The figures show examples of inline force in the range for KC -number 10 to 30. It can be observed from the figures that there was a qualitative resemblance of the shape of the force profiles with changing KC , between the experiments and the CFD models. As the KC number increased the number of vortex shedding are expected to increase as well, which leads to additional fluctuations in the force signal. An example of the vortex shedding is given in Figure 9. In this case the KC was 20 and therefore several vortex shedding per oscillation were generated.

Figure 9 Illustration of the vortex formation in a case with $KC = 20$.

These fluctuations were compared to results presented in (Sarpkaya, 1976) and we found that they qualitatively had the same shape. We also clearly see that the Morison-force model did not resolve the fluctuations due to the vortex sheddings. Both measured, and modelled forces have typical higher maximum load forces than predicted with the Morison model. This is partly related to the *least squared fit* that is the common choice for estimation of force coefficients, and partly due to inadequate representation of the vortex shedding mechanisms in a Morison force model.

Figure 10 CFD-model, $KC = 10.5$ CFD-model, $k_{\nu}/D = 0.01$. Resembles result presented in (Sarpkaya, 1976), in his fig. 24i. In his case $KC = 12$.

Figure 11 CFD-model, $KC = 15.7$, $k_{\nu}/D = 0.01$. Resembles result presented in (Sarpkaya, 1976), in his fig. 24h. In his case $KC = 16.3$.

Figure 12 $KC = 26.2$, $k/D = 0.001$. Resembles result presented in (Sarpkaya, 1976), in his fig. 24e. In his case $KC = 25.4$.

Figure 13 shows two examples of time-series of measured and modelled inline forces. The general shape of the force time series appears the same. In the measured force time series more fluctuations were observed for the low KC number, $KC_{mea} = 7.7$ compared to the CFD results. For the large KC -number, $KC_{CFD} = 31.3$, the fluctuations from the CFD calculations appear stronger compared to the measured time-series. This might stem from the use of URANS in 2D, which will be further discussed later.

Experiments for loads on rough cylinders in oscillatory flows were conducted and presented by (Sarpkaya, 1976). The results of that study cover a wide range of KC -numbers, where the interval $KC = [20, 30, 40]$ and a relative surface roughness of $k/D \leq 1.25 \cdot 10^{-3}$ is relevant for the present study and the comparison. Further, additional data from (Justesen, 1989) was included in the comparison, even though the relative surface roughness was larger, $k/D \leq 3.0 \cdot 10^{-3}$. Figure 14 shows the comparison of C_D and C_M over the tested range of the KC -number for a monopile alone. Firstly, it can be observed that the overall results of the C_D and C_M of the physical model and those of the CFD model were in good agreement, and therefore the validation of the CFD model seems confirmed in the case of the cylinders without secondary structures. Clearly, for both C_D and C_M scatter exists, which may be caused by the experimental uncertainties.

581 *Figure 13 Two examples of comparisons of force time-series from physical (measured)*
582 *and CFD models. The left panel had $KC_{mea} = 7.7$, and $KC_{CFD} = 7.9$. The right panel had*
583 *$KC_{mea} = 31.8$, and $KC_{CFD} = 31.3$. The relative roughness was in the order,*
584 *$k_s / D \approx 1.25 \cdot 10^{-3}$.*
585

586 Force coefficients for both the experimental and numerical analyses showed the
587 same behaviour as reported in (Sarpkaya, 1976), and (Justesen, 1991). The drag
588 coefficient, C_D , increased from around 0.5 to around its maximum value around 1.75 at
589 $KC = 10-11$. Then the C_D coefficient decreases and becomes almost constant as KC
590 increases from 20 to 40. The inertia coefficient decreased from its asymptotic value 2 at
591 $KC \approx 0$ to its minimum value around 0.75 based on the CFD model and 0.5 literature
592 values from (Justesen, 1991), for a KC in the range 10-15. From a KC equal to and
593 larger than 20 the C_M was constantly in the range 1.3-1.5.
594

Figure 14 The variation of C_D and C_M over the tested range of the KC -number for the monopile and jacket leg model without secondary structures attached. The relative roughness was in the order, $k_s / D \approx 1.25 \cdot 10^{-3}$ except for the case of (Justesen, 1989), where it was $k_s / D \approx 3 \cdot 10^{-3}$.

The force coefficients were found to be in good agreement with the earlier studies and also between the physical tests and numerical calculations. CFD analyses based on a URANS model is known to have difficulties in predicting the drag crisis in pure current, see for instance (Wen and Qiu, 2017). In (Saghafian et al., 2003) similar conclusions were found for pure current, while numerical simulations for an oscillatory outer flow showed better agreement with measurements. In their case the surface of the circular cylinder was smooth while it was rough in all our cases. The main problem for the URANS models have often been linked to inadequate capabilities in predicting the transition from laminar to turbulent boundary layer, which results in difficulties in finding the correct position of the separation points. The position of separation points has a significant influence on the drag force and therefore have to be modelled correct to find an accurate answer for the drag force.

URANS has difficulties in predicting the drag crisis in pure current, but we find that there were two reasons for the good agreement between measurements and CFD based on URANS modelling in our study. First of all, the outer flow was periodically oscillating and not a steady current. The dependence of the Reynolds becomes less abrupt as in pure current as reported in the literature (Justesen, 1989; Sarpkaya, 1986, 1976). Secondly, all cases were for Reynolds high numbers in the CFD calculations and trip wires were used in the physical experiments. The trip wires ensured a turbulent

boundary layer, and as earlier discussed, which was seen to results in less dependence on the Reynolds number. Often trip wires are believed to produce flow conditions valid for full scale. For high Reynolds flows we see less dependence of the force coefficients on the Reynolds number, while the relative roughness k/D has some influence. In (Bjørke et al., 2018) the forces on square cylinders were analysed based on a similar CFD model, and they also found a good agreement between physical model tests and CFD analyses in oscillatory flow. In their cases edges of square cylinders fixated the position of the separation point making it independent on the Reynolds number within the range.

Figure 15 The Maximum lift force coefficient, $C_{L,max}$, for over the tested range of the KC -number for the monopile and jacket leg model without secondary structures attached. The relative roughness was in the order, $k_s / D \approx 1.25 \cdot 10^{-3}$.

Figure 15 shows the variation of the $C_{L,max}$ coefficient over the tested range of the KC -number for both the monopile and the jacket leg models in the case of a single plain cylinder. Clearly, as expected, the lift force approached zero as KC go to 0, here unseparated flow exists and hence no lift force occurs. This trend was well preserved in both the data from the physical and the CFD model. As the KC -number increases the $C_{L,max}$ increased, with a maximum around $KC = 10 - 20$. As the KC -number increased further the $C_{L,max}$ decreased. The CFD model overestimated the lift coefficient in relation to the physical model; however, the estimated values are still in the same order of magnitude as the data of a smooth cylinder.

The enhanced lift force experienced in the CFD model may also be attributed to the two-dimensional set-up. The vortices will typically be larger in two-dimensional

models (like the present) and have difficulties in breaking down to smaller vortices. This might induce a lower pressure over larger areas leading to an enhanced lift. It should also be noted that the exact maximum obtained from the measured force time series in the physical model was slightly biased with the filtering made in the post-processing of the measured force signal. Here, the local maxima may be reduced slightly, however, given the reasonable agreement with the previous works of (Sarpkaya, 1976) this effect seems negligible.

5 Results

Results from physical and numerical experiments will be presented in this chapter. It includes examples on the force time-series, the dependence of the force coefficients on KC number and orientation of the secondary structure to the main flow direction.

5.1 Monopile with secondary structures

Figure 16 show the force time-series from the physical and numerical tests where $KC = 5.2$. In the results from the physical model experiments, the main periodic signal was overridden small short oscillations. This might stem from noise originating from the use of a carriage, which the standard filtering procedure was unable to resolve. Still, the comparison between the measured and modelled force time series showed a good agreement for the purpose estimation of force coefficients for the Morisons equation. For small KC-numbers the forces were smaller in the experiments and therefore the results become more sensitive to the filtering procedure. The CFD force time-series had approximately the same behaviour as in the experiment. Both cases were dominated by the inertia part as the maximum force was found close to the maximum acceleration.

Figure 16 Time series of inline forces for $KC = 5.2$, $k_s/D = 1.25 \cdot 10^{-3}$. Mono-pile with secondary structures at 0° orientation.

In Figure 17 $KC \approx 13$ that is in the intermediate range where both the drag and inertia terms in the Morison's force model contribute substantially to the force. The shape of the force time-series was overall the same in the physical measurements and the CFD calculations. The force had or was close to have a local maximum at maximum acceleration, but the maximum force was found close to maximum speed. However, there seems to be a tendency to larger subharmonic contributions in the CFD results.

Figure 17 Time series of inline forces for KC -numbers: $KC_{CFD} = 13.1$, $KC_{Phys.} = 12.9$, $k_s/D = 1.25 \cdot 10^{-3}$. Mono-pile with secondary structures at 0° orientation.

Figure 18 shows the force time-series for jacket-leg with secondary structures at 0° orientation. With a relative high, KC -number, $KC \approx 31$ -32, the forces will be drag

dominated. The maximum force was in both physical experiments and in the CFD calculations located close to the maximum speed. In the CFD results fluctuations in the force were present especially when the secondary structure was on the lee side of the main circular cylinder. The largest cylinders in the secondary structure could interfere with the vortex shedding from the main structure, as they were located outside the shadow zone of the main cylinder.

Figure 18 Time series of inline forces for $KC_{CFD} = 31.3$, $KC_{Phys} = 32.3$, $k_s/D = 1.25 \cdot 10^{-3}$. Jacket-leg with secondary structures at 0° orientation.

Time series with 0 degrees and 90 degrees versus for jacket leg with secondary structures are shown in Figure 19. The maximum forces appeared slightly larger for 90 degrees orientation of the secondary structures. The force time-series from especially the CFD calculations was at 0 degrees orientation slightly asymmetric compared to the 90 degrees orientation. The force-time series from the CFD-calculations appeared more fluctuating than the force time series from the physical experiments. The reason for the difference in strength of the fluctuations was not clear, but could be explained by the filtering of the signal in the experiments at 10 Hz, whereby some of the fluctuations might have been filtered out. Another explanation could be related to shortcomings of the numerical model, where the 2D URANS simulations have shown to give larger fluctuations.

Figure 19 Time series of inline forces for KC -numbers: $KC_{CFD} = 31.3$, $KC_{Phys} = 32.3$, $k_s/D = 1.25 \cdot 10^{-3}$. Jacket-leg with secondary structures 90° orientation.

5.2 Effect of orientation of the secondary structure on force coefficients

Figure 20, Figure 21, Figure 22, and Figure 23 show the variation of the C_D and C_M with the orientation of the secondary structure, θ . The individual figures represent different KC -number regimes. Note that for each plot the KC -number regime was expected to be sufficiently narrow to assume that the flow regime for each plot was the same in the physical model tests as in the CFD model tests. For the drag force coefficient, C_D , it was clear that for each of the tested KC -numbers the drag increased almost linearly with increasing θ for both the physical and CFD model results. This behaviour was expected as the “exposed” part of the overall structure that immediately was in contact with the main body of the flow increased with increasing θ , and in turn this increased the drag force on the structures.

At $\theta = 0^\circ$ the value of C_D was noticeably at the same order of magnitude or lower as in the case of a cylinder with no secondary structures. This is attributed to the secondary cylinders affecting the flow around the main cylinder similarly to a splitter plate, where the effect of the vortex shedding on the lee-side of the cylinder is damped, and in turn the drag decreases.

As $\theta \rightarrow 30^\circ$ the drag increased so C_D was of the same magnitude as in the case where no secondary structures are attached, and the C_D further increases as $\theta = 90^\circ$. Further to this, when examining the increase in C_D , one should bear in mind that some parts of the increase were inherent in the least-square-fit method applied in the present

investigation, where the diameter, D , and the cross-sectional area, A , refer to the main pile geometry.

For the drag force coefficient, C_D , it was clear that for each of the tested KC -numbers the drag increases with increasing θ until it reached a maximum around $\theta = 60$ after which it stabilized or decreased slightly. This effect was mainly caused by the “exposure” of the individual structural elements of the secondary structures to the main body of the flow being largest at this plan view orientation. Clearly, the projected length of the individual structural elements onto the transverse plane (y-plane) was largest at $\theta = 60^\circ$.

For the inertia force coefficient, C_M , the physical and the CFD model results agreed well for the lower KC -numbers. With increasing KC -number slight discrepancies were observed.. According to (Sumer and Fredsøe, 2006) the transverse vortex street appears in the range $7 < KC < 13$, where the wake consists of a series of vortices convecting out to one side of the cylinder. This effect/discrepancy can also be seen for the same interval of the KC -number in the results of the cylinders without secondary structures illustrated in the right panel in Figure 14. Apparently, C_M was more effected by this than C_D was.

Figure 20 $KC = 5.2$. Monopile with secondary structures

Figure 21 $KC = 10$, Monopile with secondary structures

Figure 22 $KC = 13$, Monopile with secondary structures

Figure 23 $KC = 31$, Jacket leg with secondary structure

The effect of the plates shown in Figure 5 was 10 % increase of the C_D coefficient when the orientation was $\theta = 90^\circ$. As the KC -number increased the discrepancy between the C_D values obtained in the physical and the CFD model decreased. A good agreement was achieved for KC in the order of 31 as shown in Figure 23.

5.3 Variation of force coefficients with KC

Figure 24 displays the variation of the C_D and C_M with KC for different θ values (0° , 30° , 60° and 90°). The figure generally support the discussions on the results of C_D and C_M made in the previous section, which are further detailed in the following. The good agreement between the C_D values obtained from the physical and the CFD models, explained in the previous section is also present in Figure 24. Here the slight discrepancies in C_D are mainly observed at very low KC -numbers ($O(2)$), which are preserved from the results of the cylinders without secondary structures presented in section 4. Similarly, the discrepancy in C_D for this range of the KC -number, where the inertia part of the force dominates, will not affect the total load (consisting of drag and inertia contribution) response significantly. Hence the discrepancy seems negligible. The C_D values obtained in the physical model are generally slightly higher than those in the CFD model. This can be attributed to the form drag and the skin friction exerted by the oscillating flow on to the secondary cylinders-carrying plates in the physical model. The lowest markers in Figure 24 represent CFD results for each orientation.

For the C_M coefficient the good agreement between the physical and the CFD model results gave confidence in the simulations. The previously mentioned discrepancy in the C_M coefficient around $KC \sim 10$ is visible in the present Figure 24. One can observe that this discrepancy is largest for the orientations $\theta = 30^\circ - 60^\circ$ in Figure 24. No clear explanation for this has been offered in the present investigation.

Figure 24 Variation of force coefficients with KC -number for 4 different orientations of the secondary structure. Structure: Monopile with secondary structures. Filled symbols: Physical model tests. Empty symbols: CFD test.

Figure 25 displays the variation of the C_D and C_M versus 4 orientations of the secondary structure attached to the jacket leg. For the drag force coefficient, C_D , it was clear that for each of the tested KC -numbers the drag increased with increasing θ until it reached a maximum around $\theta = 60^\circ$ after which it stabilized or even decreased slightly. This effect could be caused by the “exposure” of the individual structural elements of the secondary structures to the main body of the flow being largest at this plan view orientation. Clearly, the projected length of the individual structural elements onto the transverse plane (y -plane) was largest at $\theta = 60^\circ$. The general agreement between the physical and the CFD model result of C_D was reasonable. For the lower range of the KC -numbers ($KC \sim 4 - 5$) the C_D value was larger in the physical model, which could be explained in conjunction with the effect of supporting plates shown Figure 5. Here, as previously explained in the case of the monopile model, the carrying plates exert a contribution to the overall drag, which was not simulated in the CFD model. The effect of the plates on the C_D relative to a single plain cylinder was up to 10% at 90° . It should be noted that for small KC -numbers the drag part of the total force in the Morison

equation is small, typically the drag contribution to the maximum load is negligible when the KC number is smaller than 5-6. Therefore, the estimation of the drag force coefficient will be attached with some uncertainty, without affecting the force estimated from the Morison formula. As the KC number increased the discrepancy between the C_D values obtained in the physical and the CFD model decreased, and the agreement was good at $KC = 30-40$, where the drag contribution to the maximum force dominates.

The inertia coefficient, C_M , remained practically constant when changing the orientation θ . This was in agreement with the results of C_M obtained in the monopile case explained in the previous subsections. For the inertia force coefficient, C_M , the physical model and the CFD model results agreed well in the range of the KC -number 13 – 23. For larger KC -numbers discrepancies existed in the prediction of C_M as can be observed in Figure 25, which is in line with the observation discussed above, as the drag part will dominate the force for KC numbers larger than 30-40. Also observe the general offset in between C_M values measured in the physical model relative to the ones produced in the CFD model at $KC = 4.4 – 4.7$. Here it seems that the physical model overestimated the C_M value, and this was supported by the higher C_M value for the same flow climate in the case of the cylinder with no secondary structures, as shown in Figure 14. Here the $C_M = 2.1 – 2.13$ which was slightly higher than expected for this KC range considering the specific relative surface roughness when observing Fig. 4.19 in (Sumer and Fredsøe, 2006). Discrepancies can be also attributed to the CFD calculations modelling the turbulence and vortex structures as strictly two-dimensional, which will not be the case in nature/full scale nor in the physical model. The effect is not fully explained by this study. However, asymptotic theory based on potential theory needs a correction for the boundary layer to include viscous effects that could increase the C_M number due to boundary layer velocity deficit at small KC values, as reported in (Bearman et al., 1986). It is not fully clear if a similar effect is introduced when using 2D CFD where the vortex shedding cannot break up in 3D structures. This could for instance lead to a change in boundary layer thickness and therefore a difference of C_M between CFD and physical model experiments. The presented discrepancies between the physical model and the CFD model results for C_D and C_M were favoured by the relative influence of the drag and inertia force over the KC regime. (Sumer and Fredsøe, 2006) reported the range of the KC -number $0 < KC < 20 – 30$ as the inertia dominated regime, while $KC > 20 – 30$ as the drag dominated regime.

Figure 25 Variation of force coefficients with KC -number for 4 different orientations of the secondary structure. Structure: Jacket leg with secondary structures. Filled symbols: Physical model tests. Empty symbols: CFD test.

6 Summary and conclusions

In this study experimental and numerical analyses have been carried out to study the effect of secondary structures, e.g. boat-landings, on a circular cylinder as part of a support structure for offshore wind turbines. The study consisted of validation tests of forces on a circular cylinder in oscillatory flow, which were followed with studies of forces of the effect of the secondary structures on the total load. The secondary structures were represented by circular cylinders placed at different orientations around the main circular cylinder. All the analyses assumed uniform transverse to the flow direction. Therefore the numerical simulations with OpenFOAM were carried out on two spatial dimensions.

The validation tests with a circular cylinder without secondary structures showed good agreement between the physical and numerical results, as well as to the results available in the open literature. The comparison included force coefficients as well as qualitative comparison of force time history. Lift force coefficients appeared larger for the numerical calculations compared to the experimental results. This difference could be attributed to the two-dimensional set-up of the CFD-model.

The validation gave confidence in both the physical and numerical methodologies abilities to accurately reproduce the effect of the confined space in between the structural parts, acceleration of flow, vortex shedding and coherent structures of

turbulence around the secondary structures relative to the main cylinder on the simulated force response time series.

The orientation of the secondary structures had a clear influence on the drag related part of the force as the drag coefficient C_D when the secondary structure was orientated towards $\theta = 90^\circ$, i.e. perpendicular to the flow direction. When the orientation was large a large part of the structure was exposed to the flow which was found to increase the drag coefficient. The inertia part of the force, i.e. the inertia force coefficient C_M , was only affected by the orientation to small extent. The volume of the structure is important to the inertia force, and as this did not change with changing orientation, it could be expected that the inertia coefficient would be unaffected as we found.

In general there was a good agreement of the estimated force coefficients between the experimental and numerical results. It should be noted that some slight discrepancies existed for C_D at very low KC -number ($KC < 5$) and for C_M at very high KC -numbers ($KC > 25 - 30$). In both cases these discrepancies seem negligible due to the ranges of the KC -number being in the inertia dominated regime when $KC < 5$ and in the drag dominated when $KC > 20 - 30$.

7 Acknowledgement

Ørsted A/S supported the study. Their support is greatly appreciated.

8 References

- Alam, M.M., Sakamoto, H., Zhou, Y., 2005. Determination of flow configurations and fluid forces acting on two staggered circular cylinders of equal diameter in cross-flow. *J. Fluids Struct.* 21, 363–394. doi:10.1016/j.jfluidstructs.2005.07.009
- Bearman, P.W., Downie, M.J., Graham, J.M.R., Obasaju, E.D., 1986. Force on a circular cylinder in viscous oscillatory flow at low keulegan—carpenter numbers. *J. Fluid Mech.* 154, 337–356. doi:10.1017/S0022112086002999
- Bearman, P.W., Graham, J.M.R., Obasaju, E.D., 1984. A model equation for the transverse forces on cylinders in oscillatory flows. *Appl. Ocean Res.* 6, 166–172. doi:10.1016/0141-1187(84)90006-3
- Bjørke, A.S., Christensen, E.D., Carstensen, S., Petersen, K.P., Mandviwalla, X.,

889 Kristiansen, T., Baarholm, R., 2018. Study of hydrodynamic forces on complex
 890 structures, in: Proceedings of the 37th International Conference on Ocean,
 891 Offshore and Arctic Engineering, OMAE2018. pp. 1–10.

892 Bredmose, H., Hansen, E.A., Pedersen, L.M., Skourup, J., Christensen, E.D., Mitzlaff,
 893 A., 2006. Numerical reproduction of extreme wave loads on a gravity wind turbine
 894 foundation, in: Proceedings of the International Conference on Offshore
 895 Mechanics and Arctic Engineering - OMAE. doi:10.1115/OMAE2006-92258

896 Bredmose, H., Jacobsen, N.G., 2011. Vertical Wave Impacts on Offshore Wind Turbine
 897 Inspection Platforms, in: In Proceedings of the ASME 2011 30th International
 898 Conference on Ocean, Offshore and Arctic Engineering, OMAE2011. pp. 645–
 899 654. doi:10.1115/OMAE2011-49785

900 Bryndum, M.B., Jacobsen, V., Tsalhalis, D.T., 1992. Hydrodynamic forces on pipelines:
 901 Model tests. J. Offshore Mech. Arct. Eng. 114, 231–241.

902 Caretto, L.S., Gosman, A.D., Patankar, S.V., Spalding, D.B., 1973. Two calculation
 903 procedures for steady, three-dimensional flows with recirculation, in: Proceedings
 904 of the Third International Conference on Numerical Methods in Fluid Mechanics,
 905 Lecture Notes in Physics, Volume 19. Springer-Verlag, Paris, pp. 60–68.

906 Chen, H., Christensen, E.D., 2016. Investigations on the porous resistance coefficients
 907 for fishing net structures. J. Fluids Struct. 65, 76–107.
 908 doi:10.1016/j.jfluidstructs.2016.05.005

909 Cheng, X., Wang, Y., Ren, B., Wang, G., 2012a. Numerical study of hydrodynamic
 910 forces on a submarine piggyback pipeline under wave action, in: Proceedings of
 911 the 31st International Conference on Ocean, Offshore and Arctic Engineering,
 912 OMAE2012. Rio de Janeiro, Brazil, pp. 177–183.

913 Cheng, X., Wang, Y., Ren, B., Wang, G., 2012b. Numerical Study of Hydrodynamic
 914 Forces on a Submarine Piggyback Pipeline Under Wave Action. J. Offshore Mech.
 915 Arct. Eng. 135, 021701–1 to 021701–8. doi:10.1115/OMAE2012-83214

916 Cheng, X., Wang, Y., Wang, G., 2012c. Hydrodynamic Forces on a Large Pipeline and
 917 a Small Pipeline in Piggyback Configuration under Wave Action. J. Waterw. Port,
 918 Coastal, Ocean Eng. 138, 394–405. doi:10.1061/(ASCE)WW.1943-5460.0000144

919 Christensen, E.D., Bredmose, H., Hansen, E.A., 2005. Extreme wave forces and wave
 920 run-up on offshore wind- turbine foundations, in: Copenhagen Offshore Wind. pp.

921 1–10.

922 Christensen, E.D., Lohmann, I.P., Hansen, H.F., Haerens, P., Mercelis, P., Demuynck,
923 A., 2011. Irregular wave loads on a gravity based foundation in shallow water, in:
924 Proceedings of the ASME 2011 30th International Conference on Ocean, Offshore
925 and Arctic Engineering. ASME, Rotterdam, The Netherlands, pp. 1–10.

926 Christensen, E.D., Stuiver, M., Guanche, R., Møhlenberg, F., Schouten, J.-J., Svenstrup
927 Pedersen, O., He, W., Zanuttigh, B., Koundouri, P., 2015. Go offshore -
928 Combining food and energy production. Technical University of Denmark.
929 Department of Mechanical Engineering, Kgs. Lyngby, Denmark.

930 Damsgaard, M.L., Gravesen, H., Andersen, T.L., 2007. Design Loads on Platforms on
931 Offshore wind Turbine Foundations with Respect to Vertical Wave Run-up, in: In
932 Proceedings of Offshore Wind 2007 Conference & Exhibition The European Wind
933 Energy Association. p. 7.

934 Hover, F.S., Tvedt, H., Triantafyllou, M.S., 2001. Vortex-induced vibrations of a
935 cylinder with tripping wires. *J. Fluid Mech.* 448, 175–195.
936 doi:10.1017/S0022112001005985

937 Igarashi, T., 1986. Effect of Tripping Wires on the flow around a Circular Cylinder
938 Normal to an Airstream. *Bull. JSME* 29, 2917–2924.

939 Issa, R.I., 1986. Solution of the implicitly discretised fluid flow equations by operator-
940 splitting. *J. Comput. Phys.* 62, 40–65. doi:10.1016/0021-9991(86)90099-9

941 James, D.F., Truong, Q.-S., 1972. Wind load on Cylinder with Spanwise Protrusion. *J.*
942 *Eng. Mech. Div.* 98, 1573–1589.

943 Jensen, B., Carstensen, S., Christensen, E.D., 2018. Mixing of stratified flow around
944 bridge piers in steady current. *J. Hydraul. Eng.* 144, 1–13.
945 doi:10.1061/(ASCE)HY.1943-7900.0001481

946 Justesen, P., 1991. A numerical study of oscillating flow around a circular cylinder. *J.*
947 *Fluid Mech.* 222, 157–196. doi:10.1017/S0022112091001040

948 Justesen, P., 1989. Hydrodynamic forces on large cylinders in oscillatory flow. *J.*
949 *Waterw. Port, Coastal, Ocean Eng.* 115, 497–514.

950 Kalghatgi, S.G., Sayer, P.G., 1997. Hydrodynamic Forces on Piggyback Pipeline
951 Configurations. *J. Waterw. Port, Coastal, Ocean Eng.* 123, 16–22.
952 doi:10.1061/(ASCE)0733-950X(1997)123:1(16)

953 Menter, F.R., 1994. Two-Equation Eddy-Viscosity Turbulence Models for Engineering
 954 Applications. AIAA J. 32, 1598–1605.

955 OpenFOAM, The open source CFD toolbox [WWW Document], 2018. URL
 956 <https://www.openfoam.com/>

957 Paulsen, B.T., Bredmose, H., Bingham, H.B., Jacobsen, N.G., 2014. Forcing of a
 958 bottom-mounted circular cylinder by steep regular water waves at finite depth. J.
 959 Fluid Mech. 755, 1–34. doi:10.1017/jfm.2014.386

960 Pearcey, H.H., Cash, R.F., Salter, I.J., 1982. Flow Past Circular Cylinders: Simulation
 961 of Full-Scale flows at Model Scale. NMI R 131; OT-R-8208, Feltham.

962 Saghafian, M., Stansby, P.K., Saidi, M.S., Apsley, D.D., 2003. Simulation of turbulent
 963 flows around a circular cylinder using nonlinear eddy-viscosity modelling: Steady
 964 and oscillatory ambient flows. J. Fluids Struct. 17, 1213–1236. doi:10.1016/S0889-
 965 9746(03)00066-5

966 Santo, H., Taylor, P.H., Day, A.H., Nixon, E., Choo, Y.S., 2018a. Current blockage and
 967 extreme forces on a jacket model in focussed wave groups with current. J. Fluids
 968 Struct. 78, 24–35. doi:10.1016/j.jfluidstructs.2017.12.015

969 Santo, H., Taylor, P.H., Williamson, C.H.K., Choo, Y.S., 2018b. The relative-velocity
 970 version of the Morison equation for obstacle arrays in combined steady, low and
 971 high frequency motion. J. Fluid Mech. 842, 188–214. doi:10.1017/jfm.2018.130

972 Sarpkaya, T., 1986. In-Line and Transverse Forces on Smooth and Rough Cylinders in
 973 Oscillatory Flow at High Reynolds Numbers. Monterey, California.
 974 doi:10.1177/003754970107600513

975 Sarpkaya, T., 1977. In-line and transverse forces on cylinders near a wall in oscillatory
 976 flow at high Reynolds numbers, in: Proceeding of the 9th Annual Technology
 977 Conference, Paper OTC 2898. Houston, Texas, pp. 3:161-166.

978 Sarpkaya, T., 1976. In-line and transverse forces on smooth and sand-roughened
 979 cylinders in oscillatory flow at high reynolds numbers, NPS-69SL76062.
 980 Monterey, California.

981 Sumer, B.M., Fredsøe, J., 2006. Hydrodynamics Around Cylindrical Structures
 982 (Revised Edition), Advanced Series on Ocean Engineering. World Scientific
 983 Publishing Co. Pte. Ltd., New Jersey. London, Singapore, Beijing, Shanghai, Hong
 984 Kong, Taipei, Chennai. doi:10.1142/9789812772770

985 Wen, P., Qiu, W., 2017. Investigation of drag crisis phenomenon using CFD methods.
986 Appl. Ocean Res. 67, 306–321. doi:10.1016/j.apor.2017.07.012
987 Zhao, M., Cheng, L., Teng, B., 2007. Numerical modeling of flow and hydrodynamic
988 forces around a piggyback pipeline near the seabed. J. Waterw. Port Coast. Ocean
989 Eng. 133, 286–295. doi:Doi 10.1061/(Asce)0733-950x(2007)133:4(286)
990
991

Figure text

Figure 1 Schematic description of the test flume.

Figure 2 The two layouts, Left: Monopile with secondary structures, Right: Jacket leg with secondary structures.

Figure 3 Definition sketch (schematic) of the monopile set-up with the orientation of the secondary structures relative to the direction of oscillatory motion defined by θ .

Figure 4 (a) Monopile without secondary structures (b) Monopile with secondary structures.

Figure 5 Left) Support frame cantilevering the monopile model. (Right) Close-up of connection between support frame and the suspended monopile model

Figure 6 Sketch of arrangement of the load cells.

Figure 7 Illustration of the model attached to the carriage. The measuring section was 30 cm high. The total water depth was 0.75 m. The top of the measuring section was 0.23 m below the free water surface.

Figure 8 Mesh for 6 m diameter monopile with secondary structures at 60° rotation, $k_s=0.0045$ m. a) View of full domain. b) Close-up of the monopile and secondary structures with graded refinement. c) Close-up of secondary structures with partial view of monopile with boundary layer elements visible.

Figure 9 Illustration of the vortex formation in a case with $KC = 20$.

Figure 10 CFD-model, $KC = 10.5$ CFD-model, $k_s/D = 0.01$. Resembles result presented in (Sarpkaya, 1976), in his fig. 24i. In his case $KC = 12$.

Figure 11 CFD-model, $KC = 15.7$, $k_s/D = 0.01$. Resembles result presented in (Sarpkaya, 1976), in his fig. 24h. In his case $KC = 16.3$.

Figure 12 $KC = 26.2$, $k_s/D = 0.001$. Resembles result presented in (Sarpkaya, 1976), in his fig. 24e. In his case $KC = 25.4$.

Figure 13 Two examples of comparisons of force time-series from physical (measured) and CFD models. The left panel had $KC_{mea} = 7.7$, and $KC_{CFD} = 7.9$. The right panel had $KC_{mea} = 31.8$, and $KC_{CFD} = 31.3$. The relative roughness was in the order, $k_s / D \approx 1.25 \cdot 10^{-3}$.

Figure 14 The variation of C_D and C_M over the tested range of the KC -number for the monopile and jacket leg model without secondary structures attached. The relative

roughness was in the order, $k_s / D \approx 1.25 \cdot 10^{-3}$ except for the case of (Justesen, 1989), where it was $k_s / D \approx 3 \cdot 10^{-3}$.

Figure 15 The Maximum lift force coefficient, $C_{L,max}$, for over the tested range of the KC-number for the monopile and jacket leg model without secondary structures attached. The relative roughness was in the order, $k_s / D \approx 1.25 \cdot 10^{-3}$.

Figure 16 Time series of inline forces for $KC = 5.2$, $k_s/D = 1.25 \cdot 10^{-3}$. Mono-pile with secondary structures at 0° orientation.

Figure 17 Time series of inline forces for KC-numbers: $KC_{CFD} = 13.1$, $KC_{Phys.} = 12.9$, $k_s/D = 1.25 \cdot 10^{-3}$. Mono-pile with secondary structures at 0° orientation.

Figure 18 Time series of inline forces for KC-numbers: $KC_{CFD} = 31.3$, $KC_{Phys} = 32.3$, $k_s/D = 1.25 \cdot 10^{-3}$. Jacket-leg with secondary structures at 0° orientation.

Figure 19 Time series of inline forces for KC-numbers: $KC_{CFD} = 31.3$, $KC_{Phys} = 32.3$, $k_s/D = 1.25 \cdot 10^{-3}$. Jacket-leg with secondary structures 90° orientation.

Figure 20 $KC = 5.2$. Monopile with secondary structures

Figure 21 $KC = 10$, Monopile with secondary structures

Figure 22 $KC = 13$, Monopile with secondary structures

Figure 23 $KC = 31$, Jacket leg with secondary structure

Figure 24 Variation of force coefficients with KC-number for 4 different orientations of the secondary structure. Structure: Monopile with secondary structures. Filled symbols: Physical model tests. Empty symbols: CFD test.

Figure 25 Variation of force coefficients with KC-number for 4 different orientations of the secondary structure. Structure: Jacket leg with secondary structures. Filled symbols: Physical model tests. Empty symbols: CFD test.