6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
1115 June 2018, Glasgow, UK

STARTING A WAVE STRUCTURE INTERACTION CFD SIMULATION FROM AN ADVANCE TIME: HOT-START

MUSIEDLAK P.-H.[1], RANSLEY E.[1], BROWN S.A.[1], CHILD B.F.M.[2], HANN M.[1], IGLESIAS G.[1] AND GREAVES D.[1]

[1] School of Engineering, Plymouth University
Reynolds Building, Drakes Circus, PL4 8AA Plymouth, United Kingdom
pierre-henri.musiedlak@plymouth.ac.uk

[2] DNV-GL
One Linear Park, Avon Street, Bristol BS2 0PS, United kingdom
benjamin.child@dnvgl.com

Key words: Hot-start, CFD, wave-structure interaction, Mesh-deformation, Coupling

Abstract. This paper details the reduction of an OpenFOAM CFD WSI simulation to its major event, by starting it at an advance time: a new procedure named hot-start, which is a first step towards a future coupling development. The investigations concern the fluid flow and a structure motion hot-start, taken separately, and are restricted to numerical comparison. Four design waves based upon the NewWave theory are simulated with increasing starting times - where the wave field is initialized as the sum of the linear components of the considered wave - and compare to conventional ones - which are initialized with still water. The structure motion hot-start is assessed using a heave decay test: a conventional heave decay simulation is compared against several ones where the motion, velocity and acceleration of the structure are assigned for several time-steps. An initial mesh-deformation library is specifically created to assign the structure at the hot-start position by incrementally deforming the mesh towards the right structure position. Independent to the non-linearity of the case, a start 4 s prior to the main event is found to be enough to accurately represent the wave field. The motion of the structure was found to require at least 5 time-steps in order to converge to the reference one. Those results aim to be usable for other CFD WSI applications.

1 INTRODUCTION

Numerical modelling is widely used in offshore and coastal engineering to assess wave-structure interaction (WSI) since it gives increased understanding of processes such as: the evolution of the coast line; the manoeuvrability of ships; the mechanical design of floating oil and gas platforms; or of wind turbines. Many offshore standards are based upon numerical modelling, and often adapted with the experience gathered by success and failures. Marine Renewable Energy (MRE) developers use the offshore oil and gas industry standards. But, nowadays the numerous failures of the different MRE devices has proven those standards to be misfits to the sector [1]. Often a MRE - especially Wave Energy Converters (WEC) - device motion needs to be accentuated to generate power [2], unlike traditional oil and gas floating structure which are designed to controlled and limit their motion. Hence, this dynamic behaviour of MRE devices requires models which are capable of accurately simulating large motions. Also, a MRE device is often composed of multiple components which interact with each other, resulting in complex, and often highly non-linear, device motion which depends heavily on past events [3].

Therefore, these industries require a more complex numerical model which is able to assess such levels of physical complexity. Computational Fluid Dynamics (CFD) simulations solve the full Navier-Stokes equations with limited simplifications. Their use as a design tool is growing in several industries, where the offshore industry starts to recognize their reliability compare to empirical methods use in industry standards. But this increase in complexity induces an increase - sometimes drastic - in Central Processing Unit (CPU) effort. This is a major issue if such methods want to be utilized in routine design processes [1], since it limits the use of CFD to very case specific physics representation, or research bases cases.

In WSI, CFD simulations are mainly used for mechanical design of extreme loads. Typically, the engineering method uses a design wave, which hits the structure, hence allowing prediction of the loads. Using a Numerical Wave Tank (NWT) starting from still water, the CFD simulation generates the wave at the inlet, and then propagates it towards the outlet. The period of time necessary to create the wave from still water is required to build-up of fully non-linear fluid flow. This is the common set-up of a WSI CFD simulation: a similar example to this study is the WEC developer Carnegie who simulated in a NWT, the dynamic response of their device under extreme events using the NewWave description, where the simulations started from still water [4].

However, the main interest of a WSI simulation is the impact of the wave on the structure, rather than the propagation of the wave itself, and this constitutes only a small amount of the full CFD simulation. Therefore, this paper presents a novel approach that limits WSI CFD simulations to the times of interest: the simulation will start slightly before the impact - this strategy is termed 'hot-start'. It is expected to result in significant CPU savings without substantially compromising the accuracy of the results. In the case of use for WSI problems, and to maintain the accuracy of the results, a hot-started simulation requires consideration two mains issues (taken separately in this study): 1) the wave field reproduction, and; 2) the hot-start assignment of the motion of the structure.

This study is a first step in the development of a coupling between an industry standards based numerical model - WaveDyn, developed by DNV-GL in Bristol UK , and the open-source, CFD code OpenFOAM. To maximize efficiency, the coupling strategy utilises the computationally efficient method, WaveDyn, preferentially reserving the expensive NS solver for instances in which the linear assumptions of WaveDyn are violated [5]. The coupling strategy is outside of the scope of this paper. This study focuses on the achieving the hot-start for a CFD simulation in a purely numerical approach. No comparison with experimental data will be conducted. The study objective is to prove the feasibility of a CFD hot-start for the wave field, and for a rigid body motion.

2 METHODOLOGY

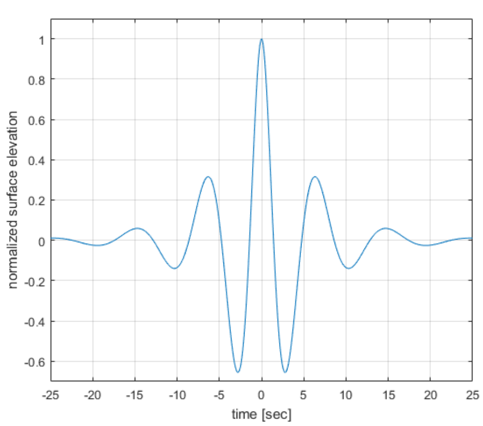
2.1 Initial conditions for the wave field variables

The objective of the following method is to be able to accurately hot-start a $2D$ wave-only simulation using a design wave. It aims to be adaptable for other wave design. Investigations are compared against CFD simulations using the usual strategy, i.e. starting from still water at time $t = 0$.

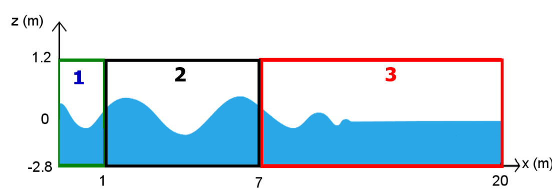
2.1.1 The reference NWT and design wave set-up

Design waves are typically used to assess the survivability of a structure in extreme wave events, [4], [2]. They aim to generate the maximum loads the structure would be exposed to during its design lifetime. This engineering method has been subjected to debate: indeed, for structures subject to large motions, more extreme loads can be found outside the scope of this extreme representation depending on the historic of the device motion [6]. But, this method is still widely used, and is used in this study.

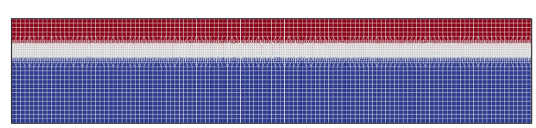
This study uses a Pierson-Moskowitz spectrum from a 100 year storm hindcast data obtained at the Wave Hub site ($Tz = 14\,\text{s}$, $Hs = 4.4\,\text{m}$, [7]). The design wave is defined using the NewWave [8] wave representation, which produces, for a given sea-state, the average shape of the highest wave with a specified exceedance probability [9]. This shape is a focus event which occurs at a specific position in time and space, as shown in Figure 1a. The mathematical description at first order is defined by a sum of linear waves; thus it is easy to implement at the NWT inlet. Four NewWave type focus events of increasing steepness are obtained from this hindcast [6].



(a) Theoretical shape of the surface-elevation of a NewWave event at focus location, generated by a Pierson-Moskowitz spectrum



(b) A schematic representation of the $2D$-NWT: 1 is the inlet; 2 is the working region; and 3 is the relaxation zone



(c) The domain is a $20 \times 0.1 \times 4\,\text{m}$ cuboid consisting of cubic background of 6 cells per meter, refined to level 3 around the mean-water line

Figure 1: The reference design wave and the $2D$-NWT set-up

The four focus event are reproduced in the 2 dimensional (2D) NWT represented in

Figure 1b, which dimensions are based upon a previous study [10]. The *waves2Foam* library is used for the wave generation and absorption methods [11]. The wave is generated at the inlet, which is the region number 1 in Figure 1b. At the inlet, a superposition of linear wave components, obtained using a Fast Fourier Transform (FFT) of the experiment wave gauge located the furthest upstream, is used to generate the wave. The components are selected by incrementally adding waves in order of magnitude (largest first) until a user prescribed precision is achieved [5]. This optimization of the selection of the number of components was found to save Random Access Memory (RAM) and CPU effort [12]. The wave then propagates in a fully non-linear manner along the NWT in the working region, which is the region number 2 in Figure 1b. The outlet, or relaxation zone, is the region 3 in Figure 1b, where the wave is absorbed. To assure the fully non-linear propagation of the four waves cases, the reference CFD simulations are starting from still water at time $t = 0$; the conventional CFD set-up.

The $2D$-NWT mesh shown in Figure 1c is generated using the commands *blockMesh* and *snappyHexMesh*. The background mesh is made of 6 square cells per meter, as square meshes were found to converge more rapidly than one of increasing ratios [2], and to more accurately reproduce the physics. This background mesh is then refined three times using the octree refinement strategy [13] around the mean water line along the full length of the tank (as simulations were found to be slower and different from a fully square mesh if a shorter refinement region was used).

The experimental equivalent position of the structure (here the X-MED buoy [6]) is considered as the focus space location, abscissa $x = 5.58\,\text{m}$. The focus time, t_{focus}, is chosen as the time where the highest surface-elevation is observed at this location. The flow field characteristics are measured at this position: the surface-elevation as a function of time; the water velocities and pressure along the water-column. The surface elevation is measured during the simulation by the library *waveGaugesNProbes* [11]. The velocity and pressure at time $t = t_{focus}$ along the water column are post-processed using Paraview by slicing the domain at the structure position, and then extracting the two profiles.

2.1.2 Method

A hot-start CFD NWT set-up only differs from a conventional one by its starting time, t_{hot}, and the initial set-up of the wave fields. The wave field is described at the inlet by the same sum of linear components in both cases, but for a hot-start simulation because the starting time is different than 0, the description applies to the full NWT length at the hot-start time, t_{hot}. Therefore, compare to a conventional CFD simulation where the fluid has been propagating in a fully non-linear manner across the NWT until the hot-start time, a hot-started simulation at this starting time, i.e. t_{hot}, will lack in accuracy.

But the differences are expected to reduce as the two simulations run, as the hot-started simulation build-up to a fully non-linear description. In other words, in order to reproduce a non-linear event, it is expected that a period of time is required for a hot-started simulation to converge to the reference one. So, the time of the focus extreme event is considered as the location in time where the simulation needs to be hot-started:

$t_{focus} = t_{hot}$. Additionally, a period of time, t_{minus}, is subtracted to the hot-start time so that the hot-started simulation can build up to the solution. It is required that this parameter is smaller than the hot-start time; otherwise the conventional CFD set-up would be used.

Therefore, the method consists of running hot-started simulations with increasing t_{minus}, from 0 to t_{hot}. As t_{minus} increases, the hot-started simulation have more time to build up towards the reference one, and by comparing the reproduction of the focus event, a convergence is expected. The surface elevation, velocity and pressure fields predicted by the hot-started simulations are benchmarked against the predictions from the reference simulation, which used the conventional setup. To investigate the dependency of the hot-start with the non-linearity of the event, the four focus event of increasing steepness are used. The convergence of the hot-started simulation are expected to be depend on the non-linearity of the case.

2.2 Positioning the structure and assigning initial motion state

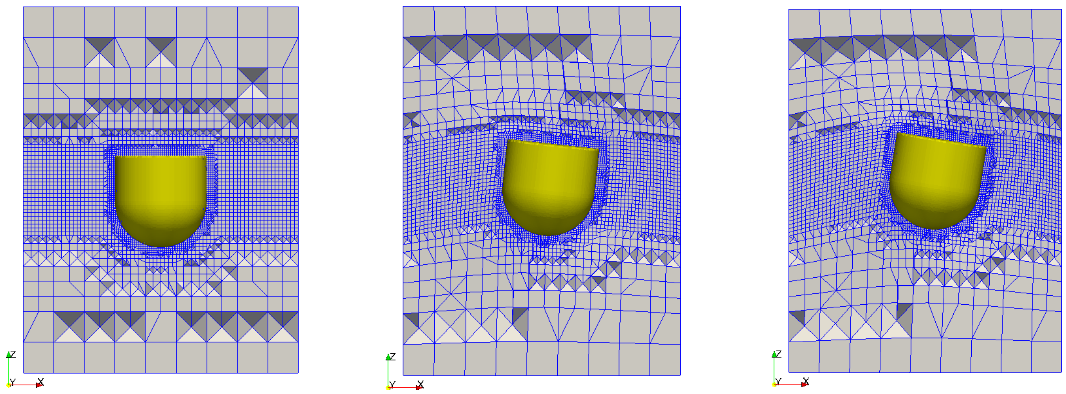
In this second sub-section, the study focuses on the hot-start for a rigid-body only, trying to avoid the influence of the fluid. Its objective is to accurately hot-start a $3D$ simulation involving a simple motion of a rigid-body to provide the proof of the hot-start concept for WSI applications.

2.2.1 The initial mesh deformation - $deformDyMMesh$

As the structure position at the hot-start is supposed to be known, the usual approach is to generate a new mesh with the up-to-date geometry file (*.stl* using a Computer Aided Design software (CAD)) from this new position. The final mesh is undeformed, and any new deformation due to the movement of the structure, will deform the mesh. Also, if the hot-start position is different from the structure's equilibrium, the deformation of the mesh will increase as the structure returns to its equilibrium. This might led to a mesh of lower quality at the equilibrium which can generate some instabilities. This pre-process step can be quite time-consuming, and, in the future use of this study for a software coupling, the structure position could be different for each simulated cases. And, this would require to generate a different geometry file each time, or to make this automatically, which means another coupling process with the CAD software.

This study uses a new approach, where only the mesh with its structure at equilibrium is first needed. By deforming this mesh, the structure is moved to its position at the initial hot-start time. One advantage of this method is that only one geometry model is required, and that any structure position can be obtained simply be deforming the mesh.

For this purpose, a new library was created based on the $waveDyMFoam$ solver and the $rigidBodyDynamics$ library from OpenFOAM-4.1; named $deformDyMMesh$. Using as input an offset from the structure position and a number of iterations, it moves the structure by the amount defined by the amplitude of the offset divided by the number of iterations. The mesh is then updated, and this results in a new deformed mesh. By repeating this process the number of iterations, the structure ends at the wanted the



(a) Original mesh (b) Half-deformed mesh (c) Mesh fully deformed

Figure 2: Steps of the $deformDyMMesh$ use

position, and the mesh is deformed accordingly. The larger the offset, the more iterations are required to ensure the mesh quality of each iteration. A poor mesh quality results often in squeezed cells, and an inability of using the deformed mesh for any further simulations.

An example of the capacity of this library is shown in Figure 2, where a structure, here the X-MED buoy [6], is successfully moved from its initial position using the $deformDyMMesh$ library, and that it results in a deformed mesh of quality.

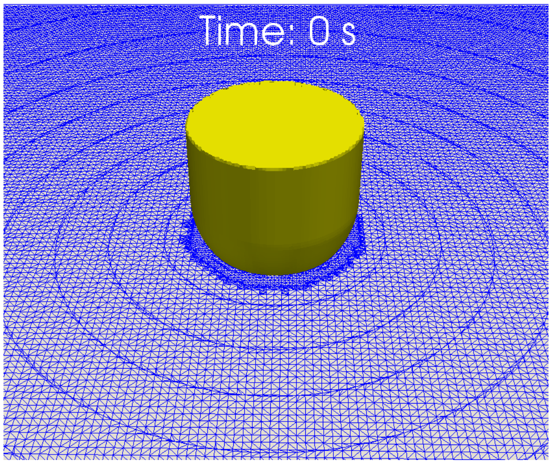
The hardware used for this study is the Viglen Genie computer, equipped with Intel Xeon E5-1680 v4 at 3.40 GHz with 16 processors, where the 64 version of Ubuntu is directly installed. All the commands are run in serial mode.

The execution of the original mesh in Figure 2a takes 14 s for this hardware: $blockMesh$ of a $1.5 \times 1.5 \times 2$ m box with 6 cells per meter, refined to level 3 between $[-0.25; 0.25]$, and to level 4 at the structure surface; a total of 160306 cells. The deformation done in this example moves the structure by: 0.06 m in surge, 0.14 m in heave, a 10 degree angle in pitch, and none in sway, roll and yaw. Depending on the number of iteration required for the deformation, $deformDyMMesh$ takes between 50 s for 50 iterations, and drops to 10.2 s for 15 iterations, which is the minimum number of iterations found for this amplitude of deformation for this case.

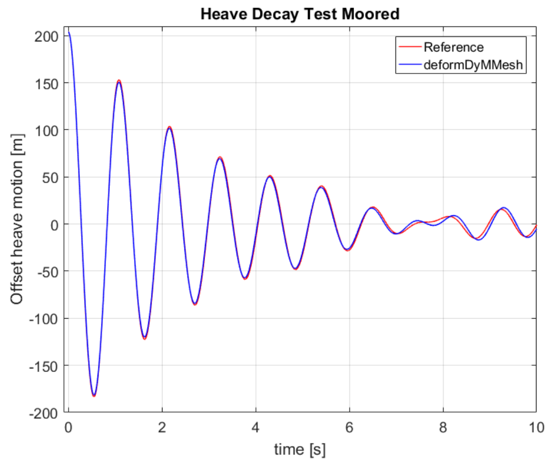
However the proven quality of the mesh generated by the $deformDyMMesh$ library, no proof of its ability not to influence the results has been done so far.

2.2.2 Proof of use of $deformDyMMesh$

To prove the use of this library, a heave decay test is performed. The simulation reference is carried out using a geometry file updated according to the heave decay release position to insure the initial mesh to be undeformed. And, it is opposed to a simulation starting with a mesh deformed by the $deformDymMesh$ library from the structure at its equilibrium position to the release position. The flow fields are set as still water. No hot-start are considered here, and the structure used for the proof of $deformDyMMesh$ is the X-MED buoy, which has no velocity nor acceleration set at the starting time for both cases.



(a) The case using $deformDyMMesh$ with a cut at the mean water line at time $t = 0\,\text{s}$



(b) The heave displacement of a conventional simulation compared to one using $deformDyMMesh$ library

Figure 3: A decay test to prove the use of the $deformDyMMesh$ library

Figure 3b shows the results of the simulations by comparing the two structures heave displacement as an offset from the structure equilibrium position. Very slight differences are found before time $t = 6\,\text{s}$. And, those are expected to be due to the small difference in the initial flow fields as the deformed mesh generates small circular like waves, which can be seen in Figure 3b. After time $t = 6\,\text{s}$, the small are growing due to the radiated waves generated by the structure motion and reflecting on the walls. But, those differences are considered as negligible, where the circular waves and the slight differences are expected to be sensitive to the heave decay test only. Therefore, this proves the use of the $deformDyMMesh$ library for the structure initial position assignment without influencing the simulation.

2.2.3 The initial structure velocity and acceleration

However, for an hot-started simulation, the assignment of the position expected to be not sufficient on its own, and that the initial velocity and acceleration of the structure are of importance, and therefore, are required as additional initial conditions on the structure motion.

The importance of specifying the structure initial velocity and acceleration for a hot-start is investigated using the same heave decay test, with the same reference case, and compared against hot-starting simulations of different initial velocity and acceleration set-up. In order to investigate only the initial structure motion conditions, it is necessary to avoid, or at least reduce, the effects of the fluid. So, the hot-started simulations are starting from an early time, $t_{hot} = 0.1\,\text{s}$, where the influence of radiated waves due to the structure drop can be neglected, but where the structure motion is significant enough not to be neglected. Using the previous conclusion, the hot-started simulations use the $deformDyMMesh$ library to assign the structure position at the hot-start time. Firstly, a simulation will not specify the velocity and acceleration to prove its necessity for an hot-started case. Secondly, several simulations will specify the velocity and acceleration found by the reference simulation for an increasing number of time-step, before releasing the structure.

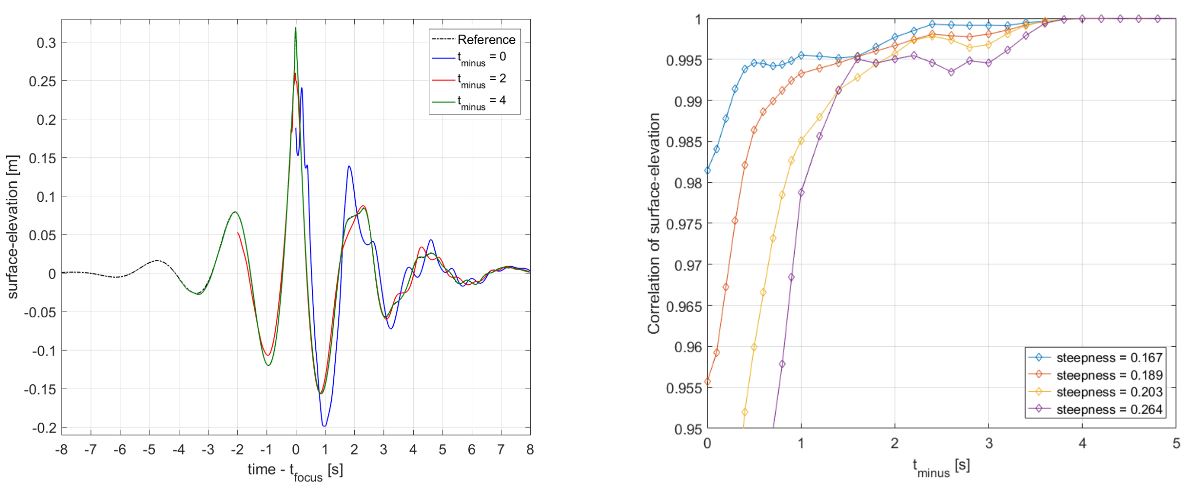
The comparison is limited to the first second of simulation to avoid reflections, and because the motion is expected to converge towards the reference one. An investigation on the number of corrective time-steps required for the convergence is also conducted.

3 DISCUSSION

3.1 Initial conditions for the wave field variables

3.1.1 Surface-elevation

In Figure 4a the surface-elevation of $2D$ wave-only simulations of the steepest case are plotted in colour against the reference simulation, the dotted line. Figure 4b presents the correlation between a hot-started simulation and the reference one, as a function of the t_{minus} used for each simulation. The wave cases are numbered by their experimental measured steepness [6].



(a) The surface-elevation of three different hot-started $2D$ wave-only simulations, for the steepest case compared with the reference

(b) Correlation comparison of the surface-elevation of 2D simulations starting at a specific time against the reference one

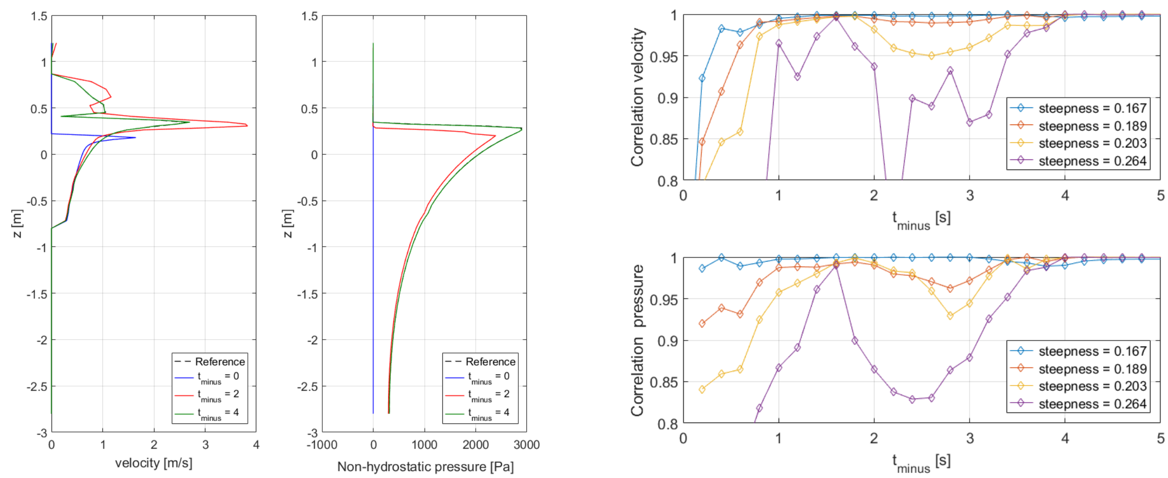
Figure 4: The surface-elevation hot-start results

The surface-elevation of the hot-started simulation with $t_{minus} = 4$ is the green line in Figure 4a, and is on top of the reference one. Indeed, in Figure 4b, $t_{minus} = 4$ clearly appears as the first converged solution for the surface-elevation representation. And, unexpectedly, this result is valid for the four cases, hence not depending on the steepness or the non-linearity of the wave. But, the convergence of the solution is slower as the non-linearity of the wave increases. Consequently, if a lower correlation criterion would be used, a lower t_{minus} could be used for the less steep cases.

As expected, when no build-up period is allowed, for $t_{minus} = 0$, the sum of linear component is set as the initial condition across the NWT, which results in significant differences, as shown by the blue curve in Figure 4a.

3.1.2 Velocity and Pressure

Figure 5a presents the fluid velocity and pressure profiles at focus time, at the tank location $x = 5.58$m, for the steepest case. The black dotted line is the reference case, and three t_{minus} hot-start simulations are plotted on top. For the four waves cases, the correlation between a hot-started simulation and the reference one, is plotted against t_{minus} in Figure 5b.



(a) Velocity and pressure profile at focus time, at the tank location $x = 5.58$m, for the steepest case

(b) Correlation comparison of the velocity (top) and pressure (bottom) profiles of 2D simulations starting at a specific time against the reference one

Figure 5: The fluid velocity and pressure hot-start results

As for the surface-elevation, the solution converges at $t_{minus} = 4$ (Figure 5b), where the two profiles are exactly on top of the reference in Figure 5a. Therefore, all the results and behaviour previously obtained with the surface-elevation comparison are valid for the velocity and pressure profiles comparison.

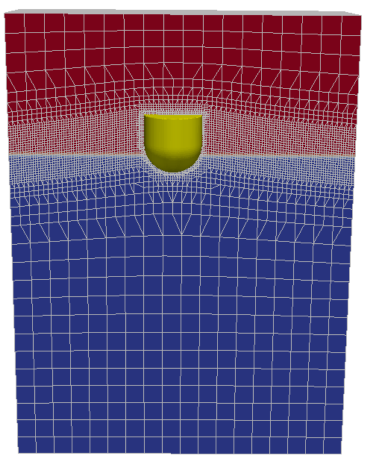
But it could be noted that for the steepest case, there is a real need of using $t_{minus} = 4$ as the correlation drops down for t_{minus} values between 2 and 3; purple curves on Figures 5b top and bottom. A less significant reduction can be observed for less steep cases, with a minor amplitude in the correlation of the surface-elevation in Figure 5b.

Using a $t_{minus} = 4$ hot-start simulation compare to a conventional one allows a reduction of 25% of CPU for the least steep case, and of 12% for the steepest one.

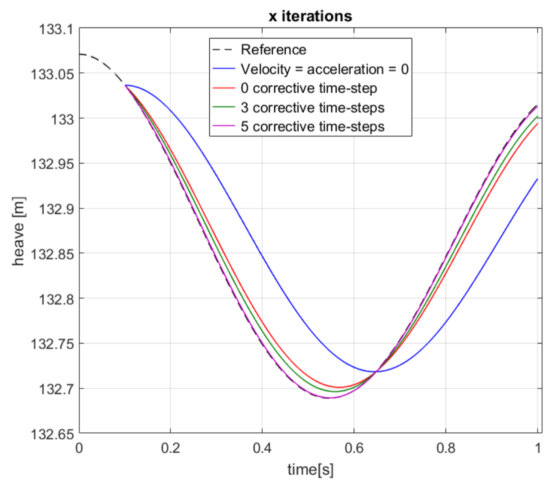
3.2 Positioning the structure and assigning initial motion state

3.2.1 Hot-start heave decay test

In Figure 6a, the initial position of the structure is set, thus the mesh is deformed. The flow fields are set as flat water across the domain (water is in blue, air in red). This initial set-up is used for the four different hot-started cases presented in Figure 6b, where several rigid body hot-start tries are compared. The referent heave decay is plotted in black dotted line.



(a) The mesh deform using the $deformDyMMesh$ library, at the hot-start time $t = 0.1\,\mathrm{s}$ for a heave decay test



(b) Comparison of different initial motion-state procedure for the 0.1$\dot{s}$ heave decay hot-started simulation: when the velocity and acceleration are or are not specified; and when additional corrective time-steps are used

Figure 6: The structure motion hot-start results

If the velocity and acceleration of the structure are not specified - hence zero - in the initial set-up, then the simulation results as a decay test released from a different height; the blue curve on Figure 6b. However, once the velocity and acceleration in the initial hot-start set-up are specified, significant improvements can be observed; the red curve on Figure 6b. Finally, the addition of corrective time-steps causes the simulation to converge towards the reference solution; Figure 6b shows that, the assignment of an initial motion can be done using at least 5 corrective time-steps.

4 CONCLUSIONS

- This paper investigates the starting of a WSI CFD simulation from an advance time, named hot-start. It aims to reduce the CFD simulation strictly around the non-linear event, by reducing the build-up time usually used in CFD to launch a simulation. It also constitutes an important piece of a future coupling procedure. In this study, the hot-start investigations are restricted to the wave field and the structure motion, separately.

- Compared to a non hot-started CFD simulation, the representation of the focus event was found to be accurately reproduced if started $4s$ before the main event; hence not requiring more than $6s$ of simulation. This conclusion was found to be independent on the non-linearity of the wave, and also confirmed through the three flow field: surface-elevation, water column velocity and pressure.

- A new library - $deformDyMMesh$ - was achieved in order to deform the mesh according to the structure position at hot-start, and its use was proven to have no influence on the results of the simulation.

- The assignment of motion of a structure was found to require at least 5 corrective time-steps - where each position and motion found by the previous time-step were corrected using the reference case - before converging to the referent solution.

- Therefore, this study proves the use and possibility of an advance start for CFD simulations in WSI cases based on a focus event. The method and the results are expected to be adaptive to other WSI in different CFD applications.

5 ACKNOWLEDGEMENTS

The code presented here is available through the Collaborative Computational Project in Wave Structure Interaction (CCP-WSI) [EP/M022382/1], which brings together the WSI community to share CFD developments in an open source environment (https://www.ccp-wsi.ac.uk) [14].

The author would like to thank his supervision team for their everyday help and advices. Thanks to S.A. Brown for its reliable help in OpenFOAM coding. Also, the author would like to thank all people involved in OpenFOAM development, from small codes to online tutorial and forums.

REFERENCES

[1] CCP-WSI Working Group, *Wave Structure Interaction Computation and Experiment Roadmap - Part 1: A Report on the 1st CCP-WSI Focus Group Workshop*, 2016

[2] Ransley, E (2015). "Survivability of wave energy converter and mooring coupled system using CFD", *PhD thesis, School of Marine Science and Engineering, University of Plymouth*

[3] Pinna, R., Cassidy, M., Dynamic analysis of a monopod platform using constrained NewWave, International Conference on Offshore Mechanics and Arctic Engineering, *23rd International Conference on Offshore Mechanics and Arctic Engineering, Volume 2*. (pp. 141-148). doi:10.1115/OMAE2004-51158.

[4] Rafiee, A.; Wolgamot, H.; Draper, S.; Orszaghova, J.; Fivez, J.; Sawyer, T. Identifying the design wave group for the extreme response of a point absorber wave energy converter. *In Proceedings of the Asian Wave and Tidal Energy Conference (AWTEC), Singapore, 2428 October 2016*

[5] Musiedlak, P-H, Ransley, E, Greaves, D, Child, B, Hann,Mand Iglesias, G (2017). Investigations of model validity for numerical survivability testing of WECs , *Proceedings of the 12th European Wave and Tidal Energy Conference (EWTEC)*, Cork, Ireland.

[6] Hann, M., Greaves, D. & Raby, A. (2015), 'Snatch loading of a single taut moored floating wave energy converter due to focussed wave groups', *Ocean Engineering 96*, 258271. URL: http://dx.doi.org/10.1016/j.oceaneng.2014.11.011

[7] Halcrow (2006), South West of England Regional Development Agency, Wave Hub development and design phase, coastal processes study report, pp19, 2006.

[8] Chaplin, J. R. (1996), On frequency-focusing unidirectional waves, International Journal of Offshore and Polar Engineering, 6(2), 131137.

[9] Xu,L., Barltrop,N., Okan, B.,(2008). Bow impact loading on FPSOs 1 - experimental investigation, Ocean Engineering 35, 11481157, https://doi.org/10.1016/j.oceaneng.2008.04.013

[10] Musiedlak PH., Greaves D., Child B., Iglesias G., Hann M., and Ransley E.J., Geometric parameters for a 2D-Numerical Wave Tank, *International Network for Offshore Renewable Energy*, 12-18 June 2016, Nantes, France; and *PRIMaRe* 2016, University of Bath UK.

[11] Jacobsen, N, Fuhrman, D and Fredse, J (2012). A wave generation toolbox for the open-source CFD library: OpenFOAM®, *International Journal for Numerical Methods in Fluids*, Vol. 70, pp. 10731088

[12] Brown S.A., Musiedlak P.H., Ransley E.J. and Greaves D.," Numerical simulation of focused wave interactions with a fixed FPSO using OpenFOAM 4.1", *ISOPE* (2018)

[13] Greaves, D., Borthwick, A., 1999. Hierarchical tree-based finite element mesh generation. Int. J. Numer. Meth. Eng. 45, 447471.

[14] A Collaborative Computational Project in Wave Structure Interaction http://www.ccp-wsi.ac.uk/