Efficient Long Term CFD Simulation of the Tri-Floater Using ComFLOW

MSc. Thesis Aerospace Engineering

Calvin Vriends
2021

Efficient Long Term CFD Simulation of the Tri-Floater Using ComFLOW

MSc Thesis Report

by

Calvin Vriends

Wind Energy Group, Faculty of Aerospace Engineering, Delft University of Technology, GustoMSC

to obtain the degree of

Master of Science

in Aerospace Engineering

at Delft University of Technology,

to be defended on Friday November 5th, 2021 at 13:00.

Student number:	4592336	
Supervisors:	dr. A.C. Viré,	TU Delft (Wind Energy)
	ir. F.A. Huijs,	GustoMSC (Team Lead, Tri-Floater)
Thesis committee:	dr. A.C. Viré,	TU Delft (Wind Energy, AE)
	ir. F.A. Huijs,	GustoMSC (Team Lead, Tri-Floater)
	dr. ir. W.A.A.M. Bierbooms,	TU Delft (Wind Energy, AE)
	dr. O.J. Colomés Gené.	TU Delft (Offshore Engineering, CiTG)

Preface

This work is the final result of my master thesis project for the Aerospace Engineering degree at Delft University of Technology.

First of all, I would like to thank my supervisors, Fons Huijs at GustoMSC and Axelle Viré at TU Delft. During my time working on this thesis, I have learned a lot. While it was not always easy working primarily from home, your guidance helped me bring this thesis project to a good end. The chance to work on a new sustainable development like floating offshore wind turbines has motivated me during the project.

I would also like to thank everyone at GustoMSC, from the first day arriving at the office, I felt welcome in the company. I have met a lot of friendly people and had good discussions. A special thanks to my fellow graduate students Niels, Laura and Nina. The coffee breaks, office walks and ping pong lessons were always very entertaining and a welcome distraction.

I want to thank the graduation committee for reading my thesis work and grading the project.

When I started studying at De Haagse Hogeschool six years ago, I did not dare to dream that I would be performing a graduation project to obtain a masters degree at the TU Delft six years later. The road here was not always easy, this is why I would like to thank some people who were not involved in the project but also helped me. I wish to thank my little family at home, Rosanne and Siem, for always supporting me and putting everything into perspective. The confidence in my ability has always motivated me to keep going. Lastly, I would also like to thank my family and friends, who were always there for me and which I could always go to to take my mind off studying.

Calvin Vriends,
Rotterdam, October 2021

ABSTRACT

One of the emerging technologies in sustainable energy is Floating Offshore Wind. GustoMSC designs a structure for Floating Offshore Wind, the Tri-Floater. One of the problems that need to be investigated before building the unit is wave impact loading. Wave impact occurs when a large wave hits the structure, placing large forces on the structure.

In this research project, a model is developed to find the wave impact loads in 3-hour sea states in which the significant wave height and period can be set. The model will work using CFD solver ComFLOW and time-domain solver aNySIM.

In the first part of the thesis, a CFD model and an aNySIM model are created and verified with model test data using decay test analysis. Decent results were found for the period and damping of the motion by both the CFD and aNySIM model. When more time is spent on the CFD model, it could closely match the model test results.

In the second part of the thesis, waves are included in the simulation. A grid sensitivity study is done with 2 m, 1m and 0.5 m grid sizes. The difference between these grids is slight regarding wave height dissipation. Irregular waves are used to do longer simulations. Two tests are done, a full 3D CFD simulation and a combination of a 2D CFD simulation with time-domain solver aNySIM. When comparing the computational time, the 2D CFD simulation proved to be the most practical solution, saving much computational time.

Finally, a methodology has been designed in which a long term wave impact study can be done. A 2D CFD simulation is performed to simulate the wavefield in the domain. This 2D CFD simulation is used as input for aNySIM, in which the motion of the Tri-Floater is simulated. From this motion, the air gap is calculated, and events are selected that will be simulated in a full 3D CFD simulation. The 3D CFD simulation uses initial and boundary conditions from the 2D wave field. It also uses the motions that are calculated in aNySIM for stability. In this 3D CFD simulation, the wave impact pressures can be obtained.

The methodology results are hard to judge for value, as there are still some problems that need to be solved. When these problems are solved and more time is spent on improving the individual parts of the simulation, it would be interesting to compare the results to model test data if this is to be done in the future.

Contents

List of Abbreviations

CFD Computational Fluid Dynamics.

CFL Courant–Friedrichs–Lewy condition.

DNV GL Det Norske Veritas.

ECN Energy Research Centrum.

FOWT Floating Offshore Wind Turbine.

GABC Generating and Absorbing Boundary Conditions.

JONSWAP Joint North Sea Wave Project.

LCOE Levelized Cost of Electricity.

MARIN Maritiem Research Instituut Nederland.

RAO Response Amplitude Operator.

List of Symbols

H_s Significant Wave Height.

S Wave Spectrum.

T_p Peak Wave Period.

X Surge.

Y Sway.

Z Heave.

α Surge.

β Pitch.

$\beta_{\mathbf{crit}}$ Percentage Cricical Damping.

γ Roll.

ω Wave Frequency.

φ Amplitude.

ζ Wave Amplitude.

dt Time Step.

List of Figures

List of Tables

1

INTRODUCTION

When this thesis is written, the whole world is in the middle of a global pandemic, significantly impacting our daily lives. A crisis on this scale, with the whole world involved, has seldom been seen before. Many people believe that climate change is the next global crisis in the making. In the Paris Climate Agreement, it is agreed upon that global warming should be less than 1.5 °C compared to pre-industrial levels. The European Commission believes that wind energy will play a prominent role in becoming carbon-neutral by 2050. They have estimated that the installed capacity of offshore wind energy in 2050 should be between 230 GW and 450 GW [1].

1.1. FLOATING OFFSHORE WIND ENERGY

If this goal is to be reached, the development and ambition need to be increased massively. An emerging technology that will contribute to this ambition is Floating Offshore Wind Turbine (FOWT). In 2020 Det Norske Veritas (DNV GL) has released the Energy Transition Outlook Report 2020, in which they predict that by 2050, floating wind will reach an installed capacity of 250 GW, which is 20% of the total offshore wind market. The Levelized Cost of Electricity (LCOE) for floating wind is expected to be approximately 40 USD/MWh, compared to about 30 USD/MWh for fixed offshore wind in 2050.

Floating offshore wind gives access to new areas with a large wind resource availability that is hard to reach by conventional fixed offshore wind because of its water depth. There have been multiple pilot FOWT farms and prototypes to test the technology. Currently, the technology is getting closer to commercial development, with some large projects being planned in the short future.

As the knowledge, expertise and equipment from the oil and gas sector can be used in the FOWT technology, this can be an opportunity for companies in this sector to move on to more sustainable business. However, next to the opportunities, there are also some challenges to be overcome. The challenges are not only technical but also related to the market and the risks that come with developing a new technology. Generally, it is believed that these challenges will be overcome, and FOWT will continue to develop into a widely used technology worldwide.

There are three main types of floating offshore wind energy structures. The main difference in the systems is the type of stabilization. An overview of the techniques is given in Figure 1.1 and below:

- The spar buoy concept uses an underwater ballast. The foundation is made such that the centre of gravity is below the centre of buoyancy. A long cylindrical structure is used. The bottom of the structure is filled with ballast to lower the center of gravity.
- The semi-submersible platform concept gets it's hydrostatic stability from a floating structure. The structure is connected to the seabed with mooring lines connected to anchors. Most of the designs use several large columns that are connected to the wind turbine.
- The tension leg platform concept uses a highly buoyant structure with arms connected to tensioned tendons connected to anchors on the sea bed. Unlike the semi-submersible platform, the tension leg platform is only stable when the tension tendons are connected.

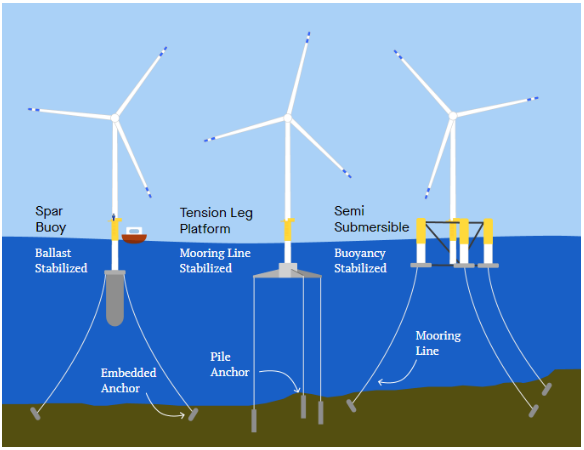


Figure 1.1: An overview of the three main floating offshore wind structures. Image taken from [2].

All of these systems have pros and cons and are being developed by multiple companies around the world.

1.2. GUSTOMSC AND THE TRI-FLOATER

GustoMSC is an offshore company with a broad product portfolio designing for offshore oil & gas, offshore wind & renewable and civil construction markets.

For this research project, GustoMSC's Tri-Floater is used as a test case. The Tri-Floater is a concept for a floating offshore semi-submersible wind turbine structure. In cooperation with the Energy Research Centrum (ECN) and the Maritiem Research Instituut Nederland (MARIN), GustoMSC developed the tools and the methodologies to deliver basic designs on commercial projects with the Tri-Floater [3].

Some of the design concepts are summarized, as these are of interest to this research project.

- Tri-Floater not only stable due to mooring lines, gets its stability from the structure as well.
- Mooring system consists of catenary mooring system consisting of three chains.
- Natural periods of motion outside of the range of wave periods, gentle and mild accelerations.
- No active ballast system.

After various iterations of the design loop, model tests were performed on the Tri-Floater, which are useful for this project. The design concepts and the fact that model tests were performed make the Tri-Floater a good test case for the research project.

Figure 1.2: Artist impression of the Tri-Floater designed by GustoMSC.

1.3. Research Motivation

In this section, information will be given on the motivation of the project, and wave impact in general.

Designing a floating offshore structure consists of many design loops. When looking, for example, at the Tri-Floater, one can imagine that there are many coupled components. The design of the floating structure and the mooring system influences the control system of the wind turbine and vice-versa. The design methodology consists of many loops; changing something in the design at a later stage could also influence earlier design decisions. This is an iterative process that is repeated until an acceptable concept design is found. As this can take many loops, the benefit of short simulations is significant.

In this research project, an investigation is made on a methodology to include wave impact loads in the design process.

When designing floating offshore structures, wave impact calculations have become more relevant in the last few years. A proper analysis should be integrated into the design process. A good example where mistakes were made in this process is the accident with a semi-submersible offshore drilling unit COSL Innovator in 2015. In this accident with deadly consequences, a large wave impacted the COSL Innovator. After the incident, the Norwegian Authorities started an investigation on how this incident could have happened and how it could be prevented in the future.

In the incident report [4], a thorough investigation is done. It includes a part on the causes in the design process, which is very relevant. The simple cause for the death and injuries is that a large wave hit the unit on the structure. The wave impact pressure was a multiple of what the windows were designed for. During the air gap analyses that were performed for the COSL Innovator, it was found that some of the analyses showed negative air gap situations, up to four meters. According to DNV GL, attention was directed at vertical slamming only. It was not usual practice to include horizontal wave slamming on the topside structure. The authorities indicate that this is one of the underlying causes of the accident that occurred.

DNV GL has concluded that wave impact loads should have a more significant emphasis in the design process. Two Offshore Technology Guidelines have been developed for this cause, DNVGL-OTG-13 and DNVGL-OTG14. These guidelines give insight into the air gap calculation and the prediction of horizontal wave impact loads.

There are multiple ways to change a design to limit the impact of wave slamming. For example, it is possible to increase the air gap by increasing the height of the deck box. However, this has a significant impact on the stability of the Floater. Another possibility is to accept that wave impact will occur and reinforce the deck box. This solution will also add weight to the structure. The problem's solution will probably lie between those options, and optimization is helpful to include in the design process. If Computational Fluid Dynamics (CFD) can be included in the design process, it will reduce time and cost, as model testing, many different models is very extensive. With CFD, it is possible to simulate the Tri-Floater for a long time (hours), in irregular seas. However, accurately simulating for multiple hours can become computationally expensive. In this project, the goal is to find a methodology such that it is possible to assess wave impact loading using CFD for long time spans as efficiently as possible.

1.4. RESEARCH QUESTIONS AND THESIS OBJECTIVE

The research project will answer the following research question:

> How can we develop an efficient framework in which it is possible to do a long term CFD analysis of the Tri-Floater which captures wave impact loads accurately using ComFLOW?

To answer this research question, smaller sub-questions are set up:

1. How close can the results of the CFD simulation match the model test data with respect to the decay tests and irregular wave response tests?

2. How coarse can the grid of the CFD simulation be made such that it predicts motion accurately enough to predict the occurrence of wave impact events?

3. Is it more efficient to use CFD or another motion solver in a complete long-term simulation?

4. To what extent can ComFLOW be used to achieve accurate predictions of wave impact loads?

The objective of this project is to do a long term (CFD) simulation of the Tri-Floater in which it is possible to efficiently and accurately find the wave impact loads that the Tri-Floater experiences in a given sea state for multiple hours. If this objective is achieved this framework can be used in combination or as a replacement of model tests, which can be expensive and time-consuming. The framework can also be used with other floating offshore structures.

Research objectives have been set up to answer the research questions:

1. Create a CFD model of the Tri-Floater by:
    - creating the geometry from drawings,
    - using the proper parameters such that it is realistic and
    - creating a grid and domain on which the simulations can be performed.
2. Creating an aNySIM model from earlier models used at GustoMSC.
3. To perform a motion analysis by:
    - carrying out free decay test CFD and aNySIM simulations and
    - comparing the results to model tests done by MARIN.
4. Perform irregular wave tests to:
    - compare 3D CFD and aNySIM simulations with model tests response data and
    - investigate the calculation times in the different methods.
5. Develop and test a methodology in which it is possible to:
    - perform a long simulation of at least 3 hours,
    - find events in which wave impact might occur and
    - retrieve wave impact loads during those events.

1.5. THESIS STRUCTURE

This thesis work is divided into several chapters. Each chapter has its own methodology, results and conclusion. Chapter 2 contains theoretical background on the subjects in this thesis. Chapter 3 provides information on the ComFLOW and aNySIM models and the decay test comparison. Chapter 4 analyses the motion of the Tri-Floater in CFD and aNySIM. Chapter 5 presents the method and results of the long-term wave impact simulation. Chapter 6 is the conclusion, all results are summarized, and recommendations are done for further investigation of the subject.

2

THEORY

This chapter contains information about the theory used in this research project.

2.1. HYDRODYNAMICS

2.1.1. DAMPING AND PERIOD DURING DECAY TESTS

To calculate the damping, the logarithmic decrement method is used. With this system only linear damping is taken into account. It is better to use a different system which takes quadratic damping in to account. In this project the motion analysis is not the main goal, therefore, the logarithmic decrement method will suffice.

The general motion of a decay motion test in can be written as shown in Equation (2.1).

$$a_{\varphi\varphi}\ddot{\varphi} + b^{(1)}_{\varphi\varphi}\dot{\varphi} + b^{(2)}_{\varphi\varphi}\dot{\varphi}|\dot{\varphi}| + c_{\varphi\varphi}\varphi = 0 \tag{2.1}$$

In this equation φ is the amplitude, $a_{\varphi\varphi}$ is the total mass of the ship, which consists of the ship mass and the added mass in the motion φ. Coefficients $b^{(1)}_{\varphi\varphi}$ and $b^{(2)}_{\varphi\varphi}$ are the linear and quadratic damping coefficients respectively. The restoring coefficient is $c_{\varphi\varphi}$.

When only assuming linear damping, coefficient $b^{(2)}_{\varphi\varphi}$ is zero. A solution for the equation of motion is then given in Equation (2.2).

$$\varphi(t) = \exp\left(-\frac{b_{\varphi\varphi}}{2a_{\varphi\varphi}}t\right)(c_1 \cos\omega_n t + c_2 \sin\omega_n t) \tag{2.2}$$

In this equation ω_n is the natural frequency, c_1 and c_2 are the constants.

To find the damping percentage it makes sense to investigate the $\exp\left(-\frac{b_{\varphi\varphi}}{2a_{\varphi\varphi}}t\right)$ term. This exponential term is the cause of the damping in the motion during the decay test. A plot is made to visualise the damping, because of the exponential behavior the plot has a logarithmic y-scale. The peaks of the time series are plotted, to add accuracy the local minima are also plotted. However, the local minima are flipped in sign, such that they are also a maxima. In a logarithmic plot, these values should lay on a linear line. A linear fit is made with Equation (2.3).

$$e_i^{\varphi} = \delta i + b \tag{2.3}$$

In this equation φ_i is the amplitude of the i-th peak. The coefficient δ is the slope and is important for the damping.

The percentage of critical damping β_{crit} is related to the slope as given in Equation (2.4).

$$\beta = \frac{b_{11}}{B_{\text{critical}}} = \frac{-\delta}{2\pi} \tag{2.4}$$

In this equation B_{critical} is the critical damping. In this research project it is sufficient to know the percentage of critical damping β. With this percentage known it is possible to compare between different decay tests.

2.1.2. IRREGULAR WAVES

To get a more realistic wave pattern than regular waves, irregular waves will be used to represent the sea surface. In this project only one direction will be used for simplicity (and computational effort), a more realistic sea surface is obtained when multiple directions are included.

When looking at irregular waves, a long time frame is investigated, to do a proper analysis, it is useful to look at statistics. The average wave period is introduced as T_s, which is the average period of all wave crests. The peak wave period is T_p, this is the wave period with the highest wave energy. One of the statistics is the mean wave height, this is obtained by grouping wave heights in intervals and then finding the number of waves in these intervals. The sum of the average wave height in the interval multiplied by the number of waves in the interval and divided by the total number of waves gives the mean wave height. The most common statistic is the significant wave height H_s, which is the average of the highest 1/3 of the waves in a wave record. This is often used because there is a correlation between the highest 1/3 of waves and the visually observed wave height.

WAVE SPECTRA

There are two common ways to analyse a wave pattern, via the time domain and via the frequency domain. As wave pattern signals can become very long, the time domain can prove hard to analyse. A more common method is to use frequency domain analysis, where energy density spectrums are investigated.

To go to the frequency domain, the Fourier transformation is used. When it is applied, the amplitude is given as ζ_{a_n} as a function of ω. The wave amplitude is expressed as a wave spectrum $S_\zeta(\omega_n)$, it is defined as:

$$S_\zeta(\omega_n) \cdot \Delta\omega = \sum_{\omega_n}^{\omega_n+\Delta\omega} \frac{1}{2}\zeta_{a_n}^2(\omega) \tag{2.5}$$

In this equation $\Delta\omega$ is an interval, together with $S_\zeta(\omega_n)$ multiplied by the density and gravitational constant this can be interpreted as the energy in the interval. The interval and the spectral density is visualised in Figure 2.1.

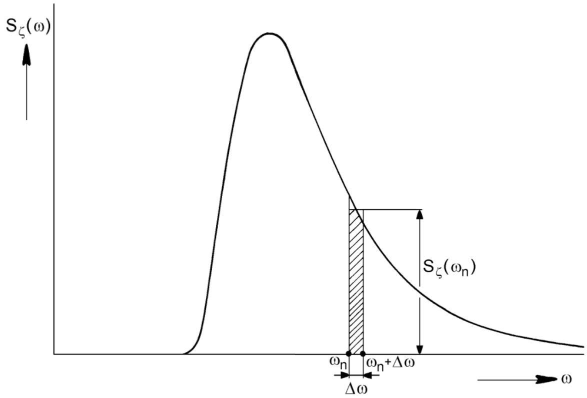


Figure 2.1: The definition of the spectral density. Image taken from [5].

When the limit of $\Delta\omega$ goes to zero, the wave energy density spectrum S is found:

$$S_\zeta(\omega_n)\,\mathrm{d}\omega = \frac{1}{2}\zeta_{a_n}^2 \tag{2.6}$$

From the energy density spectrum it is possible to obtain many parameters, the significant wave height and the period can easily be found.

It is useful to compare the energy density spectrums to theoretical spectrums. The two most important wave spectrums are the Joint North Sea Wave Project (JONSWAP) and the Bretschneider (Two-Parameter Pierson-Moskowitz) wave spectrums. In this project the Bretschneider spectrum is used, this spectrum is suited for open seas, when the sea is fetch-limited, or a limited depth, the JONSWAP spectrum is used.

The Bretschneider spectrum is described by the following equation:

$$S(\omega) = \frac{5}{16}\frac{\omega_p^4}{\omega^5} H_s^2 \exp\left(-\frac{5}{4}\left(\frac{\omega_p}{\omega}\right)^4\right) \tag{2.7}$$

In this equation the ω_p is defined as $2\pi/T_p$ and H_s the significant wave height. These are the two parameters used to define a wave spectrum. This equation can be used to compare time series to a theoretical spectrum.

In this project the Bretschneider spectrum is mostly used in the inversely, to construct a time series from a known spectrum. With this method it is possible to use parameters T_p and H_s to define a spectrum, which can be used as a inflow boundary condition in the simulations.

To reconstruct a time series from the spectrum, an assumption is made that the wave field can be described by a linear superposition of cosine functions. The equation for the wave amplitude ζ is given as:

$$\zeta(x,t) = \sum_{n=1}^{N} \zeta_{a_n} \cos(k_n x - \omega_n t + \epsilon_n) \tag{2.8}$$

In this equation ζ_a is the amplitude, k the wave number, ω the wave frequency and phase ϵ. These parameters are required for every wave component n with a total amount of wave components N. Firstly, a range of frequency ω is defined, between a low- and high cutoff frequency, with N components. Typically the low cut-off is chosen at $0.25\,\mathrm{rad\,s^{-1}}$ and the high cut-off $2.5\,\mathrm{rad\,s^{-1}}$, outside this range the energy can be considered zero. To calculate ζ_a, it is required to rewrite Equation (2.5) as follows:

$$\zeta_{a_n} = 2\sqrt{S_\zeta(\omega)\cdot\Delta\omega} \tag{2.9}$$

The phase angle ϵ is chosen from an uniformly distributed set of random numbers between 0 and 2π. This is done to create a realistic wave field in time, when enough components are used, the created time series should be identical to spectrum computed from the input parameters [5]. An overview of this method is given in Figure 2.2.

RESPONSE IN IRREGULAR WAVES

The analysis of the wave spectrum is also usable when looking at the motions of bodies in water. The response of a structure can also be expressed in an energy spectrum. It is useful to look at the response of the structure to the wave spectrum, this is done with the transfer function:

$$S_x(\omega) = \left|\frac{x_a}{\zeta_a}(\omega)\right|^2 S_\zeta(\omega) \tag{2.10}$$

In this equation x is the degree of freedom, energy density spectrum $S_x(\omega)$ is found in a similar method as Equation (2.6). The term $\frac{x_a}{\zeta_a}(\omega)$ is known as the Response Amplitude Operator (RAO).

2.2. COMFLOW

In this project ComFLOW is used to do the Computational Fluid Dynamics (CFD) simulations. The ComFLOW simulation method is a result of work by [6] at the Rijks Universiteit Groningen. ComFLOW is suited for free surface flows because of its features like the improved Volume of Fluid method (iVOF), generating and absorbing boundary conditions (GABC) and local grid refinement. In this section some principles behind the features will be presented. This description will be conceptual, the mathematical description will be omitted as this is discussed in many other works. However, it is useful

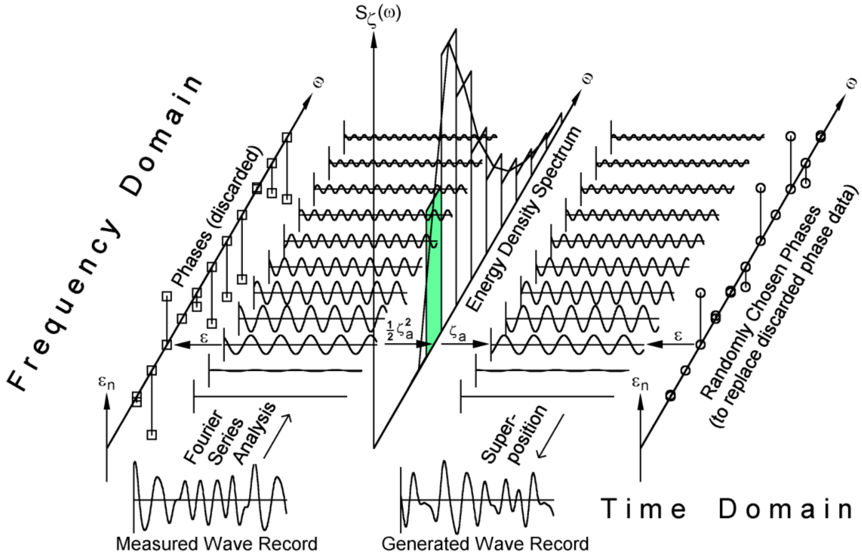


Figure 2.2: Wave record generation, this image shows the methodology of the spectrum generation and the linear superposition. Image taken from [5].

to know which methods are applied and what it's strengths and weaknesses are. The information given in this chapter also shows the motivation behind using ComFLOW for the purposes of this research project.

ComFLOW was originally designed to study the sloshing fuel on board of spacecraft. For this application an accurate description of the free surface was required [7]. This methodology was extended to offshore applications, for example, green water loading, impact loading and sloshing tanks. In the years following the ComFLOW methodology was improved with many useful techniques, including numerical techniques and also interactive moving bodies. These developments were made in combination with several industry partners in joint industry projects. Currently, the application of ComFLOW is mostly academically.

2.2.1. MATHEMETICAL MODEL

ComFLOW is a Navier-Stokes equation solver, it uses a 2nd-order upwind spatial discretisation on a staggered Cartesian grid, combined with Adams-Bashforth time-stepping. The Cartesian grid method is a relatively simple geometrical framework which works well for free surface flows. The pressure is located in the cell center and the velocities are located in the middle of the cell faces. Cell labeling is used as shown in Figure 2.3. This cell labeling is done on every time step, the Navier-Stokes equation are then only solved in the cells containing fluid.

After solving the Navier-Stokes equations numerically, the free surface is calculated. The free surface is initially calculated, and displaced every time step by the new calculated

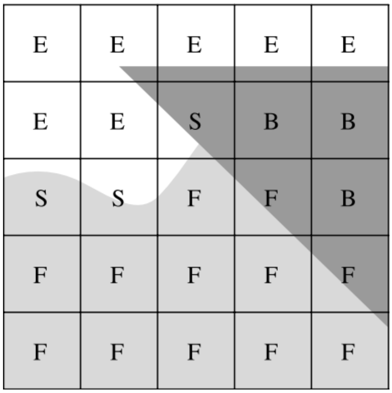


Figure 2.3: Cell labeling method used in ComFLOW. (E)mpty, (B)oundary, (S)urface and (F)luid cells are shown. Image as given in [6].

velocity field. To model the free surface displacement a volume-of-fluid (VOF) function is used. This function can be interpreted as a fill ratio of the cell. Another ratio to indicate the amount of the cell that is available for fluid is used, called the volume apertures. Edge apertures are used to indicate the part of the edge that is blocked by a solid.

The original VOF function had problems with isolated droplets and mass losses, this has been solved by using a local height function. An example of the local height function method is given in Figure 2.4. In this method, the horizontal or vertical height is calcu-

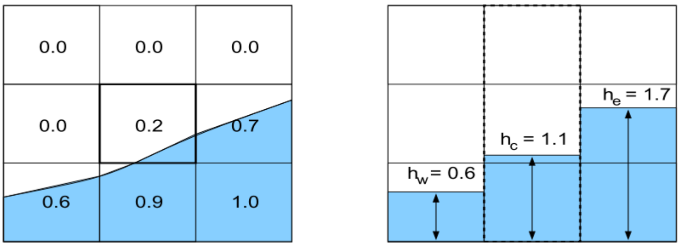


Figure 2.4: Example of the construction of the local height function. Image as given in [7].

lated for every row or column, depending on the orientation of the free surface. This is done in a 3 x 3 x 3 block around the central surface cell at every time step. Depending on the velocity field, fluid is transported from the donor cell, to the acceptor cell. This method makes sure that no fluid is lost.

2.2.2. INTERACTIVE MOVING BODIES

In ComFLOW it is possible to simulate interactive bodies. The model used in ComFLOW is the quasi-simultaneous coupling. First, the basics of fluid-structure coupling will be explained, then the weak coupling method is explained as this gives insight in to the method. Finally, the quasi-simultaneous method is discussed. An overview will be given, the mathematical details are given in [8, 9].

An object floating on the surface of a fluid is a coupling between the object and its physical properties and the fluid. This coupling is located along a surface that is shared by both the object and the fluid, this interface is specified as Γ. Along this interface there are coupling conditions due to the fact that the physics have to be the continuous along the interface. These conditions can be split into the kinematic condition and the dynamic condition, The kinematic condition can be explained by the fact that when the object and the fluid are in contact the velocity and the acceleration should be equal for both the object and the fluid. The dynamic condition is based on equilibrium of forces, the fluid exerts a force on the object, and the object exerts an opposing force on the fluid.

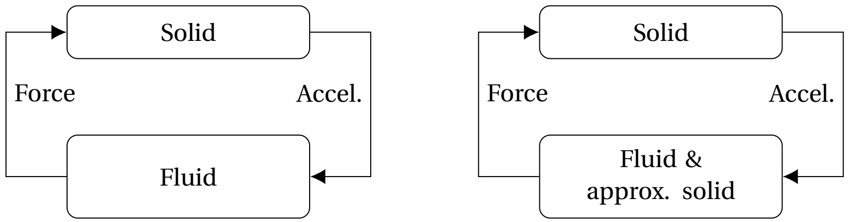
The objects dynamics are modeled by an equation that describes the acceleration of the body as a result of the forces exerted on the object by the fluid, in this equation the mass properties of the body are included by the mass operator. In this equation it is also possible to add external forces and damping, due to for example mooring lines, which will be used in this research project. The forces on the object can be found by integrating the pressure exerted by the fluid along the interface.

The fluid dynamics part of the interactive motion problem adds a boundary condition to the Navier-Stokes equations that are solved by ComFLOW. When accelerating or decelerating an object in fluid, the fluid also has to move, because the fluid and the object cannot occupy the same volume. The inertia that is added to the system can be seen as the added mass of the system.

The object dynamics and the fluid dynamics give two equations with two unknowns. These equations can be solved in two different methods, in fluid-structure interaction field these methods are called the monolithic approach and the partitioned approach. In the former, the fluid and the object equations are solved simultaneously with a single solver, and in the latter they are solved separately in two separate solvers.

Weak coupling is a partitioned approach. The load field is determined by the fluid dynamics by integrating the pressure forces, this moves the solid body, and this motion is then transferred to the fluid. A loop is created and when the mathematical definitions are followed, the convergence condition is found. The convergence condition can be related to the ratio between the added mass and the body mass. When convergence is a problem, for example when the added mass is not very small compared to the body mass, under relaxation can be used. Under relaxation is a method to keep the computation stable, the calculated solution for the next iteration is a linear combination of the solution at the previous iteration, and the calculated solution. When under relaxation is used a lot of additional iterations are required, which is bad for the efficiency.

A more efficient method is the quasi-simultaneous coupling method used by ComFLOW. An approximation for the full dynamics operator is made. This approximation is an anticipation to the response of the body, and its simple enough to be used in the boundary condition for the Navier-Stokes equations. Using this approximation it is possible to use the monolithic approach as much as possible, the approximate solution for the solid is solved together with the fluid equations. The difference between the weak and quasi-simultaneous coupling is shown in Figure 2.5. The benefit of using the quasi-simultaneous method becomes clear when the added mass is large, or the added mass is changing rapidly. The efficiency and the stability increase is shown in [8].



(a) Weak Coupling **(b)** Quasi-simultaneous

Figure 2.5: Weak and quasi-simultaneous coupling difference. Image reproduced from [9].

The use of the quasi-simultaneous method saves a lot of computational time in this research project. The method is the standard method used by ComFLOW and is thus easy to set up.

2.3. ANYSIM

In this project aNySIM is used to calculate the Tri-Floater motions in the time domain. aNySIM is a time-domain software package developed by the the Dutch Maritime Research Institute (MARIN). The software package is widely used in the maritime and oil and gas sectors. It is used for mooring calculations, dynamic positioning systems, crane operations, loading and offloading and much more. In aNySIM the calculation of the motion is based on the Cummins equations. In this section the theory will be briefly discussed, the information is based on the manual released for aNySIM.

2.3.1. THEORY

The basic equation used in classic ship motion theories is given as:

$$\sum_{j=1}^{6} \left(M_{kj} + a_{kj}\right)\ddot{x}_j + b_{kj}\dot{x}_j + c_{kj}x_j = F_k \tag{2.11}$$

In this equation M is the mass, a is the added mass, b the damping, c the hydrostatic restoring coefficient, F the external force and kj are the modes of the motion. This equation is simple to understand and based on a mass spring system. However, aNySIM is able to include the dependence of frequency in the coefficients and also non-linear restoring force characteristics. This is done by using an adjusted equation, based on the impulse response theory:

$$\sum_{j=1}^{6} \left(M_{kj} + m_{kj}\right)\ddot{x}_j + \int_{-\infty}^{t} R_{kj}(t-\tau)\dot{x}_j(\tau)\,\mathrm{d}\tau + C_{kj}x_j = F_k(t) \tag{2.12}$$

Where m_{kj} is the frequency independent added inertia matrix, R is the retardation function matrix, and C is the hydrostatic restoring forces matrix. The external force (varying in time) $F(t)$ is used, this includes the first and second order wave forces and the mooring system.

To solve Equation (2.12), a hydrodynamic database is required. The database contains the frequency dependent added mass and damping, and the first- and second order wave forces. The hydrodynamic database is created with three dimensional radiation-diffraction theory with WAMIT internally by GustoMSC for the Tri-Floater. To account for the viscous effects quadratic damping has been used, these have been calculated by CALMOT in the same project.

The mooring system used in the aNySIM model is based on lumped mass lines. In this method the mooring lines are modelled by a line split in to multiple elements connected with nodes, the ends are fixed to the Tri-Floater and to the earth-fixed system. The nodes are connected by a spring stiffness and damping. In this model design by GustoMSC the motions were compared and tuned according to the results of model tests.

2.4. Alternative Methods

For completeness, two different methods will be introduced in this section.

2.4.1. Numerical Wave Basin

In [10] a method is given to do an efficient long-duration simulation of hull motions in a realistic wavefield. Here, a semi-submersible with four mooring lines is simulated in a 3-hour extreme sea state. The method is divided into three components. A potential-flow-based nonlinear wave model is applied to create the incoming wavefield. A 2D computation based on Hamilton's principle is done. The motion is simulated with STAR-CCM+. This method also includes a coupling to a mooring model. In this CFD method, attention is given to capture the viscous effects adequately, and prism layers are applied close to the body. A mesh sensitivity study is done to find the optimal settings. Simulations are done on a cluster; the computational time of the 3-hour sea state is approximately 3.5 days using 640 processing cores. Parallelization in time could reduce the time to 2.5 days but would require more pre-and post-processing. The results are checked against model tests and achieve good agreement.

This paper is interesting as its goal is similar to this research project, to do long term CFD simulations. The long-term simulation is achieved in this work, however, at a high computational cost. The difference with our project is that the end goal is not the same. The main objective of this research project is to create a methodology to find the wave impact loads in a 3-hour simulation, not a very accurate simulation of the motions. However, this paper still includes interesting workflows and results that can be of use in the project.

2.4.2. Event Matching

Another interesting take on long term analysis is done by DNV GL in [11]. In this research, event matching is performed. A Monte Carlo simulation of the wave climate is done with undisturbed linear waves as an input. The method then compares and matches the linear waves with a CFD result from a database to include the nonlinear wave properties. An overview of this method is given in Figure 2.6.

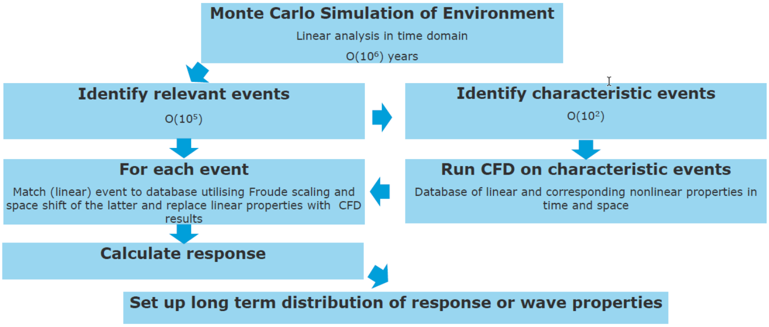


Figure 2.6: An overview of the Event Matching method taken from [11].

This method of event matching seems like a very promising method. The linear wave Monte Carlo simulation can simulate many waves, and the selected events have been used as an input for ComFLOW. The main difference between the work done in this article and this research project is that the motion of the structure is taken into account in this research project. In this article, the focus is more on the waves and then using these waves as an input to a structural model to find the loads.

3

Decay Test Validation

This chapter consists of sections on the model tests and decay test simulations performed in this research project. The model tests are performed by the MARIN, information from the model test that is useful for this research project is given. The section on the decay tests contains information about the simulations done in ComFLOW and the linear solver aNySIM.

3.1. Model Tests

The Dutch Maritime Institute did the model test for GustoMSC's Tri-Floater in 2013. The model that is used in the model tests is built at a scale of 1:50. In this thesis work only relevant information will be given. More information on the model tests can be found in the final text report 24796-3-OB.

3.1.1. Tri-Floater model

The Tri-Floater is build at a scale of 1:50. The materials used for the hull consist of PVC, carbon fibre and foam. The mooring system of the Tri-Floater consists of a brass chin, a spring at the anchor point for axial stiffness correction and a load cell at the fairlead point to measure loads.

On the Tri-Floater structure, a wind turbine model is placed. This model is based on the standard NREL 5 MW wind turbine. The wind turbine particulars are not interesting for this project. The mass distribution of the model is different from the turbine on the Tri-Floater, this is included in the parameters for mass and other parameters. The model is made such that the model parameters match the specified parameters, taking into account the scaling factors, as closely as possible. To do a proper comparison between the CFD simulation and the model tests, the parameters as built by MARIN are used, not the parameters specified for the Tri-Floater. The parameters will be given in the section on simulation model description.

3.1.2. OFFSHORE BASIN

The model tests are done in MARIN's Offshore Basin. In this basin, it is possible to accurately simulate environmental conditions in both wind and waves. To generate waves in the basin, flap-type wavemakers were used. The wavemakers are oscillating with varying frequencies and amplitudes, according to the sea state given by the JONSWAP wave spectra. Details on wind generation are not relevant for this work and are omitted from the report.

3.1.3. MEASUREMENTS

Numerous measurement equipment is present on the model. The measured data signals were sampled at a 100 Hz at model scale, this corresponds to a frequency of 14.1 Hz full-scale. In this project, the wave height is relevant, which is a resistance type wave probe placed at 4 different locations. Model translations and rotations are measured by a contact-less optical position system for 6 degrees of freedom motions.

3.1.4. EXPERIMENTS

In the series of model tests, two of the test series are relevant for this research project. The decay tests are performed by pulling the floater from its equilibrium position and released, this is done for all the modes of motion. From the decay tests, it is possible to determine the natural frequencies and damping for each degree of freedom separately.

The irregular wave experiments are started from a situation where the model is completely at rest, the signals are then set to zero such that the motions are relative to this zero point. The first 30 minutes of measurements are omitted from the results to prevent the transient phenomena from showing in the results. After these 30 minutes, the measurements are started for a 3-hour experiment. For this research project, only the waves only tests were used. These include various significant wave heights and wave periods.

3.2. DECAY TESTS

In this section, the experimental method, the experiments and the results are presented and discussed. The decay test will give insight into the modelling of the motions of the Tri-Floater. With the results, it is possible to fine-tune the simulation settings and the settings for the Tri-Floater itself, which includes the mooring settings and damping. This process will be iterative, and the settings will be improved after simulations until the model is working correctly. The goal of this part is not to do accurately model all the physics of the Tri-Floater, but to simulate the motions as efficiently as possible and gain insight in to the CFD simulation.

3.2.1. CFD SETUP

To create a grid in which the decay test simulations are performed, the ComFLOW function for grid refinement is used. This setup is possible as ComFLOW works with a Cartesian grid system. With this grid refinement, it is possible to have a very rough grid where the Tri-Floater is not, such that fewer cells can be used for the simulation.

In the decay test simulations, there are at least four sub-grids. The largest grid size for the cells far away from the Tri-Floater are 16 m, the smallest cells are either 1 m or 2 m. The grid is made such that the motion of the Tri-Floater is inside the sub-grid with the smallest size.

The grid is given in Figures 3.2 and 3.3. The grid 320 m by 320 m with a water depth of 96 m. The boundaries are chosen. There are two Generating and Absorbing Boundary Conditions (GABC) boundaries for the boundary conditions: inflow at $-X$ and outflow at $+X$. The other boundaries have no-slip condition boundaries. The wave model is the Airy wave model, with a height of 0 m because, in ComFLOW, it is required to have a wave condition. The simulation time step dt is driven by the Courant–Friedrichs–Lewy condition (CFL) condition. The CFL has to be between 0.2 and 0.5, as advised in the ComFLOW manual.

The mooring system of the Tri-Floater in the CFD simulation is modelled as a spring system. The spring parameters are taken from internal work at GustoMSC.

The Tri-Floater is placed such that the centre of mass (CoM) coincides with the origin of the simulation. This means that when the position of the Tri-Floater is (0,0,0), the CoM is placed on the water surface. An overview of the axis layout is given in Figure 3.1. From these axes, it is possible to define the roll, pitch and yaw degrees of freedom. Roll rotation, the Tri-Floater rotates around the x-axis, and positive roll rotation means that the starboard side of the Tri-Floater is going down. Pitch rotation, the Tri-Floater rotates around the y-axis, and positive pitch rotation means that the 'bow' of the Tri-Floater is going down. Yaw rotation, the Tri-Floater rotates around the z-axis, and positive yaw rotation means that the 'bow' of the Tri-Floater is turning to port side.

3.2.2. aNySIM setup

The aNySiM simulations are set up based on earlier work done internally at GustoMSC. The hydrodynamic data from that project is used in this project. It also uses quadratic damping; this damping is scaled from a basic semi-submersible scale. The mooring system consists out of three lumped mass catenary lines. Simulations are done with a time step of 0.05 s.

The initial amplitude is set differently than with ComFLOW. In aNySIM, a force is placed at the centre of mass. In the first 400 seconds, the force is build-up; at 600 seconds, the motion should be stable at the desired amplitude. Shortly after that, the force is removed, and the Tri-Floater will start moving towards equilibrium.

3.2.3. Data analysis

Opening Time Series

The data analysis starts with opening the three different data types. A parent object is created in python for this purpose, the timeSeries object is initialized with empty arrays and a label. Separate child objects are created to fill the arrays, as the model test, aNySIM and ComFLOW do not share the same type out output file.

For all time series a wave height at the origin is stored, and if applicable the motions for

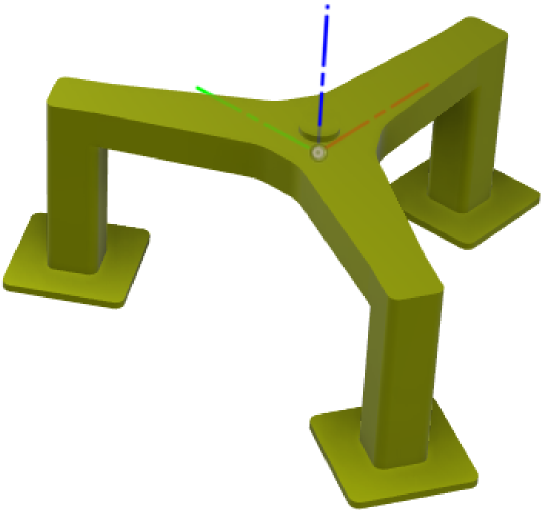


Figure 3.1: The axis of the Tri-Floater, with the origin at the center of mass. The red axis is x, the blue axis is z and the green axis is the y-axis.

each degree of freedom. With all simulation and model types in similar objects, with the same notation, it is possible to compare all the time series and plotting the results is trivial.

Calculating Period and Damping

A python function `damping_percentage` is made to compute the damping percentages and plot the fit lines. The function takes multiple time series objects as input, this includes the time and the motions in all degrees of freedom. Other important parameters to let the function work properly are explained below.

The function uses the `scipy` function `find_peaks`, which works by finding local maxima by comparing neighboring values. As our time series signal is not perfect, local maxima might occur where they are not expected. To overcome this, the minimum distance between the maxima is set, this is possible because the maxima are expected to be separated by one period.

The height offset is also used, the `find_peaks` function uses the height as a minimum height for the peak. In the `damping_percentage` function this means that the motion will oscillate around 0.

To find the minima of the time series, the `find_peaks` function is used with the negative motion, now the minima are the maxima and the peaks are calculated.

The function also uses a lower bound input, this contains the first point in the peaks that the fit has to include. This is done because the first oscillation has a large amplitude,

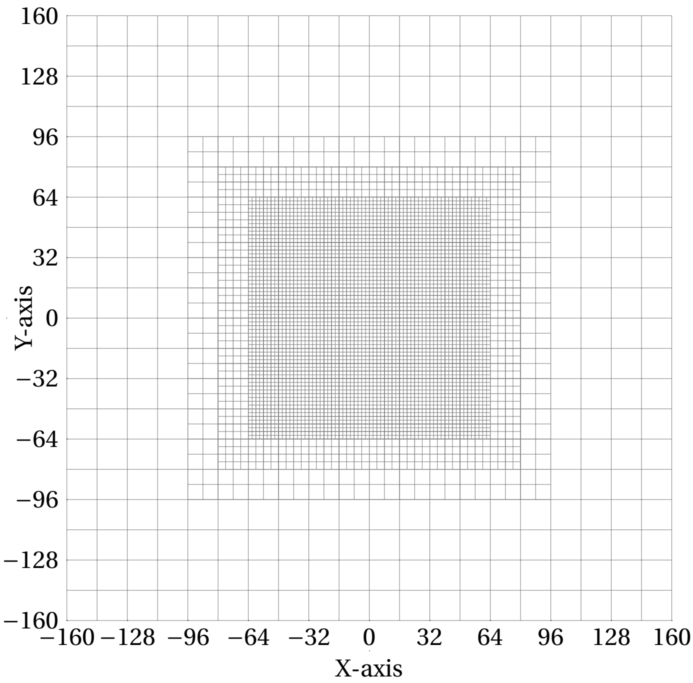


Figure 3.2: Decay test grid setup for the X and Y axis, the grid size around the Tri-Floater is 2 m.

which is an outlier in the fit.

The local minima are flipped around 0, such that their amplitudes are positive. These are then concatenated to the local maxima and sorted based on the heights.

For each time series, the motion is plotted with the peaks included, such that it can be visually assessed that the peaks are properly placed, if this is not the case the parameters need to be tweaked. When the peaks are in the proper location, a new plot is made with a logarithmic y-scale for the heights of the peaks. The x-scale is the oscillation number, because minima and maxima are used the peaks can also be at .5 locations.

From `scipy` the function `curve_fit` is used, which uses a least squares function to fit a chosen function to data. As explained in Section 2.1.1, a linear line function is used, which has the oscillation number for x and the logarithm of the peak amplitudes as the y input.

The result of the fit, the parameters, are used to calculate the critical damping percentage and to plot the fit line and the peaks in the logarithmic plot to see how good the fit is. For each degree of freedom the peaks of multiple time series (including model tests) and fit are plotted in one plot, such that they can be compared.

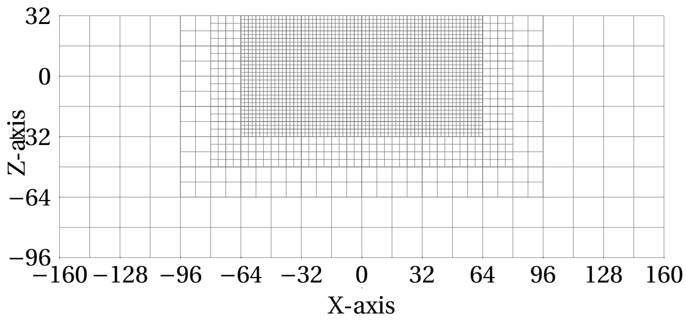


Figure 3.3: Decay test grid setup for the X and Y axis, the grid size around the Tri-Floater is 2 m.

3.2.4. RESULTS

The results of the decay tests are first compared in the time domain. The time series results are normalized, the amplitude is divided by the maximum amplitude in the decay test. This is done because the Tri-Floater is released from different amplitudes. Damping and the period of the motion are also calculated for all degrees of freedom with the method described in the previous section. The results will be presented for each degree of freedom in a separate section. These will then be discussed together in a section, to prevent repetition.

SIMULATION STATISTICS

The aNySIM simulations are done within minutes.

The CFD simulations are more costly, even in this simulation, with a large grid size. There are approximately 80000 fluid cells in the grid with 2 meter grid size, the total number of cells is approximately 150000. The decay simulations have an approximate CPU time of 18 hours for a simulation time of 300 seconds, the exact time varies per simulation. This CPU time is the time that it would take on 1 thread, if multi-threading is used the real time taken is greatly reduced.

SURGE

The results of the time series simulations of the surge direction are presented in Figure 3.4. In these results it can be seen that the motions of both the CFD and the aNySiM simulations are different from the model test. In the ComFLOW simulation the initial deviation was less, if the same deviation would be used as in the model test, the fine grid would have to be larger, while not adding much information.

The periods are calculated to be 66.6 s, 80.8 s and 76.9 s for the model test, ComFLOW and aNySIM respectively.

The initial simulations without damping had issues when the amplitude was small. Linear damping of $25 \times 10^3\,\mathrm{N\,s\,m^{-1}}$ was added for this reason, and to match the larger amplitude deviations.

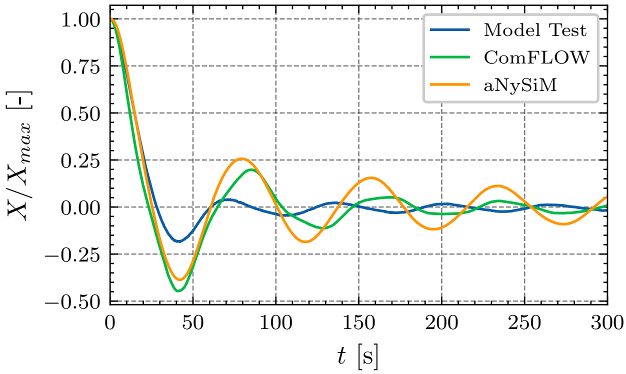


Figure 3.4: The time series result for surge.

In Figure 3.5 the logarithmic decrement method is applied. The damping percentages are found to be 5.5%, 15.8% and 5.4% for the model test, ComFLOW and aNySIM respectively.

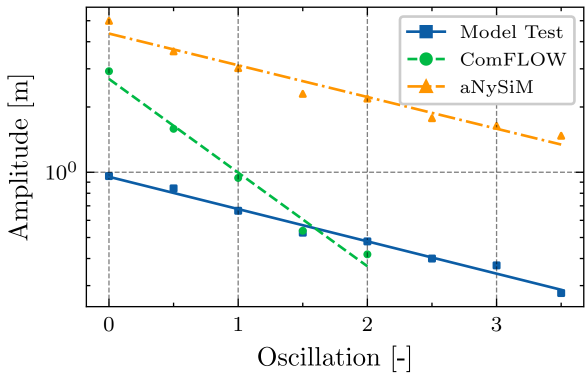


Figure 3.5: The plot of the logarithmic decrement. The damping percentages are found to be 5.5%, 15.8% and 5.4% for the model test, ComFLOW and aNySIM respectively.

SWAY

The results of the time series simulations of the sway direction are presented in Figure 3.6. The results of sway are similar to surge, the same patterns are found.

The period of the motion is found to be 75.9 s, 80.5 s, 77.0 s for the model, ComFLOW and aNySIM respectively.

In Figure 3.7 the logarithmic decrement method is applied. Linear damping of $25 \times 10^3\,\mathrm{N\,s\,m^{-1}}$ is added to the ComFLOW model to make sure the motion is stable on small oscillations. The damping percentages are found to be 6.6%, 10.7% and 4.8% for the model test, ComFLOW and aNySIM respectively.

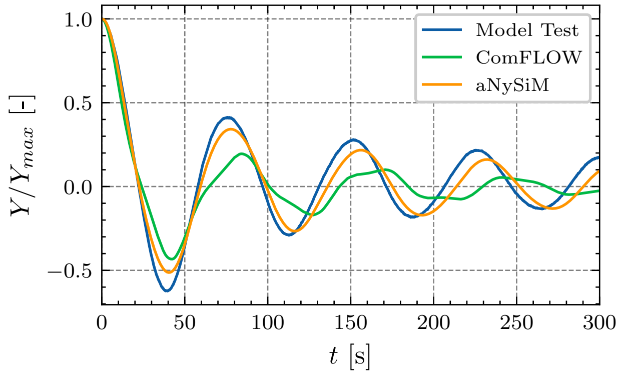


Figure 3.6: The time series result for sway.

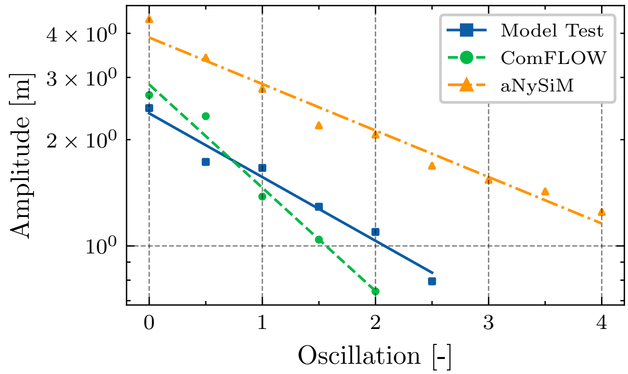


Figure 3.7: The plot of the logarithmic decrement. The damping percentages are found to be 6.6%, 10.7% and 4.8% for the model test, ComFLOW and aNySIM respectively.

HEAVE

The results of the time series simulations of the heave direction are presented in Figure 3.8. In the time series plot it can be seen that the shape of the motion is generally well-matched.

First, the natural periods are compared. The period of the motion of the Tri-Floater in the model test is 16.5 s, 16.1 s in ComFLOW and 16.5 s in aNySiM.

In Figure 3.9 the logarithmic decrement results are presented for heave. Because of the similar damping percentages no damping is added to the system for heave. The damping percentages are found to be 4.2%, 4.3% and 4.2% for the model, ComFLOW and aNySiM respectively.

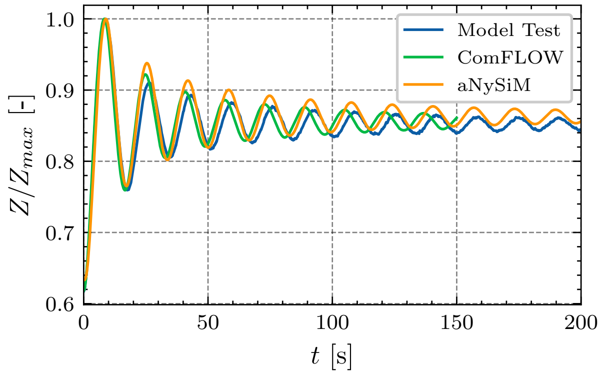


Figure 3.8: The time series result for heave.

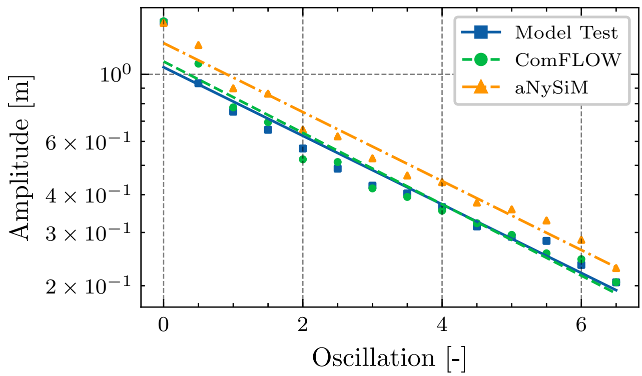


Figure 3.9: The plot of the logarithmic decrement. The damping percentages are similar as the lines in this graph are almost parallel.

ROLL

The time series results for roll is presented in Figure 3.10. The period of the motion of the Tri-Floater in the model test is 34.5 s, 30.2 s in ComFLOW and 33.7 s in aNySiM.

In Figure 3.11 the logarithmic decrement results are presented for roll. In the ComFLOW simulation, linear damping has been added such that the results of CFD match the aNySiM simulation more closely, 1×10^6 N s deg^{-1} is added. The damping percentages are found to be 4.3%, 4.0% and 3.6% for the model, ComFLOW and aNySiM respectively.

PITCH

The time series results for pitch is presented in Figure 3.12. In these results the model test is missing, due to the coupling effects in the model test. There is also no information available on the period and damping, therefore only the aNySIM results are compared with ComFLOW results.

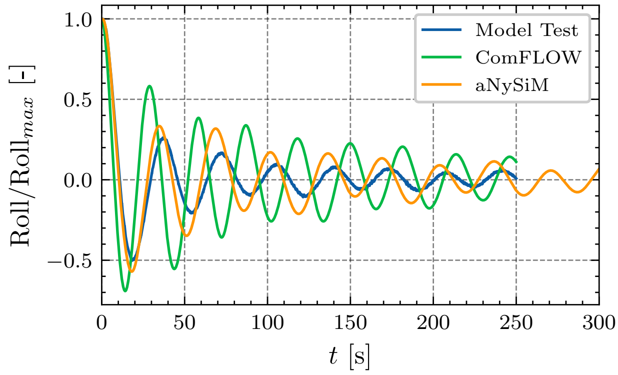


Figure 3.10: The time series result for roll.

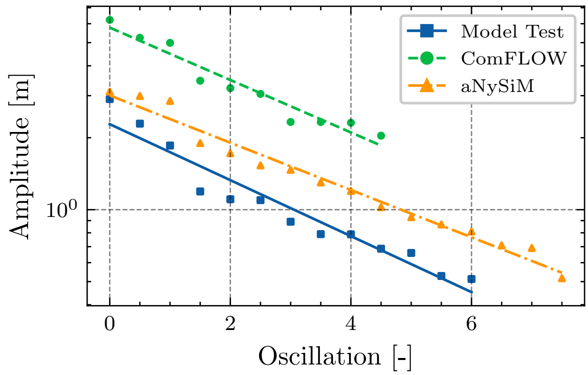


Figure 3.11: The plot of the logarithmic decrement. The damping percentages are found to be 4.3%, 4.0% and 3.6% for the model test, ComFLOW and aNySIM respectively.

The periods of motion are found to be 29.7 s and 33.9 s for ComFLOW and aNySIM.

The results of the logarithmic decrement are presented in Figure 3.13. In the ComFLOW simulation, linear damping has been added such that the results of CFD match the model tests more closely, 2×10^6 N s deg^{-1} is added. The damping for the ComFLOW simulation is found to be 5.3% and for aNySIM 4.5%.

YAW

The time series results for roll is presented in Figure 3.14. The periods of motion are found to be 59.9 s, 61.9 s, 66.5 s for the model, ComFLOW and aNySIM respectively.

In Figure 3.15 the logarithmic decrement results are presented for roll. In the ComFLOW simulation, linear damping has been added such that the results of CFD match the model tests more closely, 0.7×10^6 N s deg^{-1} is added. The damping percentages are found to be 5.8%, 5.0% and 5.97% for the model, ComFLOW and aNySiM respec-

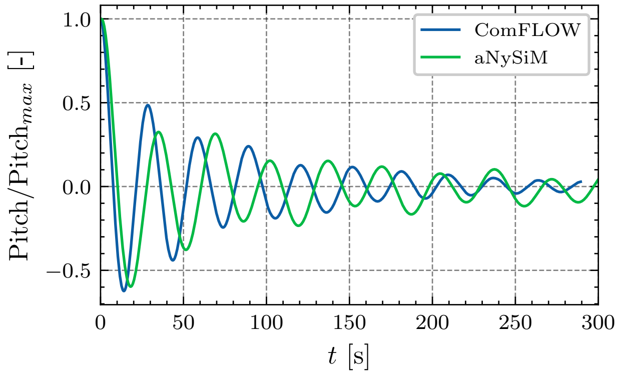


Figure 3.12: The time series result for pitch.

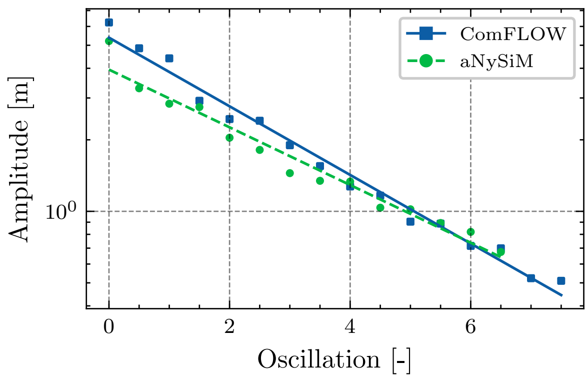


Figure 3.13: The plot of the logarithmic decrement. The damping percentages are found to be 5.3% for ComFLOW and 4.5%.

tively.

3.2.5. DISCUSSION

In this section the results will be discussed. For convenience a table is included where the results are compared with the use of ratios in Table 3.1. In this table the model period and the damping is compared to the model test. The difference is given as a percentage increase or decrease.

It can be seen that the shape of surge and sway is not the same as the other degrees of freedom, it is suspected that this shape difference comes from the coupling effects that are seen in the motions. The coupling effect is seen in the model test, and also both the aNySiM simulation and the ComFLOW simulation. Surge has a coupling with pitch, and sway with roll. However, it is hard to compare the coupling effects in the time series, because they have different initial starting positions.

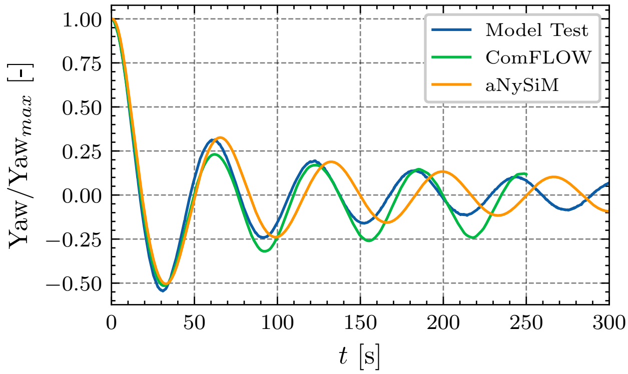


Figure 3.14: The time series result for yaw.

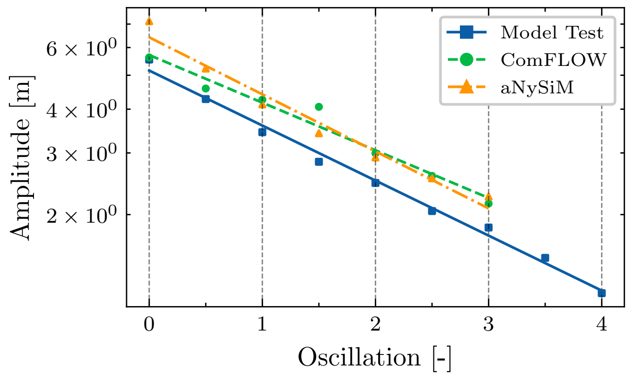


Figure 3.15: The plot of the logarithmic decrement. The damping percentages are found to be 5.8%, 5.0% and 5.97% for the model, ComFLOW and aNySiM respectively.

PERIOD

It can be seen that there is a large variance in the motion periods, in all degree of freedom there are differences, however in surge the largest deviation is found. However, it is interesting to see that both ComFLOW and aNySIM have similar periods. In the simulations surge and sway have a similar period, while in the model test the period of surge is 66.6 s against 75.9 s. It is unclear why this occurs, going in to more detail is outside the scope of this report.

For the other degrees of freedom, the difference is smaller. The difference is suspected to be due to the differences in mooring system. ComFLOW uses a linear spring system and aNySIM a lumped mass system. The latter should be more accurate and this is also seen in the results, with the exception being yaw. It is possible to change the results of the simulation by changing the spring matrix. This should be relatively simple, however, in this project the focus is on the simulation and not so much on the details of the mooring system. Therefore, the spring matrix is left unadjusted, and it is accepted that there is a

Table 3.1: Summary of the results. The ratios compared to the model test is given in this table. Pitch is excluded as there is no pitch model test data available.

	Period T_p			Damping β		
	Model [s]	ComFLOW	aNySIM	Model [%]	ComFLOW	aNySIM
Surge	66.6	+21%	+20%	5.5%	+190%	-2%
Sway	75.9	+5%	+2%	6.6%	+62%	-27%
Heave	16.5	-3%	+0%	4.2%	+4%	+0%
Roll	34.5	-13%	-2%	4.3%	-6%	-15%
Yaw	59.9	+3%	+11%	5.8%	-13%	+4%

difference in periods.

DAMPING

The variability in the results of the damping is larger than in the period.

In surge and sway the large motions are underdamped in the ComFLOW simulations, and problems occur when the amplitude of the motion becomes small, on the order of the grid size. To prevent these problems, linear damping is added to the motion system of the Tri-Floater, this solves the instability problems. However, this change introduces the problem that the smaller motions are overdamped in comparison to aNySIM and the model tests. This process is a balance between capturing larger motions and smaller motions. For this research project it makes sense to place the emphasis on matching the larger motions, as wave impact will occur in high wave sea states, where the oscillations are large. It can be seen that the surge and sway degrees of freedom still need adjustments, this could include changing the mooring system to a more realistic system. Another option is to introduce quadratic damping in to the motion model. For this research project however, the scope is limited to linear damping on the linear spring model due to time constraints.

In the other degrees of freedom, after adding slight linear damping, the difference is less and the shape of the motions are also similar. These degree of freedoms require less work, however, it should not be forgotten that there is coupling between the degrees of freedom, which means that changing the system in for example surge, this also has an influence in pitch.

A side note should be given for the calculation of the damping with the logarithmic decrement method. The damping percentage is calculated from the slope of the fitted line in the logarithmic plot, therefore it is sensitive to which peaks are included on in the fitting process and the amount of peaks included in the fit. This fact explains why the period and damping that are calculated for the model test are slightly different from the model test results given in the model test report. In this research project the starting point are kept constant, which means excluding the initial amplitude and for some simulations the second peak is also excluded to improve the fit quality.

GRID SIZE

To fully capture the physics, it would be required to implement a turbulence model in to the CFD simulation. Work on turbulence modeling in ComFLOW has been done in the past [12]. However, the goal of this research project is related to wave impact, in wave impact situations the viscous effects are negligible. Implementing turbulence in to the model could improve the CFD simulation results, as it might be possible to properly model the drag of the columns of the Tri-Floater, therefore improving the results especially in the surge and sway direction, and possibly also in the heave direction.

3.2.6. CONCLUSION

In this chapter decay tests done in aNySIM and in ComFLOW are compared to model tests. The research done in this chapter contribute to the sub-questions relating to the accuracy of the CFD simulation, and if it is possible to run the CFD simulation on a grid with a large grid size.

Simulations are done in aNySIM and ComFLOW. The results are compared to the model tests performed by MARIN, the period and the damping of the motion are investigated as these should give a good insight in the motion of the Tri-Floater.

The periods of the motions are matched quite well for most degrees of freedom, except surge. In the surge direction, the period in both ComFLOW and aNySIM is similar, but 20% higher than during the model test. The ComFLOW results for the period could be improved by tuning the linear spring matrix of the Tri-Floater. Either tuning the linear spring matrix or improving the mooring system to a more realistic model, like the lumped mass model used in aNySIM should be achieved if better motion analysis is the desired result.

The damping is matched well in heave and in the rotational degrees of freedom. In surge and in sway problems stability problems occur when the amplitude of the motion gets small, on the order of the grid size. Linear damping was added to the motion system for this reason, and also to match the damping of the rotational degrees of freedom. To improve the results, especially on surge and sway, it would be advised to add quadratic damping to the motion model and do more iterations to find if this improves the results.

When keeping the limitations given in this chapter are taken in to account, it can be concluded that the CFD simulation of the Tri-Floater in ComFLOW requires more work to properly match the model tests, but should be suitable for the next parts of the project.

4

Floater Motions in Waves

This chapter presents results on the irregular wave simulations. In this chapter different investigations are made. The first part of this chapter a grid sensitivity study is done, this is done with regular waves for two wave conditions. This part is done in 2D, without the Tri-Floater present.

In the second part irregular waves will be used for the input of the ComFLOW and aNySIM simulations. The spectra will be compared for the different simulations and also against model test results. Lastly, the run times of the different simulation settings will be compared.

The conclusions of this chapter will be the basis for the next part the research project, which is the main part.

4.1. Grid Sensitivity

Before studying irregular waves, a grid sensitivity study is done. The goal of the grid sensitivity study is to ensure that the grid used in the project is suitable. An investigation is made into the dissipation of the wave height. This method is chosen because it gives insight into wave propagation when undisturbed, without Tri-Floater. It is helpful to know whether the wave propagation works well before moving on to more difficult simulations, including motion.

4.1.1. Method

In the ComFLOW manual, there are rules for undisturbed wave propagation. At least 60 cells per wavelength and 6 cells in the wave height or 10 cells in the wave height when looking at higher waves. In these simulations, a domain of 384 m long is used. Only one cell is used in the y-direction as this is a 2D simulation.

For the test, a Stokes wave of the 5th order is chosen with a wave height of 10 m with a period of 12.5 s. ComFLOW then calculates the wavelength to be approximately 240 m.

In this test, an investigation is made into 0.5 m, 1 m and 2 m cell sizes for the 2D case. The results will be imported into Python using the same functions used in the decay tests.

To find the amount of dissipation in the simulation, the wave heights will be compared; these wave heights will be taken at fixed points close to the inflow and at (0,0) such that a time history is obtained.

To find the wave heights, the maximum and the minimum wave amplitude have to be found. These are found with the function `find_peaks` in Python. The absolute value of the minimum is added to the maximum to find the wave height. To achieve an accurate result, an average of wave heights is taken over multiple passing waves.

Because the inflow and (0,0) are at different points in the domain, one of the signals needs to be shifted in time to ensure that the same waves are compared. The time shift is easily found because the distance between the points and the velocity of the wave is known.

Dissipation percentage is calculated in Equation (4.1).

$$\frac{H_{0,0} - H_{\text{in}}}{H_{\text{in}}} \cdot 100\% \tag{4.1}$$

Where $H_{0,0}$ is the wave height at the origin and H_{in} is the wave height at the inflow boundary. The dissipation is calculated for the different grid sizes and input wave heights.

4.1.2. RESULTS

To highlight the differences between the different grid sizes, the wave height is plotted against time in Figures 4.1 and 4.2. Sub-regions are used to zoom in on the minima and maxima. The differences between the grid sizes are slight and are as expected; the wave height is largest in the 0.5 m simulation and smallest in the 2 m simulation.

The results of the dissipation calculations are presented in Table 4.1. Again the differences in the simulations are slight, approximately one percent. The results are again as expected, and the dissipation decreases with smaller grid size. However, with a smaller grid size, the amount of cells also grows. Even in 2D, this significantly affects computational time and even more in a 3D simulation.

The results are also quite sensitive for how many waves are taken for the averaging. However, it is still good practice to ensure that the wave propagation is correct before moving to more advanced simulations. The dissipation percentages should only be used in combination with the visual check of the wave's shape at specific locations in the domains.

Because the results for the three grids are close, it can be concluded that wave propagation is not a considerable concern with these grid sizes. However, it is clear that there is the dissipation of wave height, and this should be kept in mind when doing further research.

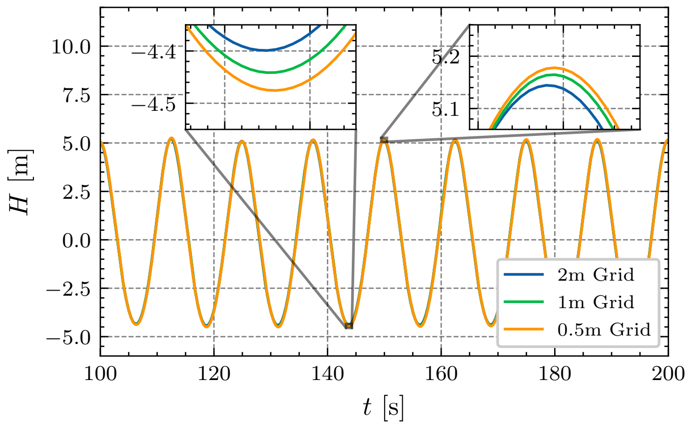


Figure 4.1: Plot of the time series of the grid sensitivity test for the 10 m case, the subregions show the small differences in wave propagation.

Table 4.1: The results of the grid sensitivity study. The results show the decrease in dissipation when refining the grid.

		Dissipation [%]	
Grid Size [m]	Cells	H = 10m	H = 5m
2	12k	3.8	2.6
1	49k	3.2	2.2
0.5	196k	2.6	2

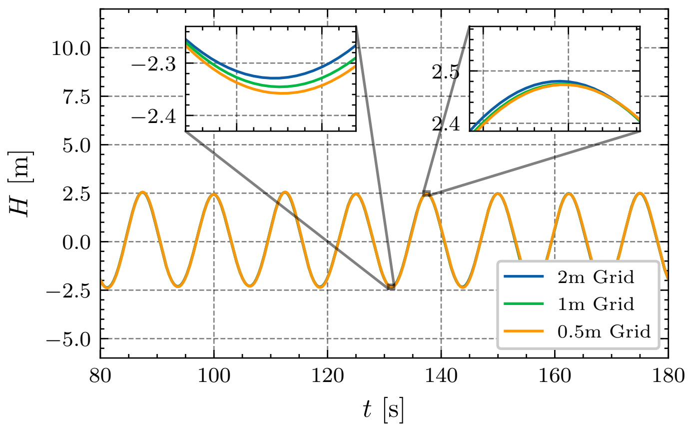


Figure 4.2: Plot of the time series of the grid sensitivity test for the 5 m case, the subregions show minimal differences in wave propagation.

4.2. IRREGULAR WAVES

In this section, the simulations with the motions of the Tri-Floater in irregular waves are investigated. An emphasis is placed on the differences between the types of simulation, ComFLOW and aNySIM, and the differences with different grid sizes. The differences between ComFLOW and aNySIM are significant concerning the computational effort. It is essential to gain insight into the simulation times. The conclusion of this chapter is significant for the next part of the project.

4.2.1. METHOD

3D SIMULATION

The 3D simulation is done in ComFLOW. In this simulation, the motion modelling is done by ComFLOW with the quasi-simultaneous coupling. All 6 degrees of freedom are interactive. The motion model uses the parameters as found in Chapter 3.

The results are compared to an irregular wave model test to investigate further the quality of the motion model and the simulation. The model test that is used is the "Waves Only" test with a significant wave height of 4.5 m and a period of 10 s. A start-up period of 30 minutes is excluded from the results such that the duration of the simulation is three hours.

For wave conditions in the simulation, the `linearsuperposition` plugin is used. It uses 500 wave components with an upper wave frequency ω limit of 4.0 rad s^{-1}. The linear components are retrieved from a Bretschneider spectrum that uses the same parameters as the model test to make a good comparison possible.

The simulations have a duration of 1 000 s. The CFL limit is set in between 0.2 and 0.5, as advised by the ComFLOW manual.

The 3D simulation is done with two different grids. These grids include refinements to save computational cells and thus computational time. The grid with 2 m grid size is shown in Figures 4.3 and 4.4. The 1 m simulation includes another grid refinement, which ensures a 1 m grid size in the critical area.

The inflow is on the left, and the outflow is on the right. The incoming wave will travel to the Tri-Floater, with its origin at (0,0), without any grid refinements, to prevent unnecessary dissipation at the refinement interfaces. Behind the Tri-Floater there are multiple refinements. With these refinements, the grid is made coarser, such that there are fewer computational cells. The coarsest grid size is 16 m close to the outflow and the bottom of the domain.

ANYSIM WITH 2D WAVE INPUT

The second method investigated in this chapter is an aNySIM simulation, with a wave that is input in the form of a time series. This wave is obtained from a 2D ComFLOW simulation, where wave propagation is simulated on the same grid as used in Section 4.1. At (0,0), the wave height is calculated. This wave height time series is then converted to an input file for the analysis simulation. For comparison, two 2D ComFLOW simulations are done with, again with 2 m and 1 m grid size. Because ComFLOW does not have a fixed

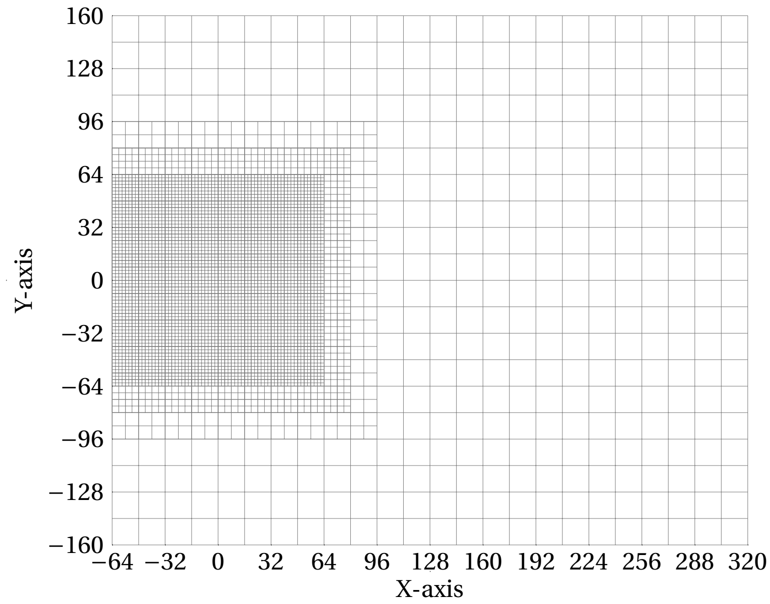


Figure 4.3: Irregular wave test grid setup for the X and Y axis

time step, the data needs to be interpolated, as aNySIM only allows a single time step. An example of this will be given in the next part.

In aNySIM, the same model is used as in Chapter 3, the main difference is that the decay tests were done with still water, while this simulation uses the `xship::TimetraceSpectrum` function with as input the time trace of the wave height.

DATA ANALYSIS

When the simulations are finished, the data is collected in Python. In this part of the project, it is necessary to use the frequency domain and spectrums. To transform from the time domain into the frequency domain, the Fast Fourier Transformation (FFT) is used. Data is required to have equidistant time steps for the FFT to work correctly. Data from ComFLOW can have different time steps as the CFL condition controls the time step. A time step close to the maximum time step of the simulation is chosen. In most cases, this is 0.05 s, because this time step is close to the ComFLOW time steps, it is possible to use linear interpolation. An example of this method is given in Figure 4.5. In this figure, an arbitrary peak is chosen to show the method that is used. This figure shows that linear interpolation can be used when the time step is appropriately chosen. The data is then also in the same form as the model tests and aNySIM output such that it starts at $t = 0.00$ s and then increases with 0.05 every step.

Next, the Fourier transform procedure is explained. The average of the input data is subtracted from the input data to detrend the data. The number of wave frequencies is half of the number of data points, and the Nyquist frequency is found as $f_{\text{nyquist}} = \frac{1}{2dt}$ To

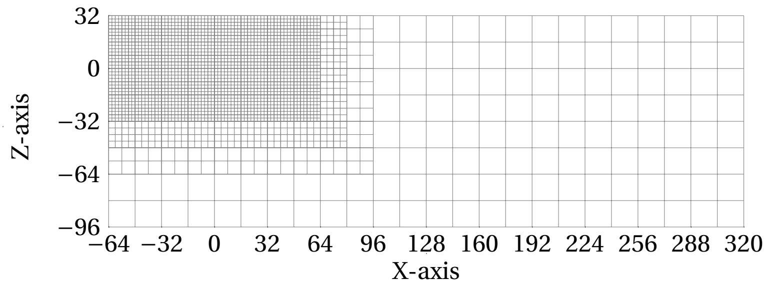


Figure 4.4: Irregular wave test grid setup for the X and Y axis

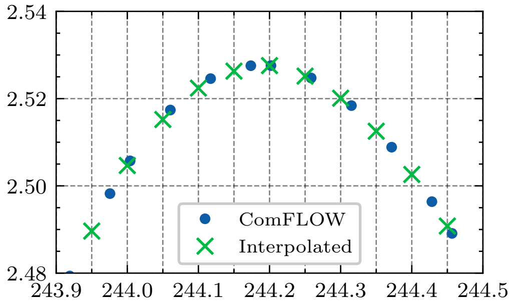


Figure 4.5: Example of the interpolation that is done when importing ComFLOW data. The crosses show the interpolated data at every 0.05 s to make it equidistant and shift it to a proper range.

find the spectral density of the time signal Equation (4.2) is used.

$$S_{xx}(\omega) = \left(\frac{dt}{\pi N_s}\right) |X(\omega)|^2 \tag{4.2}$$

In this equation the scaling factor $\left(\frac{dt}{\pi N_s}\right)$ is used to ensure the proper units. Here N_s is the number of samples in the time signal, and dt is the time step. $|X(\omega)|^2$ is the magnitude of the Fourier Transform of the time signal. The phase information is discarded, as this is of no use in the analysis.

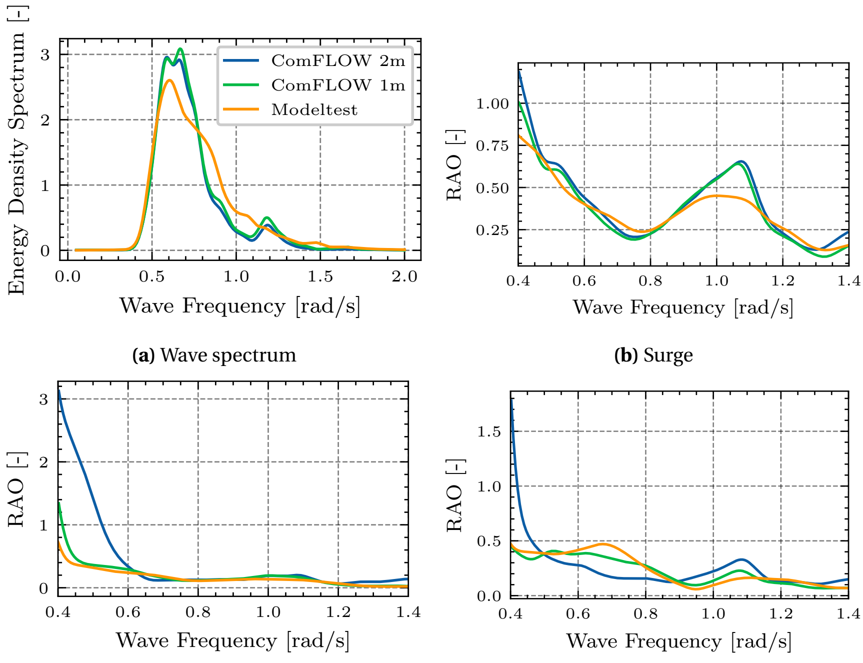
To compare different spectrums, a smoothing window is used. In this project, a hamming window is used. The hamming window is a cosine-sum window, and the window can be seen as a time averaging of the spectrum. The hamming window has been used before for wave spectra in different projects and is chosen for this project. The window has a window size as an input. This window size is variable and is set according to the frequency step size $d\omega$. In this project, a hamming window size of 0.1 rad s^{-1} was found to give good results. The hamming window is then convoluted with the spectrum using the Scipy function `filtfilt`.

4.2.2. RESULTS AND DISCUSSION

In this section, the results are presented and discussed. The conclusion will follow in the next section.

3D SIMULATION

The full 3D simulation in ComFLOW is done on 2 different grid sizes, 2 m and 1 m. The response results are presented in Figure 4.6, the spectrums are given in Figure A.1 in the appendix.



(a) Wave spectrum

(b) Surge

(c) Heave

(d) Pitch

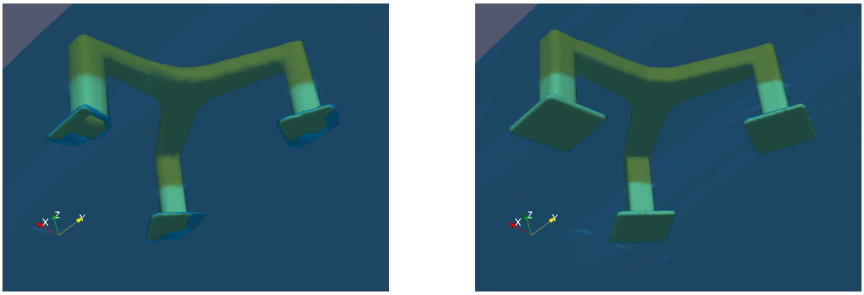
Figure 4.6: Response comparison between the ComFLOW simulation with different grid sizes. It can be clearly seen that the 2 m grid results are not correct.

In the response plots, it can be observed that the wave spectrums match closely in both simulations. As concluded from the grid sensitivity, the difference in wave propagation between the two grids is small, and the same can be concluded from this spectrum.

When looking at the surge degree of freedom, the plot shows that in the area where the most wave energy is present, between 0.4 rad s^{-1} and 1.4 rad s^{-1} the spectrum matches well. The peak around 1.1 rad s^{-1} shows a large deviation, similarly at 0.65 rad s^{-1} a deviation is found in pitch. The responses of the CFD simulations are larger than of the model test by approximately 40%. Because of this large deviation, it was decided to investigate the cause of the problem. This investigation is done in the next section.

Interesting results are found when looking at the heave and pitch plots. Significant differences between the two simulations occur when investigating the spectrums. The differences are even more significant when looking at the spectrums in the appendix. The response of the 2 m simulation in the lower frequencies, below $1\,\text{rad}\,\text{s}^{-1}$, is significant.

The cause of this significant response is found when looking at the 3D result in Paraview. In Paraview it is possible to look at the volume aperture, as mentioned in theory, to see how much of the cell is available for fluid. When there is an object, the cell is not available for fluid. When a surface is created where the volume aperture is 0.5, it is possible to see the object as ComFLOW perceives it, thus used in the motion calculation. This surface is shown in Figure 4.7. This result shows that the 2 m simulation does not correctly model the heave plates on the bottom of the Tri-Floater, which are designed to dampen the motion. The heave plate volume also varies in time, which makes the result not realistic.



(a) 2 m grid size **(b)** 1 m grid size

Figure 4.7: Visualization showing the problem with the heave plates with two different grid sizes.

With this knowledge, the responses in Figure 4.6 for heave and pitch can be explained. The lack of (part of) the heave plates explains the more significant motions because of the difference in damping with the actual case. In the 1 m simulation, the heave plates are correctly modelled, and the response matches the model test more closely.

The problem of the heave plates indicates that the 2 m simulation is not suited when a proper motion analysis is performed. A possible solution to this problem is to add local grid refinement around the heave plates. However, care should be taken that the local grid refinement does not influence the wave propagation.

DEVIATION AT $0.65\,\text{rad}\,\text{s}^{-1}$ AND $1.1\,\text{rad}\,\text{s}^{-1}$

The large deviations at $0.65\,\text{rad}\,\text{s}^{-1}$ and $1.1\,\text{rad}\,\text{s}^{-1}$ are investigated. Two methods are used for this; the first method is doing CFD simulations with a regular wave with a wave period such that this matches the frequencies where the deviation occurs. In the second method regular waves are also used, however, the Tri-Floater is now fixed in position, such that the wave forces on the Tri-Floater can be obtained from the simulation. These forces are then compared with the forces from the linear potential theory method in WAMIT.

The regular wave simulations are done on the same grid as in the previous sections. A 100 s simulation is performed with a significant wave height of 4.5 m. For the 1.1 rad s^{-1} case, three grid sizes are tested, 2 m, 1 m and 0.5 m. The results are given for surge, as this is the most clear, in Figure 4.8. The results for heave and pitch are presented in Figure A.3 and Figure A.4. In the plot it is clear that the RAO is closer to the model test

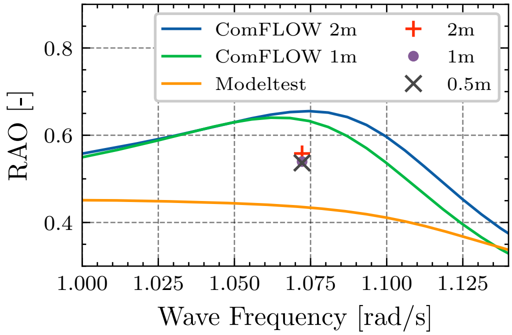


Figure 4.8: Result of the regular wave test at 1.1 rad s^{-1} placed inside the RAO plot of the surge direction.

during the regular wave simulation. It is also clear that the 1 m simulation result is closer to the model test than the 2 m simulation and that the difference between 0.5 m and 1 m is small.

For the 0.65 rad s^{-1} case, the regular wave simulation is only done with the 1 m grid. The result for pitch is given in Figure 4.9, for surge and heave the results are presented in Figure A.5 and Figure A.6. In the pitch degree of freedom, the result of the regular wave

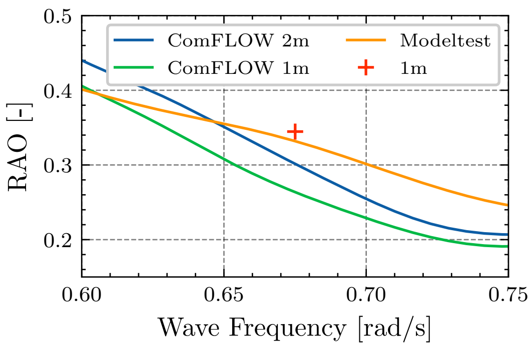


Figure 4.9: Result of the regular wave test at 0.65 rad s^{-1} placed inside the RAO plot of the pitch direction.

test is closer to the model test. For surge and heave the regular simulation RAO matches the result of the irregular wave test.

With the fixed simulation, the Tri-Floater is placed at it's equillibrium and the motions are fixed. The fluid forces on the Tri-Floater are extracted from the output files of ComFLOW. To compare the fluid forces to the forces from linear potential theory, the amplitude of the force signal has to be found. The force signal is sinusoidal, to obtain the amplitude the average is removed from the signal, and the maximum height is then taken. The forces from the linear potential theory model are in force per (wave) amplitude, it has to be multiplied by the wave amplitude for comparison.

The results are given in Table 4.2.

Table 4.2: Forces and moment compared between the CFD simulation and the linear potential theory model (WAMIT).

	F_{cfd}/F_{wamit}	
	$0.65\,\mathrm{rad\,s^{-1}}$	$1.1\,\mathrm{rad\,s^{-1}}$
Surge	1.11	0.96
Heave	0.91	0.95
Pitch	1.04	0.84

The largest outlier is the pitch moment at the $1.1\,\mathrm{rad\,s^{-1}}$, the other values are within 11% of the WAMIT results. This difference is small, the deviation in the RAO is probably not due to the difference in the wave loading on the Tri-Floater.

If the wave spectrum close to this point is investigated, as seen in Figure 4.10, the energy density is less in the CFD simulations.

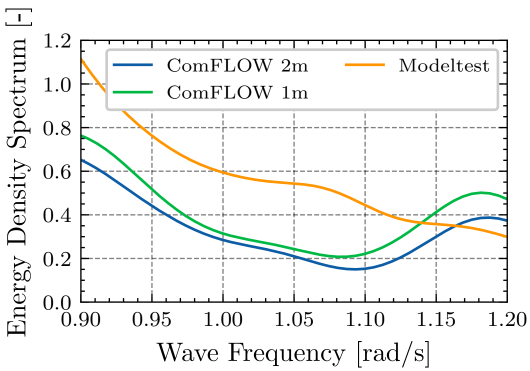
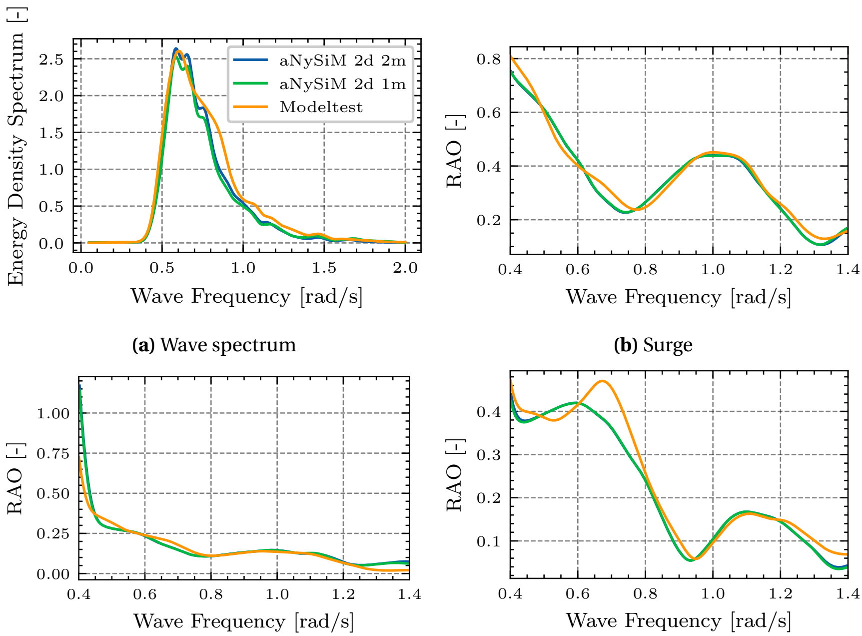


Figure 4.10: The wave spectrum at the region of interest.

The difference in energy density around this region of interest is significant. Because the results of the regular waves show that the RAO has a better match than in irregular waves, and that the difference in energy density is large, it is recommended to further investigate the irregular wave input or change the spectrum shape to match the model test more closely.

ANYSIM WITH 2D WAVE INPUT

The aNySIM simulations were done with input data imported from two different 2D ComFLOW simulations. The response results are presented in Figure 4.11, the spectrums are given in Figure A.2 in the appendix.



(a) Wave spectrum

(b) Surge

(c) Heave

(d) Pitch

Figure 4.11: Comparison between the aNySIM simulation with wave inputs of two different 2D simulations and the model test.

In the response plots, it can be observed that the method of using a wave signal from a 2D simulation is working well and also that the aNySIM model matches the model test results on the important frequency range. A large discrepancy is observed in the surge direction in the spectrum graphs, where the difference between the simulations and the model test is significant. However, this is at a frequency outside the range where most of the wave energy is present.

Another interesting result is that the difference between the 1 m and 2 m 2D simulation does not translate to a difference in the spectrums and responses. This behaviour is expected, as the difference in wave propagation between the grids is slight, as shown with the grid sensitivity test earlier in this chapter.

TIME COMPARISON

To answer the sub-questions of this research project, the time taken by the simulations is also an important aspect. The computational time results of the simulations are summarized in Table 4.3.

Table 4.3: Table summarizing the time taken by each simulation for a 1000 s simulation. All simulations are done on a computer with a 16 core Intel I9 7980XE.

		Time Taken [h]		
	Fluid Cells [k]	ComFLOW	aNySIM	Total
ComFLOW 3D 2m grid	78	8.5	0	8.5
ComFLOW 3D 1m grid	467	117.6	0	117.6
aNySIM with 2D ComFLOW 2m grid	9	0.6	0.1	0.7
aNySIM with 2D ComFLOW 1m grid	37	1.5	0.1	1.6

The most important fact that can be seen in this table is that the simulation times in the 3D ComFLOW simulations are significantly more computationally expensive. In these simulations, only 1000 s is simulated. This choice was made to keep the computation time acceptable and to still be able to make a reasonable comparison between the simulations. When looking at the 3D simulation with 1 meter grid size, the 1000 s simulation already takes 117 hours to complete on a computer with a 16 core Intel I9 7980XE. For engineering purposes, this is not a practical solution especially because multiple designs could be tested with multiple sea states. Furthermore, this is a simulation where a significant wave height of 4.5 m is used, the goal of this research project is to investigate wave impact, which will occur with larger wave heights. These higher wave heights could also increase the computation time because the time step will be smaller due to the CFL condition. It could be interesting to see if this is possible to investigate the computational time when the simulation runs on a cluster or cloud server. This could make the method more viable. During the simulations a clear improvement is seen when using more cores, however the improvement is getting smaller each time more cores are added. For a more detailed answer, a test needs to be done using multiple setups.

The time taken by the 3D simulation with a 2 m grid size is more acceptable than the 1 m size simulation. However, the heave plate problem must be solved before this simulation can be used in practice.

It can also be concluded that aNySIM simulation times are not significant compared to the ComFLOW simulations and take only on the order of minutes to complete. The results of the aNySIM simulation match the model test results closely and should be preferred in the current state.

An important side note is that for the relatively easy geometry of the Tri-Floater, aNySIM produces good results. However, in this project the limits of aNySIM for the motion modelling is not tested. It would be beneficial to test this in the future, as wave impact occurs at rough sea states where aNySIM might not always give proper results.

4.3. SUMMARY

In this chapter, the basis was laid out to create a long-term wave impact model using CFD.

First, a grid sensitivity test was done in a 2D ComFLOW simulation. The results of this grid sensitivity show the dissipation rates for three different grid sizes; 2 m, 1 m and 0.5 m. The differences between the simulations are small, and the dissipation percentages are sensitive to the number of waves used for averaging. The results should only be interpreted together with the wave shape at specific places in the domain. These two results together give a good insight in the wave propagation in the ComFLOW simulation. It can be concluded that the wave propagation on these grid sizes is not a large problem, but should be kept in mind when doing simulations further on.

Following the grid sensitivity, an investigation is made into irregular waves using ComFLOW. The irregular waves are created using linear wave superposition. Python calculates the wave components from a Bretschneider spectrum with parameters for the period and the significant wave height.

Four simulations are performed of 1000 s. Two simulations are done in 3D, with the motions being calculated with the quasi-simultaneous method in ComFLOW, with two different grid sizes. The two other simulations combine ComFLOW for the wavefield calculation and aNySIM for the motion modelling.

The main subjects being investigated in this part are the response amplitude operators (RAO) and the time taken by the simulations. The RAO analysis is done by analysing the time signals of the wave height and the motions with spectrum analysis.

An important result is that the 2 m grid in the 3D ComFLOW simulation is not sufficient for motion analysis due to problems with the modelling of the heave plates. Due to this problem, there is a large discrepancy between the response of the simulation and the model tests when looking at the heave and pitch degree of freedom.

The 2 m 3D ComFLOW simulation does not have this problem, and the motions are closer to the model test. It can also be concluded that the aNySIM simulations with the 2D ComFLOW wave input give good results. A side note should be placed on this result; the limits of the aNySIM model are unknown and not investigated in this research project. The limits could play a role when looking at rougher sea states and should be investigated in the future.

The simulation time taken by the 3D ComFLOW simulation is 118 hours for a 1000 s simulation on a 16 core desktop which means that this is not a practical solution for the problem of long term wave impact. It could be interesting to investigate how using a cluster or cloud server would decrease the simulation time. The aNySIM simulations with 2D ComFLOW wave input have shorter simulation times, closer to 2 hours.

5

Long Term CFD Simulation

In this chapter, the method for simulating long term wave impact is created. The conclusions of the previous chapter are used as a starting point in this chapter.

This chapter is set up differently from the previous chapters, as the methodology is the result. First, the goal and the problems are given, then the methodology is presented as a diagram. From this diagram, the various parts of the methodology are discussed.

5.1. Goal and problems

Before starting this chapter, the chapter's goal is given, and an overview of the problems that have influenced the methodology. These problems originate either from the previous chapters or have come up during the investigation into the method. The problems are given in bullet points and then explained in greater detail.

This chapter aims to find an efficient method to do a long-term (hours) wave impact load study on the Tri-Floater. This research project mainly focuses on reaching this goal with CFD using ComFLOW to see if this would be feasible to use in a design process for floating structures like the Tri-Floater.

The main problems are related to the following:

- **Computational time**
  The most significant problem in this project is computation time. If computation time was infinite, and the simulations would run in a short time frame, the solution would be relatively easy; create a CFD model that accurately models the motions of the Tri-Floater. After that, a verification study has to be done with, for example, model testing to ensure the quality of the results, and that would essentially solve the problem. However, computation time is finite, and it has to be possible to test designs in an acceptable time frame. Therefore a solution has to be found in which it is possible to simulate different designs efficiently.

- **Data storage**
  Another problem that comes up when doing extensive CFD simulations is the space taken by the CFD simulations. It is possible to change the amount of data that is saved by ComFLOW when doing simulations. For example, it does not make sense to have a complete snapshot of the domain on every time step when looking at a long term simulation of multiple hours. The data would quickly become hundreds of gigabytes, which is not practical at all. Data that can be saved relatively cheaply in this simulation is the wave height at specific locations and the object's motions.

- **Different programs and data types**
  Combining aNySIM, ComFLOW and Python have the benefit of using each program's strong points. However, complexity is added due to the different types of input and output for all programs.

- **Verification**
  When the method is finished, it should be verified appropriately to get insight into the methodology's value. Where earlier in the project the model tests could be used, the final result of this chapter can not be verified in the same manner. This fact should be kept in mind during the conclusion and will also be added to the recommendations for the future.

5.2. METHODOLOGY

In the chapter on irregular waves, it was found that the simulation times for a full 3D ComFLOW simulation were not practical. Longer simulations of multiple hours and different designs could take weeks of computational time. Therefore a choice was made for a hybrid option. In this solution, a 2D ComFLOW simulation is done without Tri-Floater for the desired duration. From this 2D simulation, the wave height at the origin of the Tri-Floater is extracted. This wave height is an input for the aNySIM simulation, which simulates the Tri-Floater motion during the desired period. From these motions, it is possible to find parts of the simulation where wave impact might occur. These 'events' are selected and will be simulated again in a full 3D ComFLOW simulation where the motion is prescribed from the aNySIM simulation and the wave field imported from the 2D simulation. Prescribing the aNySIM motion to the ComFLOW 3D simulation can prevent instability, such that a shorter 3D simulation can be done.

A diagram of the methodology is given in Figure 5.1. The diagram is split up into three columns, one for each program. In this section, all the parts will be discussed, in the next section an example simulation is presented to further clearify.

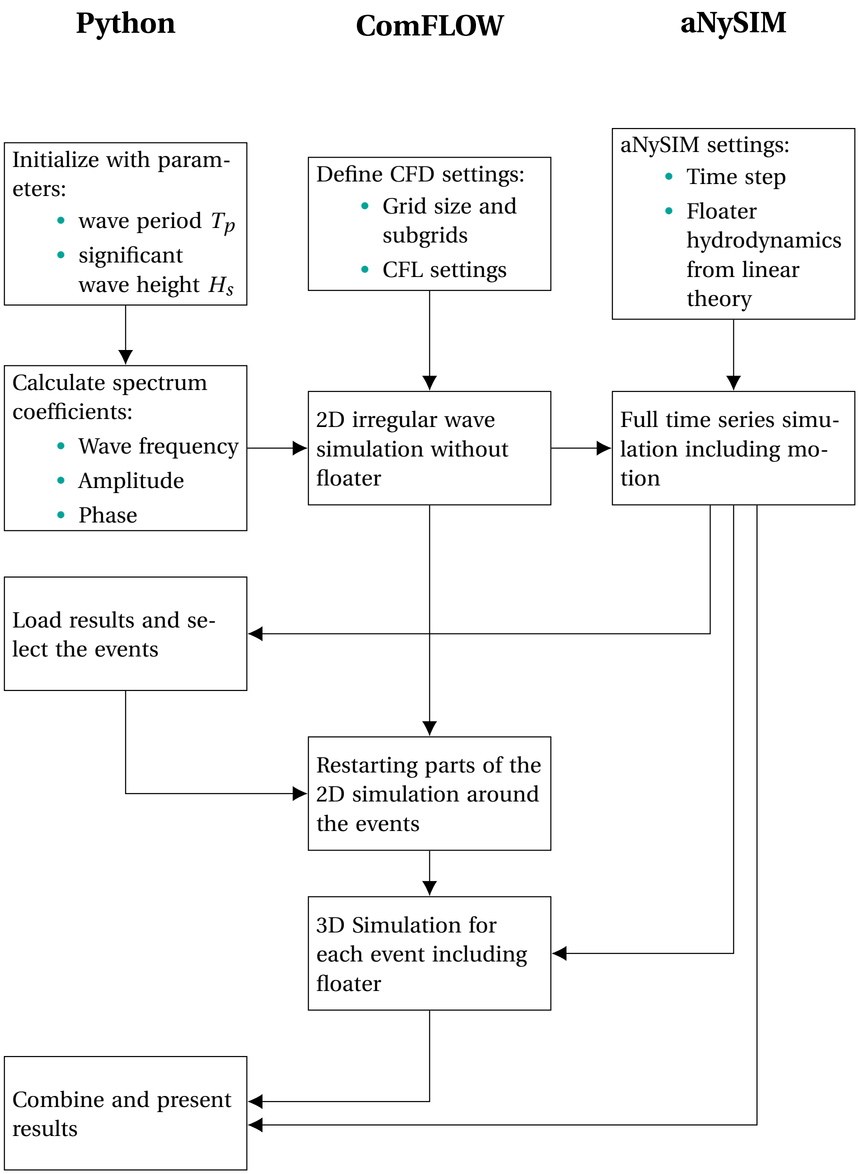


Figure 5.1: Diagram showing the different parts of the designed methodology. The diagram is split up into three columns, each with a different program. Python is an exception, as this is also used in between the different blocks.

5.2.1. INITIALIZATION AND 2D SIMULATION

The initialization of the simulation starts with creating the folder structure. It is convenient to create a new folder structure for every wave state that is desired. The copying of folders is a tedious process; therefore, it is automated. An example simulation folder is used; the standard structure and simulation files are placed in this folder. This folder is then copied to the desired location of the new simulation.

The folder stucture is shown in Figure 5.2.

```
/simulation-folder
 ├─/2d
 │  ├─/input-files
 │  ├─/output-files
 │  └─/2d-event-x
 │     └─/output-files
 ├─/anysim
 └─/3d-event-x
    ├─/input-files
    └─/output-files
```

Figure 5.2: Directory tree of the simulation folder structure. The 'x' marks the folders created for each event.

The structure is made so that every 3D wave impact event is simulated separately and can be analyzed.

When the folders are created, the 2D ComFLOW simulation has to be initialized.

Before the 2D simulation can be started, the irregular wave parameters must be set. The significant wave height H_s and the wave period T_p are used as input; a Bretschneider spectrum is created with these parameters. The 2D simulation can use this spectrum with the linear wave superposition plugin for the inflow boundary condition. The linear components for the plugin are the amplitude, the frequency and the phase. These components are calculated by Python and are exported to a file in the input files folder. In simulations done in the project, a difference in spectrums is observed, at this moment a proper solution is not found. The temporary solution is to increase the amplitude by a factor for higher wave heights, the frequency and phase is not adjusted.

Because the other input files are copied from an example simulation, the proper simulation settings should be set for these simulations. These files might need readjusting when doing different simulations with different wave conditions, Tri-Floater orientations or other settings.

In ComFLOW, it is possible to save data of the pressure, water height and velocity at each time step for a part of the domain. This option is used in this method such that the 3D event simulation can be started using this data as an initial and boundary condition. This functionality uses HDF5 field data.

During the first iterations of the method, the field data was saved for the complete (shorter) simulations. In the HDF5 field data file, much data is stored. This saving quickly becomes a problem when doing long simulations, as the data file can become several hundreds of GBs in size. The solution to this problem is found with another ComFLOW functionality, the restart function. When the restart function is enabled in the 2D simulation, snapshots of the simulation are saved every desired time step.

With the restart function, it is possible to turn off the HDF5 field data saving. This option has two advantages, less data storage and the simulation runs quicker, approximately 40%. The disadvantage is that the HDF5 field data is needed for the 3D simulation around the events. The 2D simulation has to be restarted for each event such that the output of the restarted simulation covers the desired 3D simulations time range. This restarting adds complexity to the methodology, but the advantages outweigh this added complexity.

The 2D simulation can thus be started saving only the wave height in specific locations and the restart snapshots with the desired time step. The rest of the simulation settings like the grid settings, CFL settings, domain and boundary are similar to the settings used in Chapter 4 and are not repeated in this chapter.

5.2.2. TIME DOMAIN SIMULATION AND EVENT SELECTION

When the 2D simulation is finished, the results are imported into Python. A spectrum of the wave height at the origin is created, such that this can be compared to the input spectrum.

The wave height time series is interpolated such that it has a wave height every 0.05 s. The data is then converted to an input text file for aNySIM.

The aNySIM model with its submodules was also used before in the previous chapters; no significant changes had to be made. Changes need to be made if different orientations are tested, but in this research project, only one orientation is tested with motions. The aNySIM simulation completely covers the time range.

To select events, a method is created such that the air gap of the Tri-Floater can be calculated from the results of the 2D ComFLOW simulation, and the motions from aNySIM. This is achieved by using the following steps:

1. Open the aNySIM timeseries in Python,
2. Apply transformation matrix to find the location X, Y, Z of a point that is placed on the Tri-Floater which is under the deckbox. The height of Z can be used to find the air gap.

$$\begin{pmatrix} X_{\text{transformed}} \\ Y_{\text{transformed}} \\ Z_{\text{transformed}} \end{pmatrix} = R(\alpha,\beta,\gamma) \begin{pmatrix} X_{\text{location}} \\ Y_{\text{location}} \\ Z_{\text{location}} \end{pmatrix} + \begin{pmatrix} X_{\text{motion}} \\ Y_{\text{motion}} \\ Z_{\text{motion}} \end{pmatrix} \tag{5.1}$$

In this equation, $Z_{\text{transformed}}$ is the height of the point of interest. X_{location}, Y_{location} and Z_{location} are the locations of the point of interest on the reference frame of the Tri-Floater. X_{motion}, Y_{motion}, Z_{motion}, α, β and γ are the results of the aNySIM motion analysis. The rotation matrix $R(\alpha,\beta,\gamma)$ is given as:

$$R(\alpha,\beta,\gamma) = \begin{pmatrix} \cos\alpha & -\sin\alpha & 0 \\ \sin\alpha & \cos\alpha & 0 \\ 0 & 0 & 1 \end{pmatrix} \begin{pmatrix} \cos\beta & 0 & \sin\beta \\ 0 & 1 & 0 \\ -\sin\beta & 0 & \cos\beta \end{pmatrix} \begin{pmatrix} 1 & 0 & 0 \\ 0 & \cos\gamma & -\sin\gamma \\ 0 & \sin\gamma & \cos\gamma \end{pmatrix} \tag{5.2}$$

3. In the 2D ComFLOW simulation a relative waveheight measurement is done on the X location where the point of interest would be if the Tri-Floater is placed in still water. There is a deviation when the Tri-Floater moves from equilibrium in the X and Y direction, this is neglected in the current method, but this could be changed in the future.
4. The wave height at that point is subtracted from the $Z_{\text{transformed}}$ location, such that the airgap is found between the deckbox and the water surface.
5. With a peak finding function, the peaks of the negative air gaps are found, these are the points where the air gap is smallest and wave impact is most likely to occur.

For each event, an object is created in Python. This object contains information about the event, the event time, the starting time of the simulation, and the end time. Saving these makes it easier to plot the results in a later stage.

This method could be further improved by adding more points of interest, for example, at the other columns of the Tri-Floater.

5.2.3. RESTARTS AND 3D SIMULATION

When the events and the associated times are known, the 2D restarts and the 3D simulations can be started.

ComFLOW's restart procedure has a function that makes it possible to restart simulations with different settings. The adjusted settings are the end time of the simulation because only a short period of the simulation has to be restarted. And the HDF5 field data needs to be enabled for the desired duration of the Tri-Floater.

The settings of the new simulation have to be changed in the original input file. It is good practice to copy the 2D inputs files and results so that the 2D results are also saved from being overwritten by a wrong setting in the restart command.

The output of the restarted 2D simulation should be set to the /2d/2d-event-x folder as shown in Figure 5.2. The results of these simulations can be used in the 3D simulation of the event.

3D SIMULATION

To get results regarding wave impact, a 3D simulation is used. An important function of ComFLOW is the HDF5 reader function. With this function, it is possible to set initial conditions and boundary conditions by importing an HDF5 file. Therefore, it is possible to import the wave field that is calculated in the more efficient 2D simulation. The function includes an extrapolation method to couple a 3D simulation with 2D data.

Using the HDF5 reader functionality, the 3D simulation can be short, and therefore it is possible to decrease the grid size and get more accurate results while still having acceptable computational time. The 3D simulation uses the same simulation settings as in the previous chapters.

MOTION MODELLING

Two different methods can do the motion modelling of the Tri-Floater. The first method is to leave the motion modelling to ComFLOW and the quasi-simultaneous interaction method, as explained earlier. In this method, the aNySIM motions are used as initial conditions for the object. The positions and velocities are extracted from the aNySIM time series and are placed in the input file of the 3D simulation by Python. This method could pose problems as the Tri-Floater is placed inside the waves instantly, which might give issues with the stability of the simulation. However, this is not observed in the tests that were done. An explanation for this could be that the Tri-Floater has a large mass and is not influenced significantly by the sudden placement.

The second method is to prescribe Tri-Floater motion from the aNySIM simulation. The motion is prescribed for the whole time range of the 3D simulation, not only as an initial condition. This method should, in theory, be more stable as there is no influence of the water on the object's motion.

Both methods are easy to implement and require a similar amount of preparation. From the tests done during the project comparing the two methods, there was no clear better method. The results of the tests that were done comparing the methods were hard to judge for value. For the rest of the project the aNySIM method was chosen, because of the results from the previous chapters, where the aNySIM model has proven to be closer to the model tests.

In the 3D simulation, outputs can be added. For the wave impact case, it is possible to add pressure points and surfaces, and also force surfaces. With these monitoring points, it is possible to find the pressure and forces that are experienced by the Tri-Floater during wave impact.

5.3. RESULTS

5.3.1. FULL METHODOLOGY

To gain insight into the methodology, results are presented of an example simulation.

SPECTRUM AND 2D RESULTS

For this example, a wave height of 10 m was chosen with a wave period of 10 s. A new folder structure is copied from the example folder structure. The theoretical spectrum was created with 500 wave components and has a high cut-off of 4 $\mathrm{rad\,s^{-1}}$. This spectrum is converted to linear wave components and saved to the 2D simulation input folder.

The 2D simulation with a duration of 3600 s is performed on a regular desktop with a 16 core Intel I9 7980XE processor. When running on all 16 cores, the simulation is performed in approximately 5 hours.

With the restart snapshots being created every 10 seconds, the restart snapshots only use less than 1GB of data storage.

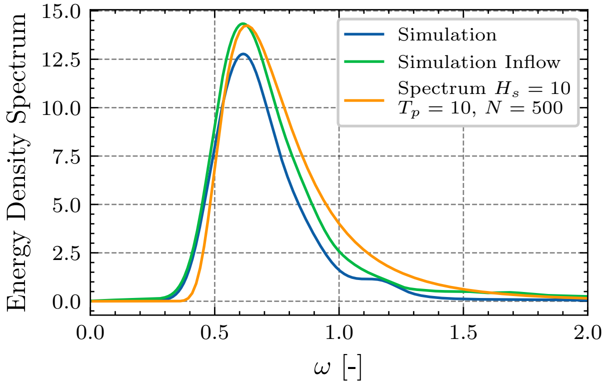


Figure 5.3: Plot showing the spectrums of this example.

A spectrum is made with the result of the 2D simulation. Because of the higher wave height, a correction factor of 1.4 is applied on the amplitudes of the inflow boundary conditions. If the wave height increases, this difference also increases, and a larger factor needs to be applied. More investigation is required on this, as setting a large factor like this, could point to a problem in the simulation as this factor should not be required. The spectrum is compared in Figure 5.3. The inflow spectrum has a larger peak in the spectrum, the difference in wave height between the inflow and the position of the Tri-Floater is larger than in the previous chapters. From the spectrum the significant wave height is calculated to be 9.0 m.

EVENT SELECTION AND RESTART PROCEDURE

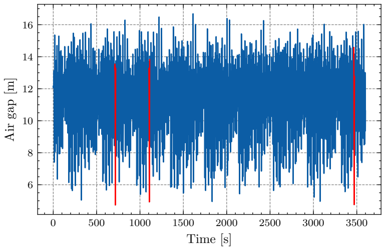
The wave signal of the 2D simulation is analyzed and the events are selected. An example is shown in Figure 5.4. In this example three events are selected.

The event that is selected for the 3D simulation is the event with its peak at 716.0 s. Five seconds before are simulated, and 4 seconds after.

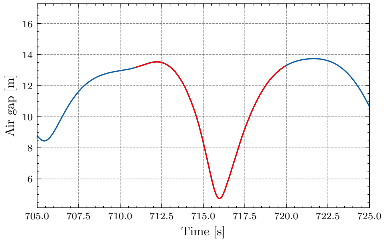
With the event times known, the 2D simulation can be restarted. Before the restarted simulation can be started the autosave file needs to be found. As the event starts at 701 s the simulation should be restarted from 700 s and stopped at 730 s. The file 'autosave0070.sav' is used for the restart. The HDF5 saving function is enabled with time step 0.05 s over the complete domain. The restarted 2D simulation is finished in a short amount of time, on the order of minutes.

3D RESULTS

With the 2D simulation, the wave field data for the 3D simulation is obtained.



(a) Complete simulation



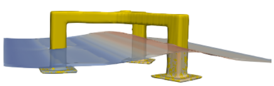
(b) Event

Figure 5.4: Event selection example.

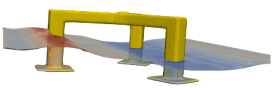
With the HDF5 reader plugin enabled, the initial and boundary conditions are extrapolated from the 2D simulation.

The motion data from the aNySIM simulation is converted in to the proper format to use with ComFLOW. In ComFLOW the type of motion is set to `timetrace` such that the motion is prescribed by the aNySIM timetrace.

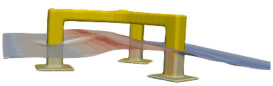
The 3D simulation of 9 s in duration has a computational time of 2 hours on 16 cores. Snapshots of the results are given in Figure 5.5.



(a) Before



(b) During



(c) After

Figure 5.5: Snapshots of the 3D simulation. This example shows the wave hitting the column. The color of the water surface is the x velocity.

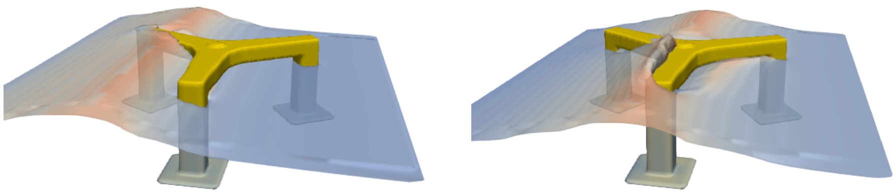
In this simulation no wave impact is observed, however, the simulation still shows that there are no large problems with 3D simulation.

5.3.2. Wave Impact Event

In the example simulation no wave impact is observed. An experiment is set up to see wave impact on the structure. The Tri-Floater is rotated 90 degrees, and lowered, such that the deckbox is just clear of the surface. The motion is disabled, and the Tri-Floater was fixed in place. Pressure points are added on the surfrace of the deckbox.

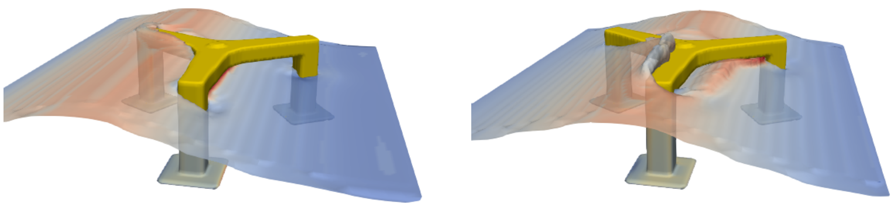
To test the method that is used in the long term wave impact methodology, the 2D simulation of a wave field, extrapolated on to a 3D simulation, two simulations are done. The first simulation is a regular 3D simulation, with a boundary condition that simulates a Stokes 5 wave train with a significant wave height of 15 m and a period of 11 s. The second simulation is a 3D simulation, that uses a 2D input simulation for the initial and boundary conditions. The 2D input simulation has the same Stokes 5 wave input boundary as the 3D simulation.

Both simulations use the same settings as used in the previous simulations.



(a) Before **(b)** During

Figure 5.6: Snapshots of the impact simulation. In this simulation the wave is a 3D stokes wave.



(a) Before **(b)** During

Figure 5.7: Snapshots of the impact simulation. In this simulation the input and boundary conditons are used from a 2D stokes wave simulation.

In the Figures 5.6 and 5.7 snapshots were given of the two simulations. The difference between the snapshots of the two simulations is minor. The pressure is plotted in Figure 5.8. In this figure it is clear that there is a difference in the pressure on the point.

The peak pressures are close in these simulations. The shapes of the peaks are also quite similar, however the signal of the simulation with the 2D input has more noise. Similar tests have been performed, with a similar result.

In both pressure fields no clear peak is seen, as would be expected in a wave impact event. In this research project this was not further investegated, however, this should be done in the future. An interesting investigation would be to set up a model test for the Tri-Floater which includes wave impact, and also test the influence of motion on

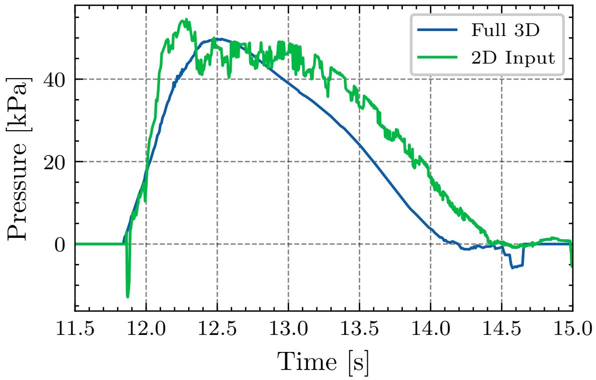


Figure 5.8: Plot showing the pressures on the Tri-Floater in both simulations.

the wave impact loading. If these model test would be done, more accurate simulations should be done with a smaller grid size, and a local grid refined could be applied around the wave impact area.

5.4. SUMMARY

In this chapter, the methodology for long term wave impact was presented. The problems that had to be solved or are recommended to be solved in further research are given at the chapter's start. The most significant problems are computational time and verification. The computational time is driving this project, as the problem would not exist with unlimited computational time. It is hard to verify the results because of the lack of verification, especially in the final wave impact result.

The method is based on the that a 2D simulation can be used as an input for a quick time domain solver and can also be used as boundary and initial condition for a full 3D simulation. With this possibility, a hybrid method is possible, where the largest part of the simulation is done in an efficient 2D setup. The wave impact load can be extracted from a 3D simulation. The current methodology of the 2D simulation boundary condition requires work. The wave spectrum that is found at the Tri-Floater's location, is not the same as the spectrum that is set in the input of the simulation. Currently a factor is applied to the input amplitudes of the irregular waves to correct for this, but this factor should not be required and can point to a fault in the simulation settings.

A diagram was created to clarify the different parts of the method. The method was explained in more detail, followed by an example simulation that demonstrates the methodology. Finally, the method was tested for wave impact pressures. In this test, two methods were compared. It was found that the 2D input method in a 3D simulation provides similar results as a full 3D simulation. No sharp wave impact peaks were detected in both simulations.

The results are hard to judge for correctness as there is no reference material. The methodology can be improved, especially on the irregular wave part, larger waves need to be simulated, preferably without the safety factor. In this project the aNySIM limitations are not tested, this could also be a problem in sea states with higher wave heights. If the 2D irregular wave method is improved and more time is spent on the (3D) wave impact part of the simulation, the simulations could be verified with model tests.

6

Conclusions and Recommendations

In this thesis work, an investigation is made into doing a long term wave impact study with CFD that includes motion. The simulations in this thesis work are done in time domain solver aNySIM and in ComFLOW, a numerical Navier-Stokes solver designed for the marine and offshore industry. In the first part of the project, motion analysis is done by using model test data that have been done on a floating offshore wind structure called the Tri-Floater. This analysis is done by performing decay tests in which the aNySIM and ComFLOW simulations are compared to the model test results in the time domain. In the second part of the project, the focus was shifted to irregular wave modelling, spectrums and response spectrums were compared to model tests. Another important outcome is the computational time analysis, which gave direction to the last part of the project. Lastly, in the final chapter of this report, a methodology was created in which it is possible to do long term wave impact analysis.

Following now is a discussion, in which the subquestions are answered based on the work that is done, followed by an answer on the main research question. Finally, recommendations are given for future research.

6.1. Discussion

In this section the research subquestions are answered using the results of the project.

- *How close can the results of the CFD simulation match the model test data with respect to the decay tests and irregular wave response tests?*
  To answer this research question, information from two chapters is taken:
  - **Decay tests**
    In the first chapter, the decay tests were performed in both CFD and in aNySIM. While the period is matched quite easily, the damping of the motion is more

difficult to properly implement. During the research project it was not possible to further improve the CFD model in a way that both the period and damping match the model test results.

- **Irregular wave tests**
  In the second chapter, motions in irregular waves were investigated. Response spectra were analysed, a full 3D CFD simulation was compared with an aNySIM simulation which used data from a 2D wave simulation. From the results, it could clearly be seen that the aNySIM model matches the model tests more closely than the ComFLOW CFD simulation.

The current results are promising. The decay test show that there are improvements to be made in the damping of the motions, it could be interesting for future to see if a different mooring system could be selected, for example, the XMF mooring option that ComFLOW provides, in this it is possible to include a similar mooring system as in the aNySIM simulations. It is also possible to add quadratic damping to the simulation to find if this improves the decay tests. The irregular wave results show that the simulation with a 1 m grid size are match the model tests in a large part of the wave frequency. There are still some areas where the deviation between the model test and the CFD simulation are significant, it is suspected that this is due to problem with the spectrum, caused by the irregular wave boundary condition.

- *How coarse can the grid of the CFD simulation be made such that it predicts motion accurately enough to predict the occurrence of wave impact events?*
  The most important limitation is the proper modelling of the Tri-Floater. In the irregular wave simulation, it was concluded that, when using a 3D CFD simulation that includes motion calculation by the quasi-simultaneous method, a grid size of 1 meter or smaller has to be used. The heave plates of the Tri-Floater are not properly modelled when using a larger grid, leading to results which are not physical and incorrect. The second limiting factor is the wave propagation in the simulation, however, in a grid sensitivity study it was found that with grid sizes 2, 1 and 0.5 meter the difference in wave height dissipation was noticeable but small.

  If performing the aNySIM simulation to predict the occurrence of wave impact events, grid sizes are only crucial for the 2D simulation that calculates the wave input. The difference between the 2 m and 1 m grid size was found to be minor, higher wave heights should be investigated in the future.

- *Is it more efficient to use CFD or another motion solver in a complete long-term simulation?*
  In the chapter on irregular waves simulation were performed in which the computational times were compared. The computational time of a 1000 s 3D simulation where the motions are calculated by ComFLOW, is approximately 117 hours on 16 cores. This is again due to the limit of the 1 meter grid size, if a 2 meter grid size would be possible the simulation would take approximately 8 hours. For engineering purposes the long simulation times are not practical when running on this computer. The full 3D simulation could be more practical when a cluster or

cloud server is used, however, more verification needs to be done, as the cost associated with running on a cluser or in the cloud are also larger than using the computer that was used in this project. It should be possible to further improve the simulation, reducing the simulation time and increasing the quality of the results. However, this will probably still give the same order of magnitude of simulation time.

The combination of a 2D simulation with a time-domain solver like aNySIM has proven to be more efficient at the current stage with simulation times of 1 or 2 hours depending on the grid size. The benefit of the reduced computational time outweighs the added complexity of using different programs. In the method, it was chosen that the 3D simulation would use the motions that are prescibed from the aNySIM simulation. This reduces the chance of instabilities in the simulation, and also enables a shorter run time. The full interactive motion method could also be used instead of the prescribed motion from aNySIM, but this should be further investigated.

- *To what extent can ComFLOW be used to achieve accurate predictions of wave impact loads?*
  Earlier verifications done with ComFLOW have showed good results in terms of free surface flows and wave impact loads. From the simulations done in this research project, it is not possible to answer this research question properly. More research needs to be done, it could be beneficial to do model tests, such that the simulation results can be verified

6.2. CONCLUSION

With the sub-questions answered it is possible to answer the main research question.

> **How can we develop an efficient framework in which it is possible to do a long term CFD analysis of the Tri-Floater which captures wave impact loads accurately using ComFLOW?**

The main conclusions are summarized:

- ComFLOW proves that even with a relatively easy model, decent results are obtained. If the CFD model were to be fine tuned and further improved, it should be possible to further match the model test results.
- A long term 3D simulation of multiple hours is not practical at this stage, this could change when the simulation can run on a cluster or cloud server.
- With some adjustments the aNySIM model in combination with a 2D CFD simulation gives good results in both the decay tests and the irregular wave tests. More work needs to be done on fixing the 2D irregular wave simulation such that higher wave heights can be tested.
- The methodology is expected to work well, however more improvements need to be made and tested before a real judgement can be made.

At the current stage, the efficient framework is made up of a combination between CFD solver ComFLOW and time domain solver aNySIM. In this setup, the advantages of both programs can be used. The aNySIM model is very efficient at solving the motions of the Tri-Floater, especially when only the motions are of concern, and not the wave impact. During most sea states it does not make sense to have a CFD simulation calculating the motion of a floating object, the computational time is not practical. There are more efficient methods for calculating motion, aNySIM is just one example.

The framework designed during this research project consists of a 2D long term CFD simulation with a duration of multiple hours. From this 2D simulation, the wave height is extracted and imported in to aNySIM, which does a full motion analysis. When the motions of the Tri-Floater are known, events can be selected for a accurate 3D simulation. From this 3D simulation information can be extracted on wave impact loading on the Tri-Floater.

In examples the methodology has shown that it should work, however, there are improvements to be made and further validation is required before a real judgement can be made on the methodology.

6.3. RECOMMENDATIONS

It is clear that this methodology should be further investigated and improved. This section gives recommendations for further research.

6.3.1. RECOMMENDATIONS FOR THIS METHODOLOGY

- The ComFLOW model is currently using a simple linear spring system for the mooring. It would be beneficial to improve the mooring system. ComFLOW has the option to use more advanced mooring systems, similar to the model used in the aNySIM simulation. When the CFD model is improved further, it is expected that the CFD simulation should match the model tests more closely.

- The problem with the irregular wave boundary condition in the 2D simulation requires attention, a temporary solution is made by using a factor to increase the amplitude of the wave components, but the use of this factor is debatable, as it should not be required and might point to wrong simulation settings or other problems at the boundary.

- When the issues are fixed, simulations with higher significant wave height need to be done such that wave impact actually occurs.

- Investigating the possibility to refine the 3D CFD simulation when using inputs from a 2D simulation, this might be beneficial when wave impact occurs and a smaller grid is required. Another interesting idea which has not been tested in this project is to have a smaller domain in the 3D ComFLOW , as the wave propagation is calculated in the 2D simulation. In this way it is possible to have the same amount of cells but with a smaller grid size, which might be beneficial when testing wave impact.

- Currently, there is no model test data available for the Tri-Floater structure on the subject of wave impact, if this model test data is available, the 3D simulation can be further improved and the method can be further tested.
- The event selection in the methodology should be improved, the current method uses relative wave height compared to the floater to select events. This relative wave height is only calculated on 1 column of the Tri-Floater, wave impact might also occur on other columns.
- Investigate more orientations of the Tri-Floater with the methodology, currently only one orientation is tested, this will enable wave impact from other angles which might be more interesting.

6.3.2. OTHER OPTIONS

The designed methodology still requires a lot of human input. In an ideal simulation a parameters would be set, like the wave height and period, and the simulation would run it self without further input. However, 2D simulation combined with aNySIM and 3D ComFLOW requires a lot of checking and input.

A more ideal solution would be that there is one program that runs the simulation automatically as fast as possible, with for example a very coarse grid, or another option for the motion modelling. And that when in the simulation a certain criteria is surpassed, automatically switches to a smaller grid and time step in which accurate wave simulations can be preformed.

The main benefit of this is that it requires less human input. However, it is also more efficient, as parts of the simulation do not have to be re-run as with the methodology used in this research project.

If the full 3D CFD simulation were to be done, an investegation should be made if it is feasible to run the full 3D simulation on a cluster or on a cloud server, as the simulations in this project were performed on a 16 core desktop and proved to be too long in duration. The cluster or cloud server might make the full 3D simulation a more feasible solution, however, the cost will be higher.

References

[1] Clean Planet. A Clean Planet for all. *A European strategic long term vision for a prosperous, modern, competitive and climate neutral economy*, 2018.

[2] Clarisa Dias and John Upton. The future of wind energy is floating turbines on the ocean. *Quartz*, 2021.

[3] Fons Huijs, Rogier De Bruijn, and Feike Savenije. Concept design verification of a semi-submersible floating wind turbine using coupled simulations. In *Energy Procedia*, volume 53, pages 2–12. Elsevier Ltd, 2014.

[4] Irja Viste-Ollestad, Terje L Andersen, Narve Oma, and Sigvart Zachariassen. Investigation of an incident with fatal consequences on COSL Innovator, 30 December 2015. Technical Report July 2016, 2015.

[5] Johan M J Journée and J A Pinkster. *Offshore Hydromechanics*. 2001.

[6] K. M.T. Kleefsman, G. Fekken, A. E.P. Veldman, B. Iwanowski, and B. Buchner. A Volume-of-Fluid based simulation method for wave impact problems. *Journal of Computational Physics*, 206(1):363–393, 2005.

[7] Roel Luppes, Henri J.L. Van Der Heiden, Peter Van Der Plas, Arthur E.P. Veldman, and Bülent Düz. Simulations of wave impact and two-phase flow with comflow: Past and recent developments. *Proceedings of the International Offshore and Polar Engineering Conference*, (October 2015):235–242, 2013.

[8] Arthur E P Veldman, Henk Seubers, Peter van der Plas, and Joop Helder. Accelerated free-surface flow simulations with interactively moving bodies. In *MARINE VII: proceedings of the VII International Conference on Computational Methods in Marine Engineering*, pages 604–615. CIMNE, 2017.

[9] Arthur E P Veldman, Henk Seubers, Peter van der Plas, Matin Hosseini Zahraei, Peter R Wellens, and Rene H M Huijsmans. Preventing the added-mass instability in fluid-solid interaction for offshore applications. In *International Conference on Offshore Mechanics and Arctic Engineering*, volume 51210, page V002T08A038. American Society of Mechanical Engineers, 2018.

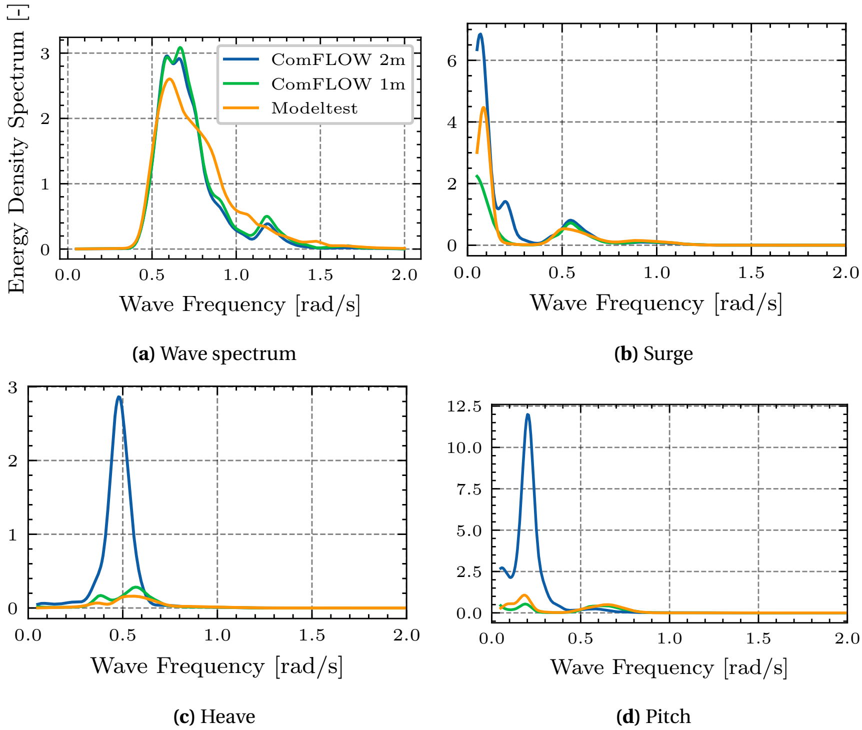
[10] Guangyu Wu, Jang Whan Kim, Hyunchul Jang, and Aldric Baquet. CFD-based numerical wave basin for global performance analysis. *Proceedings of the International Conference on Offshore Mechanics and Arctic Engineering - OMAE*, 1:1–11, 2016.

[11] Thomas B Johannessen and Øystein Lande. Long term analysis of steep and breaking wave properties by event matching. In *International Conference on Offshore Mechanics and Arctic Engineering*, volume 51227, page V003T02A040. American Society of Mechanical Engineers, 2018.

[12] Henri J.L. Van Der Heiden, Arthur E.P. Veldman, Roel Luppes, Peter Van Der Plas, Joop Helder, and Tim Bunnik. Turbulence modeling for free-surface flow simulations in offshore applications. *Proceedings of the ASME 34rd International Conference on Ocean, Offshore and Arctic Engineering*, 2015.

A

MOTIONS IN WAVES



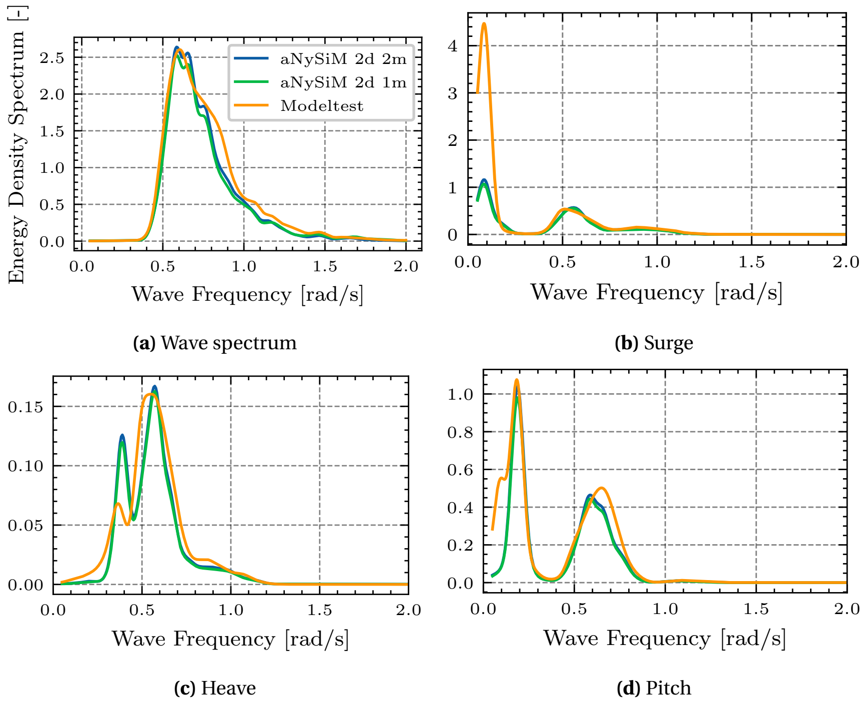
(a) Wave spectrum

(b) Surge

(c) Heave

(d) Pitch

Figure A.1: Spectrum comparison between the ComFLOW simulation with different grid sizes. It can be clearly seen that the 2 m grid results are not correct.



(a) Wave spectrum

(b) Surge

(c) Heave

(d) Pitch

Figure A.2: Spectrum comparison between the aNySIM simulation with different grid sizes for the 2D input.

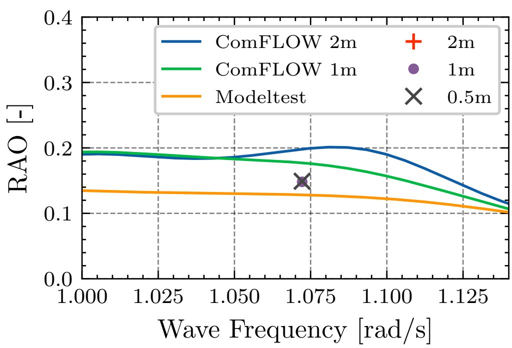


Figure A.3: Result of the regular wave test at 1.1 rad s^{-1} placed inside the RAO plot of the heave direction. The 2 m regular wave RAO is outside of the range of the plot.

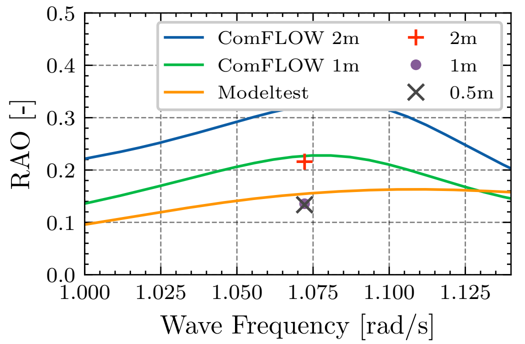


Figure A.4: Result of the regular wave test at 1.1 rad s^{-1} placed inside the RAO plot of the pitch direction.

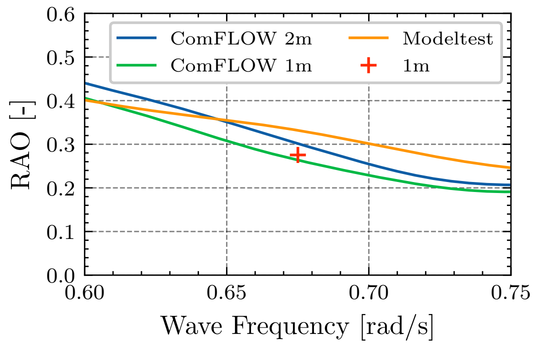


Figure A.5: Result of the regular wave test at 0.65 rad s^{-1} placed inside the RAO plot of the surge direction.

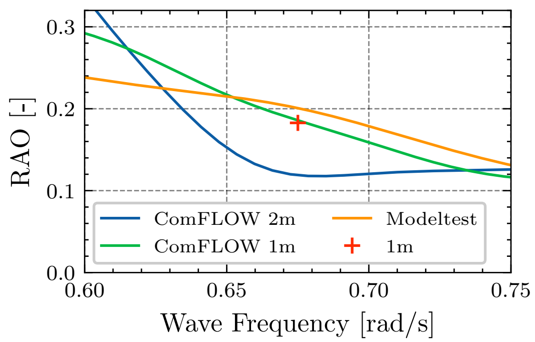


Figure A.6: Result of the regular wave test at 0.65 rad s^{-1} placed inside the RAO plot of the heave direction.