

Chapter 1

Introduction

1.1 Problem definition

In the offshore industry a structure for the production and storage of oil or gas is placed at one location for many years, often for over twenty years. These structures must survive all weather types, including heavy storms. The photograph in Figure 1.1 shows a Floating Production, Storage and Offloading vessel (FPSO) in a heavy storm at the North Sea.



Figure 1.1: Floating Production Storage and Offloading vessel (FPSO) in heavy storm

In rough seas large masses of water can invade the ship's deck when the freeboard is exceeded by high waves. This is called green water after the solid water mass that is coloured green, in contrast to white water that consists of spray and foam mainly. Green water can occur at the ship's bow or at the side of the ship damaging equipment on the deck or living quarters of personnel. Especially on FPSO's, where a lot of sensitive equipment is present at the deck, green water causes a lot of damage. Ersdal et al. [21] and Morris et al. [66] report on green water events at production ships in the North Sea. Figure 1.2 shows two photos with damage caused by green water [7].



Figure 1.2: Damage to the fire fighting platform of the Emerald FSU (Floating Storage Unit, left) and the damage and first repair of a window after a green water incident on the Varg FPSO (right)

In seas with steep fronted waves offshore structures also face the problem of wave impact on their bow or bottom. One reported case is the Schiehallion FPSO where a rather steep wave had damaged the bow, which finally resulted in an evacuation of all personnel [34]. The photo in Figure 1.3 shows the damage to the front of the bow.



Figure 1.3: Wave impact damage causing dents to the bow of the Schiehallion FPSO

For the design of offshore structures these issues have to be taken into account to prevent economic loss due to damage to the equipment and guarantee as safe conditions as possible for the personnel. Very important in the design phase of a structure are model tests to calculate the loads on different parts of the structure. Model testing is expensive and it would be convenient to replace a part of the model testing by numerical simulation,

or in any case focus model tests on critical events only. Therefore, there is a great need for calculation methods that can take into account the highly nonlinear situations that occur in heavy seas. Until recently, most calculation methods have been based upon potential theory and have not been able to calculate local flow phenomena like green water on the deck of a ship or slamming caused by (near-)breaking waves.

This thesis presents a simulation method based on the Navier-Stokes equations that is developed (and is still in development) for the simulation of water loading on offshore structures. The objective of the research presented in this thesis can be phrased by:

*To develop a robust and validated numerical method
for the prediction of local wave impact loads on floaters.*

This research builds on the work of Gerrits [26] and Fekken [23], who developed a method for local flow phenomena with the presence of a free liquid surface and moving objects. At the boundaries and at the object no-slip or free-slip conditions can be used. With their method sloshing problems, water-entry problems and problems with simple in- and outflow conditions can be handled (like channel flow with a prescribed uniform inflow). The emphasis of the current work has been on validation and further robustness improvement of the existing method and the extension of the method with complex in- and outflow conditions, such that wave impact simulations can be performed. To reach our goal, the following items need to be modelled, implemented and investigated:

- The design and implementation of wave generation options.
- The design and implementation of a zonal modelling method, where the outer domain provides the wave kinematics and vessel motions for the local domain calculation.
- The modelling of wave damping, due to the discretisation.
- The modelling and investigation of outflow conditions to prevent wave reflections.
- The investigation of the handling of the free surface (boundary conditions and displacement) for robust and accurate simulations.
- Generation of pressure output, to analyse impact results.
- The validation of the method on the different stages in impact problems, as there are wave propagation, water entry of a structure, and impact of water on the structure.

Throughout this thesis, all these items will get attention in the description of the model and the results of validation. After all the individual elements are visited a simulation can be performed containing all the elements, like green water on the deck of a moving vessel.

1.1.1 Focus on green water events

Some more attention will be paid to the green water phenomenon as an important application of the simulation method described in this thesis. Buchner [7] has conducted a thorough investigation to the characteristics of green water flow, and the effect of bow shape using model tests. Also some different structures were put on the deck to measure the effect of the shape of the structure (for example a deck house as shown in Figure 1.4).



Figure 1.4: Experiment of a green water event on the bow of an FPSO

Green water loading is a highly complex and nonlinear process. Buchner [7] showed that the following phases can be distinguished in the process of green water on the foredeck, see Figure 1.5:

- A. The combination of a high wave and the pitch motion of a vessel results in nonlinear swell-up around the bow.
- B. The water is almost at rest around the bow, after which it starts to flow onto the deck in a 'dam breaking'-type of flow.
- C. This results in a 'Hydraulic jump'-type shallow water flow on the moving deck, focussing into a high velocity water 'jet' when the water fronts from the sides meet.
- D. Water impact and water run-up occur in front of the structure, and eventually the water is turning over.

When attempting to use a numerical method to describe these phenomena, the method should be able to deal with complex nonlinear flows. If the focus of the investigation is limited to the local flow around the bow, specifically it should be able to handle:

1. Water entry of a flared bow structure.
2. Complex flow onto the deck, including the discontinuity at the deck edge.
3. 'Hydraulic jump'-type shallow water flow on a moving ship deck.

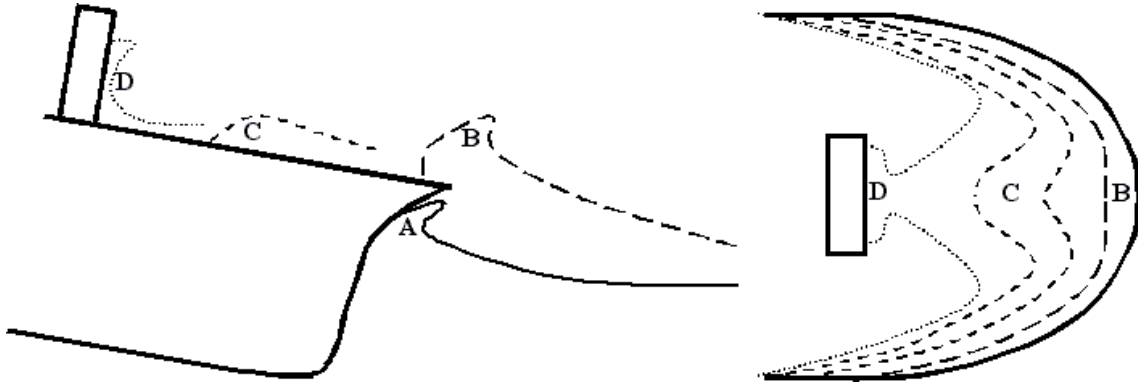


Figure 1.5: The main phases of the green water problem schematically in side view (left) and top view (right): from the nonlinear relative wave motions in front of the bow, via the complex flow onto and on the deck to the impact on deck structures

4. Meeting water flows on the deck.
5. Short duration water impact on a structure.
6. Overturning flow after run-up of the water in front of the structure.

Initial numerical investigations of green water loading focussed on the shallow water flow on the deck, using Glimm's method, see for instance Mizoguchi [63, 64], Zhou et al. [102], and Stansberg et al. [85]. Also (nonlinear) dambreaking theory is used for the simulation of the green water flow on the deck, e.g. Yilmaz et al. [97]. With this type of method only the 'hydraulic jump'-type shallow water flow on the moving deck can be simulated (Phase C). The computational domain can consequently be limited to the area on the deck. The freeboard exceedance around the deck and the related velocities were used as boundary conditions.

Greco et al. [35] use a two-dimensional fixed structure to represent the ship's deck. Waves are overtopping the freeboard of the ship resulting in green water on the deck. The calculations are performed with a Boundary Element Method and the resulting water motion at the deck is compared with experiments. Nielsen et al. [68] take into account all the phases of the green water phenomenon by using a Navier-Stokes solver with the Volume-of-Fluid method for the displacement of the free surface. Waves are generated at the inflow boundary. The ship is fixed, but the relative wave motion is modelled by velocity boundary conditions at the bottom of the domain. The results are compared with experiments of Buchner [7]. Gómez-Gesteira et al. [33] use the Smoothed Particle Hydrodynamics (SPH) method for the simulation of waves overtopping a fixed deck. The SPH method is a purely Lagrangian method, so no mesh is needed. The resulting wave profile is compared with experiments.

1.2 Description of the simulation method

In this thesis a method is described that in principle can handle all six points needed for the simulation of the green water phenomenon. The method, incorporated in the

computer program COMFLOW, is based on the Navier-Stokes equations, which describe the motion of an incompressible, viscous fluid. Originally, the method was developed to simulate liquid sloshing on board spacecraft [29]. In this application the surface tension is the driving force in absence of gravity. Also an accurate description of the free surface is essential. Another application has been found in medical science, where blood flow through elastic arteries has been studied [57, 58, 59].

In the maritime application area sloshing in anti-roll tanks was simulated [17, 18]. Also, a pilot study on the simulation of green water has been performed by Fekken [24]. An approach was used that deals with water impact and water run-up in front of the deck structure. Using the simulation method Fekken was able to simulate the flow on the deck and resulting impact accurately. However, the computational domain was limited to the area on the deck. In the left of Figure 1.6 the initial configuration is shown. The (measured) freeboard exceedance around the deck was used as boundary condition for a breaking-dam type flow. The right picture in Figure 1.6 shows the run-up and falling down of the water from the structure on the deck. The deck was not moving in this approach.

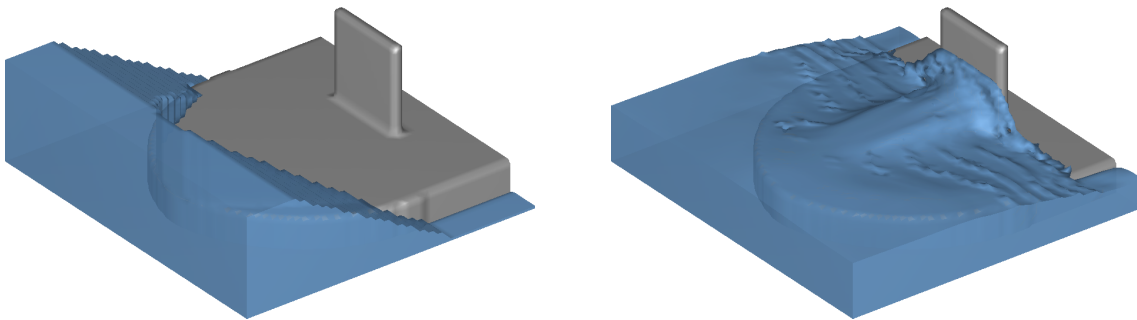


Figure 1.6: Initial configuration and run-up on a deck structure in the approach of Fekken et al. [24] for green water simulation

New applications in the marine field are sloshing in LNG-tankers [94] and wave loads calculation on subsea structures in the splash zone [9]. Also wave-in-deck calculations have been performed, which is an important issue on, for example, the Norwegian Ekofisk oil field [47].

1.2.1 Grid and discretisation

The simulation method described in this thesis numerically solves the Navier-Stokes equations. Thereto, the computational domain is covered by a fixed Cartesian grid. The geometry, which is in general not rectilinear of form, cuts through the Cartesian grid cells, resulting in cut cells. A Cartesian grid is very suitable when simulations are performed including highly distorted fluid interfaces. Then it is not an option to align the grid with the moving interface. Further, the Cartesian grid is a structured grid, which means that all the grid cells have the same number of cell faces per grid cell and the number of cells surrounding each grid point is constant. Another option is to use a boundary fitted unstructured grid. The advantage is that no cut cells are present in this method. But in our method and applications a structured grid is preferable above an unstructured grid, because the fluid interface needs to be kept sharp to correctly predict pressure impact peaks.

When using unstructured grids a diffusive interface method is commonly used, where the interface is smeared over a few cells. Certainly, the Volume-of-Fluid method adopted in our method is not suitable for unstructured grids for bookkeeping reasons. Another disadvantage of an unstructured boundary fitted grid is that when complex geometries are involved, as in simulations of offshore structures, the generation of a boundary fitted grid is very difficult. For moving structures, the grid has to be regenerated every time step, which is very time consuming. For these reasons, a fixed Cartesian grid has been chosen.

Another method that is sometimes used in hydrodynamic applications is the Smoothed Particle Hydrodynamics (SPH) method [65], which is a meshless method. Particles are put in the flow and every particle has a mass and velocity. A particle is influenced by other particles that are within a certain distance from the particle. The method can handle large deformations of the fluid interface automatically. The method has also been used in green water applications [33].

In the current method, the variables are staggered on the Cartesian grid as in the original Marker-and-Cell method [39], which means that the pressure is defined in cell centers and the velocities on cell faces. The advantage of a staggered grid is that mass conservation can be applied easily in a cell, without the need of interpolations. The finite volume method is used for the spatial discretisation of the Navier-Stokes equations. In control volumes conservation of mass and momentum is applied derived from the conservative form of the Navier-Stokes equations. This discretisation is performed in such a way, that the underlying symmetry properties of the continuous operators also hold for the discrete operators. This leads to a stable method where the kinetic energy is only dissipated (due to diffusion) [92]. For the time discretisation the first order Forward-Euler method is adopted. A Poisson equation for the pressure results, which is solved using Successive Over-Relaxation (SOR) with an automatically adapted relaxation parameter [5].

1.2.2 Free surface displacement

A very important aspect of the applications is the presence of a free liquid surface. Many methods for the treatment of the free surface are described in the literature; often the methods for flow calculations with a free surface are classed by the method for the interface treatment. An overview of the various methods available can be found in [80]. The most popular ones are the level-set method and the Volume-of-Fluid method, which is adopted in the current method.

In the level-set formulation a distance function $\phi(x, t)$ is introduced denoting the distance from x to the initial interface location at $t = 0$. The interface corresponds to the contour $\phi = 0$ at any instant [72]. In this method highly distorted interfaces can be treated and also topology changes are incorporated automatically. Although the interface is of finite thickness, the physical properties such as surface tension can be applied easily. A major problem of the level-set method is the lack of mass conservation (see e.g. [54]). Several strategies have been studied to overcome this problem, e.g. a combination with a VOF method has been used [87] or a re-distancing algorithm [86]. In our application area Iafrazi et al. used the level-set technique for unsteady free-surface flows, where results of flow over a jump and flow inside a tank have been shown [45]. The level set method

is an interface tracking method, where the interface is tracked explicitly. In interface capturing methods, the interface is resolved implicitly, no tracking function is used. Only conservation laws are used, see e.g. [6, 93].

In the Volume-of-Fluid (VOF) method, a VOF function F is introduced with values between zero and one, indicating the fractional volume of a cell that is filled with a certain fluid. Based on this volumetric data, the free surface is reconstructed and displaced, wherefore the method is termed a volume tracking method. The VOF method is extremely suitable in fixed grid simulation methods, where the free surface should be able to have an arbitrary complex topology. For example, in wave simulations the waves are sometimes overturning, such that the interface intersects itself and merges. The VOF-method automatically takes this into account. The earliest volume tracking methods were developed by Noh and Woodward [69], Hirt and Nichols [44] and Youngs [98]. Reviews of the different VOF-methods can be found in [78, 79].

The evolution of the VOF function is given by

$$\frac{DF}{Dt} = \frac{\partial F}{\partial t} + (\mathbf{u} \cdot \nabla)F = 0, \quad (1.1)$$

with $\mathbf{u} = (u, v, w)$ the velocity vector, t the time and ∇ the gradient operator. This equation states that the interface is moving with the liquid velocity. Every time step the interface is first reconstructed from the VOF data, after which it is advected using Equation 1.1. The different VOF methods are often classed by the features of the interface reconstruction algorithm and the advection of the interface. For the reconstruction step, three different methods can be distinguished: (1) a simple line interface calculation (SLIC), where the interface is said to be parallel to one of the coordinate axes, e.g. [69]; (2) SLIC with the possibility of a stair stepped interface within a cell, e.g. [44, 55]; and (3) piecewise linear interface calculation (PLIC), e.g. [40, 81, 98]. The PLIC method has become very popular in the last decade, because it results in a more accurate interface reconstruction.

In this thesis, both the SLIC method of Hirt and Nichols [44] and the PLIC method of Youngs [98] are used for the displacement of the free surface. To overcome problems in the original Hirt-Nichols' method, which are mass conservation problems and the occurrence of flotsam and jetsam (small droplets disconnecting from the free surface), a local height function is introduced. Both Hirt-Nichols' and Youngs' method with and without a local height function are used in standard kinematic tests and in a dambreak simulation to compare the performance of the different methods.

1.2.3 Moving objects

Moving objects in the domain can be accounted for in different ways. Commonly known in the fixed-grid field of methods are the immersed boundary method, the fictitious domain method and the cut-cell method. The first two methods are treating the boundaries of the object as a special region in a single phase. So, the whole domain is filled with liquid, and body forces (in cells containing the moving object boundaries) account for the presence of the moving objects. In the cut-cell method the object is solid and the sharp object boundary is cutting through the grid cells.

In the fictitious domain method, introduced by Glowinski [31], the flow computation is done on a fixed space region, which contains the moving objects, using a finite element

method. Lagrange multipliers are defined on the regions occupied by the rigid bodies to match the fluid flow and rigid body motion velocities over the interface between the regions. A variational formulation is derived involving Lagrange multipliers to force the rigid body motion inside the moving objects.

The immersed boundary method was originally developed by Peskin [73] for the calculation of blood flow through the heart. The heart is embedded in a larger periodic box that is completely filled with fluid. The interface between the fluid and non-fluid regions is defined using polynomial fitting through markers on the interface. To describe the extra forces in the non-fluid regions, a force is introduced that differs from zero only in these non-fluid regions. The interface force is spread to the nearby grid points using a discrete δ -function. This δ -function typically influences a band of four cells. The material properties are smoothed over the interface in a transition zone with a size of approximately two times the cell size.

The cut-cell method differs from the other methods in that the interface stays sharp and is not smeared over a few cell widths. A sharp interface method is needed in the applications studied in this thesis, since the peak of the water impact pressure should not be flattened due to smearing the interface over a few cell widths. In the current method the initial geometry is filled with markers with accompanying small rectangular volumes, such that the unity of all the volumes equals the object geometry. Then the markers are moved every time step according to the motion of the object. In a cut-cell method the cells can become arbitrary small when a large part of the cell is occupied by geometry. In case of fixed objects, due to the choices in the discretisation of the current method the small cells induce no extra limitations on the time step for stability [20, 92]. In the case of moving objects, a modification in the governing equations is needed to return to the same stability criterion for the time step [23].

A disadvantage of the cut-cell method is that the sudden changes of the nature of cells, from fluid to body cell and vice versa, introduce discontinuities. But by avoiding the smearing of the interface, the velocity of the fluid along moving objects, that is important in the applications at hand, is not smoothed over the object interface. Udaykumar et al. [91] use a cut-cell method for the simulation of flow with complex moving boundaries. To account for the changing nature of a cell from solid to fluid, in their method such a cell is merged with a neighbouring cell during that time step, such that the fluid kinematics in that cell are known.

1.3 Generation and propagation of waves

For the calculation of loads on offshore structures a wave generation option in the simulation method is essential. Some parts of the loads calculation can be done without the presence of waves. Part of the calculation of loads due to green water can be done without waves by modelling the water around the bow using a breaking dam model. This has been used in the early stage of the development of this simulation method [24] and also by e.g. [74, 97, 100]. Water entry simulation can be used to model the early stage of the green water phenomenon when the bow of the vessel enters the water [8]. When waves are present in the domain the phenomenon can be studied entirely. Also for the calculation of, for example, bow slamming, waves (especially steep waves) are essential.

There are three different possibilities to model wave generation in a simulation method like COMFLOW. First, the waves can be generated using a wave maker as is also done in a wave tank. The wave maker is modelled by a moving flap that can move horizontally and/or rotate about different axes. In general, this option is not very convenient in a numerical wave tank, except when exactly the same conditions need to be generated as in an experiment, of which the flap motions are known. The second option is to generate waves at an inflow boundary by prescribing velocities and water height. The velocities and wave height can be calculated using description methods of waves. The third option is to use another efficient simulation method that calculates the wave field, and prescribe velocities calculated by this efficient method at the open boundaries of the COMFLOW domain. The second and third option are elaborated below.

1.3.1 Generation of waves using wave description theory

A wave description theory is used to generate the waves at the inflow boundary of the domain. Note that at the inflow boundary positive and negative velocities can occur, so fluid can flow in and out. The term is used to indicate a boundary where the wave is generated. The wave is generated by prescribing velocities at the inflow boundary. Different kind of wave descriptions can be used to determine the velocities at the inflow boundary. The easiest is a linear wave description (Airy wave). But according to Le Méhauté [61], the range of suitability of linear theory in deep water is $H/\lambda < 0.0062$, with H the wave height and λ the wavelength. This means that only waves with a very small amplitude can be generated using linear theory accurately. In the application area of our interest the waves are nonlinear. To prescribe nonlinear waves, 5th order Stokes theory has been implemented [83]. Irregular waves can be generated by making a superposition of linear wave components. The superposition principle only holds for linear waves, so the accuracy can be insufficient when using this in nonlinear circumstances.

In head waves the wave is only generated at one boundary of the domain. At the other open boundaries a condition should be used, such that the wave can leave the domain undisturbedly. This is a difficult problem, because no information is present about the wave near the outflow boundaries. There are different options to prevent the wave from reflecting against the open boundaries into the domain. Givoli [30] gives an overview of the different outflow boundary conditions. First, a dissipation zone can be used, in which the wave is damped. Second, a non-reflecting boundary condition based on the wave equation can be used that determines the velocities at the outflow boundary. An example is the Sommerfeld boundary condition, where the wave velocity, which occurs in the wave equation, has to be chosen on forehand [84]. Both methods for letting the wave flow out of the domain undisturbedly are investigated in the current simulation program.

1.3.2 Wave field calculation by an external program

Waves can be generated using a calculated wave field by an external program. For example, the initial flow field can be calculated by means of a linear diffraction calculation and a conversion to the time domain. The linearised motion of the object is then known in advance as well. During the simulation, the incoming, diffracted and radiated fluid velocities are imposed on all the open boundaries. There are several advantages in using

this method. First, the simulation with COMFLOW can be limited to the close surroundings of the structure. The far wave field is calculated using the external code and imposed on the boundaries of the COMFLOW domain. Second, a long duration run can be performed with the fast external program, of which a time trace can be selected that is expected to give a critical event. This long duration run (typically three hours) cannot be performed with the COMFLOW program for calculation time reasons. Third, the simulation can be started with a fully developed wave field that is calculated by the external code. Also the motion of the structure is known in advance. Fourth, during the time domain simulation a good prediction of the velocities at the open boundaries of COMFLOW is given by the external code calculation.

This zonal modelling, where the domain is decomposed in a small COMFLOW domain and a large external domain is first developed using a linear diffraction code. An investigation should reveal how far this method can be stretched. The next step could be to prescribe the velocities from the incoming contribution using a higher order method. The radiated and diffracted velocity contributions and the vessel motion still follow from the linear code.

1.3.3 Wave generation for validation using an experiment

When an experiment where a wave is involved is used for validation of the simulation program, the exact wave conditions need to be generated. If the wave is not the same in simulation and experiment, the results in loading will be different and the validation results will be less valuable. There are different ways to imitate the conditions in a wave basin. First, the flap motions, with which the wave was generated in the experiment, can be used to prescribe the motion of flaps in the numerical wave tank. This seems to be a rather good option, but a large problem arises in practice. The distance from the wave maker to the structure in the wave is mostly several wave lengths. In a computational intensive program like COMFLOW this distance is too large to be able to accurately calculate the loads on the structure, since not enough computational cells can be put in the neighbourhood of the structure.

The second option is to generate the wave at the inflow boundary of the numerical wave tank using a wave description of linear or 5th order Stokes theory. The advantage is that the exact same wave can be prescribed as used in the experiment and that the inflow boundary can be put at any distance from the structure. The disadvantage is that the disturbance in the wave that is present due to the structure is not taken into account at the inflow boundary.

The third option is to use measurements of the wave elevation in front of the structure to prescribe the wave at the inflow boundary at the same distance from the structure. The measured wave elevation can be decomposed using Fourier analysis and the different components can be superposed at the inflow boundary. This is only valid in linear circumstances, so the error in the wave elevation should be investigated carefully. Another disadvantage is that the position of the inflow boundary is fixed at the position of the measured wave elevation.

1.3.4 Wave propagation

After the wave is generated at the inflow boundary of the computational domain, it propagates through the domain. The propagation of waves needs to be studied carefully, since the steepness and height of the waves determine the loads on the structures in the waves. A thorough wave propagation study with the current simulation program has been performed by Meskers [62]. He investigated the influence of grid sizes and time step and identified some issues that need to be taken into account for proper wave propagation. The influence of the boundary conditions at the free surface on the wave propagation is very large as was also concluded by Chan et al. [13]. So, valid boundary conditions need to be chosen that take care of accurate wave simulations. The accuracy of wave simulations is also influenced by the method for the displacement of the free surface. As described before, the method of Hirt and Nichols [44] and the method of Youngs [98], which are both Volume-of-Fluid methods, are used. Also a combination of these methods with a local height function is established; this combination ensures almost exact mass conservation. Further, attention has to be paid to the dissipation due to the artificial viscosity that could cause damping of the waves. The artificial viscosity is present due to the upwind discretisation of the convective term in the Navier-Stokes equations. The influence can be investigated by performing a simulation over a large number of periods in a large computational domain. The damping of the waves should be small, such that the loads on the structures in the waves are not underestimated.

1.4 Outline

In this thesis simulation of waves resulting in loading on offshore structures is presented. To describe the method to accomplish this task the thesis is outlined as follows. The first chapter is an introduction to the problem and the computational method. In the second chapter the simulation method is described without the presence of waves. Chapter 3 deals with the numerical aspects of wave simulation. In Chapter 4 the first steps in the development of a domain decomposition method for the generation of waves is presented. The last chapter, Chapter 5, contains a summary of the results. Conclusions are drawn and recommendations are made for future research.

Chapter 2 describes the method for fluid-flow simulation in a closed domain, so without the presence of waves. First, the mathematical model is described in Section 2.2. The governing equations are the continuity equation and the Navier-Stokes equations, which describe conservation of mass and momentum, respectively. The boundary conditions at the objects and the free surface are described. In Section 2.3 the spatial and temporal discretisation of the governing equations are discussed, and the stability of the resulting discretised equations is examined. Also the method for the determination of the discrete boundary conditions at the free surface is given.

The displacement of the free surface is discussed in Section 2.4. Two methods are described and compared using standard kinematic tests. The first method is the standard VOF method of Hirt and Nichols [44]. This SLIC method, which is efficient and easy to implement, has the drawback that flotsam and jetsam occur. To prevent this, a local height function is introduced. The second method that is discussed is Youngs' algorithm, which is a PLIC method. Besides the standard kinematic tests, where the velocity field

is prescribed, also a case study of a two-dimensional dambreak is presented. Section 2.5 shows that our treatment of the moving objects, using a cut-cell method, introduces numerical pressure spikes. A method to minimise the amount of spikes is discussed.

The simulation method is validated in Section 2.6 and Section 2.7. First, a dambreak simulation is performed, where the loading on a box in the domain is calculated. The computed pressures and water heights are compared with measurements. In Section 2.7 some water entry cases are studied and resulting free surface profiles and pressure time traces are compared with experimental results and theoretical predictions.

In Chapter 3 the numerical issues regarding the simulation of waves are discussed. In Section 3.2 the mathematical description of linear waves is given and the implementation of 5th order Stokes waves is shortly described. The theoretical values for velocities and wave height are used at the inflow boundary of the computational domain to generate waves. The issues at the open domain boundaries, which also include the outflow boundaries where waves should leave the domain undisturbed, are discussed in Section 3.3. The influence of the free surface velocity conditions and the displacement algorithm on the accuracy of wave propagation is shown in Section 3.4 and Section 3.5, respectively.

The validation of wave propagation is performed in Section 3.6. First, the waves are propagating without an object in the flow. The waves are two-dimensional of form. Regular waves are used and irregular wave events, where the waves are quite steep and/or high. Second, a fixed spar platform has been put into the waves, and the resulting wave loading is calculated and compared to experimental results. Finally, a floating FPSO is put into a high wave field resulting in green water on the deck. The water height on the deck and pressure time traces at the deck and the deck house are compared with measurements.

In Chapter 4 the waves are not generated within our simulation program, but the wave field is calculated by an external program that also provides the motion time traces of the object. Section 4.2 describes this domain decomposition and the interface between our simulation method and the external program. In Section 4.3 this method is used to simulate the propagation of an irregular wave. The same experimental results shown in Chapter 3 are used in Section 4.4 to compare this new method for wave generation with. Finally, green water loading on a floating FPSO is calculated using this method, so also the motion of the FPSO is prescribed from the external program, and again compared with experiments as in Chapter 3. In Section 4.5 the results are discussed and some recommendations for future research are given.

