Numerical simulation of two-phase flow in offshore environments

Wemmenhove, Rik

Publication date:
2008

Chapter 1

Introduction

1.1 Hydrodynamic wave loading

Offshore structures and sea-going vessels have encountered problems due to the presence of severe waves for a long time. The expanding offshore industry and the increase in sea transport lead to more and more interest in their behaviour during violent weather conditions. In these heavy storms, wave and ship motions can become so large that water flows onto the deck of a ship, the case of green water, studied extensively by Buchner [7]. Especially on Floating Production, Storage and Offloading vessels (FPSO's), where a lot of sensitive equipment is present on the deck, green water can cause a lot of damage. Also other complex free surface problems like slamming, runup, tank sloshing and splashing occur having their effect on the behaviour and survival of offshore structures. Given the fact that many offshore structures are positioned in areas with severe wave conditions, two hydrodynamic aspects regarding the wave loading are of importance. First, higher waves will cause higher loads on structures. Secondly, more rapidly changing wind conditions will cause increasing wave steepness, a parameter with a very large effect on stability of and impact loading on the structure, possibly resulting in severe damage. The physical phenomena accompanying extreme wave events are highly nonlinear in relation to the occuring wave elevations, and require new methods as a basis for prediction of the behaviour of the vessel and the water flow.

In this thesis a simulation method based on the Navier-Stokes equations is presented. The numerical method has been developed to simulate hydrodynamic wave loading on floating and moored structures in steep waves. To further improve the numerical method, the COMFLOW-2 Joint Industry Project has been initiated by MARIN (Maritime Research Institute Netherlands). The project is a cooperation between University of Groningen, Delft University of Technology, MARIN and Force Technology Norway and is supported by 20 companies from the offshore industry. The objective of the COMFLOW-2 project has been formulated as:

To develop a dedicated and well validated numerical tool for the offshore industry to study complex free surface problems, which is flexible in its application and has a coupling possibility to the other tools of participants.

Earlier work on the numerical method has been carried out by Gerrits [27], Fekken [24] and Kleefsman [34]. They developed ingredients of the numerical method for local flow phenomena with the presence of a free liquid surface, moving objects and several boundary conditions. Furthermore, various inflow and outflow conditions have been implemented to perform wave impact simulations and experimental validation for different test cases.
The emphasis of the COMFLOW-2 project is, aside from validation and further increase of robustness, on the modeling of two-phase flow and on the implementation of an absorbing boundary condition. In physical terms, the emphasis within the current research project is on the investigation of the following phenomena:

- Wave impact loading on floating structures
- Steep and overturning waves
- Internal sloshing in partially filled Liquid Natural Gas (LNG) tanks

To study these physical phenomena, a number of improvements in the numerical method have been scheduled:

- Adding the effects of air entrapment to better simulate the physics of wave impact
- Improving the simulation accuracy for steep and overtopping nonlinear waves
- Reducing the computational effort by zonal modeling and/or absorbing boundary conditions

The focus in this thesis is on the modeling of two-phase flow by including the effects of air entrapment and air entrainment. Entrapped air has a 'cushioning' effect during the most violent wave impacts [39]. The physics concerning air entrapment and air entrainment is introduced in section 1.2, while their effect on the modeling approach is described in detail throughout this thesis. The study of the effect of the second phase on the wave loading on fixed and floating offshore structures is a principal part of the current research. To improve the simulation of linear and nonlinear waves, an absorbing boundary condition is developed at Delft University of Technology [58].
The numerical results are validated on a number of model experiments which are introduced in section 1.3.

1.2 Air entrapment and air entrainment

The application of a two-phase flow model for wave impact is especially useful for the simulation of breaking waves, but also for the simulation of other free surface effects. Wave breaking occurs both at beaches and on the open sea (against offshore structures in particular), the latter case being of more interest for the present study. In figure 1.1 a number of two-phase flow

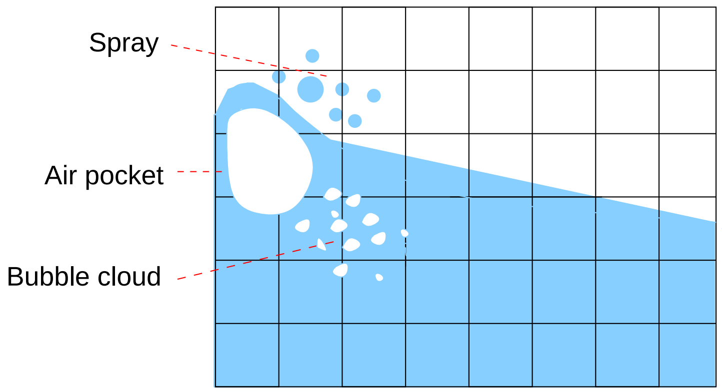


Figure 1.1: *Schematisation of two-phase flow phenomena for a breaking wave.*

phenomena are sketched that occur around a breaking wave.
Air pockets are the largest features, they occur as a result of the overtopping of the wave, either due to the wave steepness or due to the presence of an offshore structure. In confined flows, like sloshing flow, they also occur frequently. Air pockets can have a cushioning effect on peak pressure levels during wave impacts [39]. The size of these air pockets can vary greatly, but they generally have a short lifetime (generally less than one second [8, 18]).
Bubbles have much smaller dimensions, their diameter can be from millimeters to several centimeters. They are grouped in bubble clouds, having a much longer lifetime than air pockets, up to several wave periods.
Spray is located above the free surface, in contrast with air pockets and bubbles. It consists of small water particles and occurs on a large scale during wave breaking. Due to its relatively small water content, pressure levels on offshore structures as a result of spray alone are much lower than the water pressure levels below the free surface. Therefore, the focus of the current model with respect to two-phase flow phenomena is on air pockets and bubbles.
From a physical point of view, air pockets and bubbles can be distinguished by their difference in length scale ($O(10^2)$mm vs. $O(10^0)$mm). From a numerical point of view, this size in terms of one grid cell is important. Air pockets could be characterized by covering more than one grid cell, while individual air bubbles fit inside one grid cell.

The typical effect of the identified two-phase flow phenomena on the water level around and the loading on offshore structures is worthwhile to investigate. The effect of air pockets on the overall hydrodynamics is evaluated for the wave impact on a vertical wall. Figure 1.2 shows different stages in the development of an air pocket near a vertical wall. The pressure level variation during a violent wave impact is determined by the impact pressure (the high peak during stage 2) and the smaller reflecting pressure peak (during stage 4) [9, 63]. The reflecting pressure is caused by the water being accelerated to slow down at the wall as it falls back, primarily due to the gravity forcing. The reflecting pressure peak is lower, but longer-lasting than the impact pressure peak. It is, for example, observed at the wave run-up model experi-

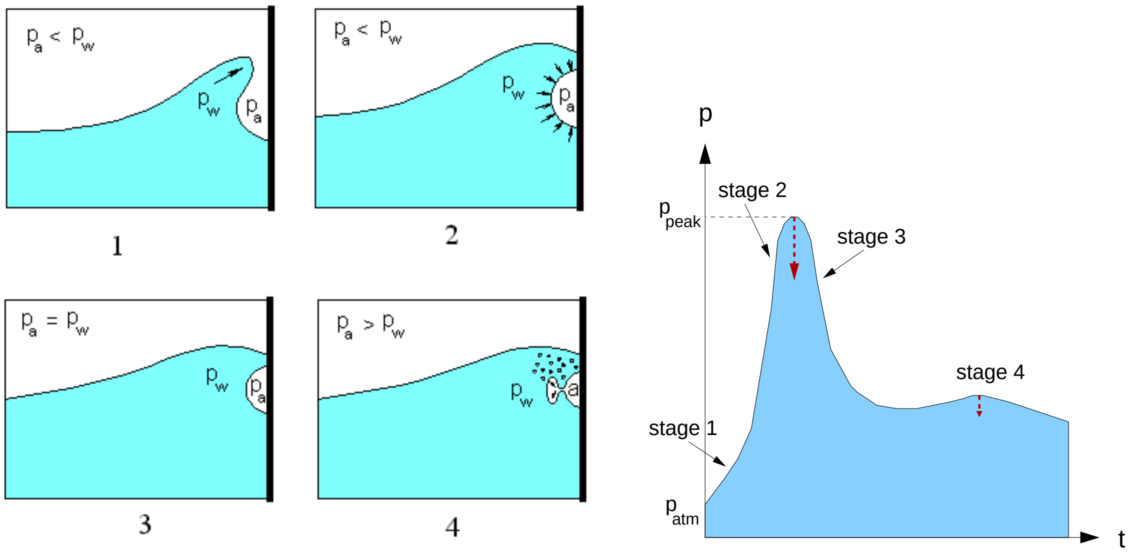


Figure 1.2: *Stages in the development of an air pocket (left) and schematic representation of the pressure development in the air pocket at the vertical wall (right). An air pocket is enclosed after stage 1, during stage 2 the air pocket is compressed by the surrounding water. After the impact, the wave flows back, yielding an equal pressure in air pocket and water (stage 3). Finally, the air pressure decreases, resulting in breakup of the pocket and bubble entrainment (4).*

ments described in Chapter 5. For highly-aerated flow, the pressure development is different: between impact and the reflecting peak the pressure shows a damped oscillating behaviour due to the stiffness of the bubbles with subatmospheric pressures of short duration [9]. The impact pressure is critical for offshore structures, but the height and duration of the reflective pressure peak affect the fatigue behaviour of the structure and have an effect on the next incoming wave. Regarding the pressure level during wave impact, the magnitude of the peak pressure level reduces in most cases due to the entrapment of air [8, 39]. The lowering effect of the air entrapment on the reflective pressure level has been estimated to be smaller [39].
Although the lifetime of an air pocket is only of the order of $1s$, the bubble cloud emerging from the collapse of the air pocket may exist for a much longer time. Such a bubble cloud consists of a large number of bubbles, with a strong variation in space and time scales. After a breaking wave event, enhanced turbulence levels persist for at least 10 wave periods [26]. With respect to the contribution of bubble clouds to pressure levels on offshore structures, there exist full-scale bubble cloud measurements for wave breaking on sea walls [8, 63] and laboratory bubble cloud measurements for wind-induced wave breaking [18]. The left of figure 1.3 shows for salt water laboratory measurements bubble size distributions during the first few seconds after wave breaking. As visible, a large air pocket has been entrapped and is surrounded by a bubble cloud. The air pocket collapses about $1s$ after initial entrapment, indicating the end of the ’acoustically active’ phase (see the right of figure 1.3). During the collapse of the air

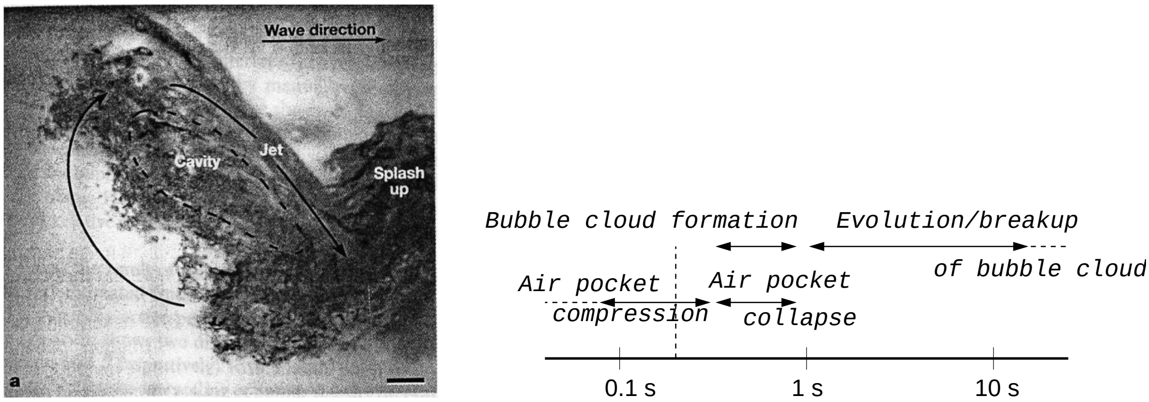


Figure 1.3: *Snapshot and time scales for the laboratory bubble cloud measurements of Deane et al. [18].*

pocket, large bubbles are created with bubble diameters up to 20*mm*. After collapse of the air pocket, bubble fragmentation occurs until the small bubbles reach the free surface or are no longer sustained by surface tension.

The described patterns for air entrapment and air entrainment concern rather ideal cases, as they describe the impact on a vertical wall and for laboratory conditions, respectively. For more complex offshore structures in deep water, which are subject to complex nonlinear waves, the given patterns and numbers for the air behaviour do not hold. To test offshore conditions in more realistic environmental conditions, numerical simulations, model experiments and/or full scale measurements are required. The experimental validation and the numerical method will be introduced in the following sections.

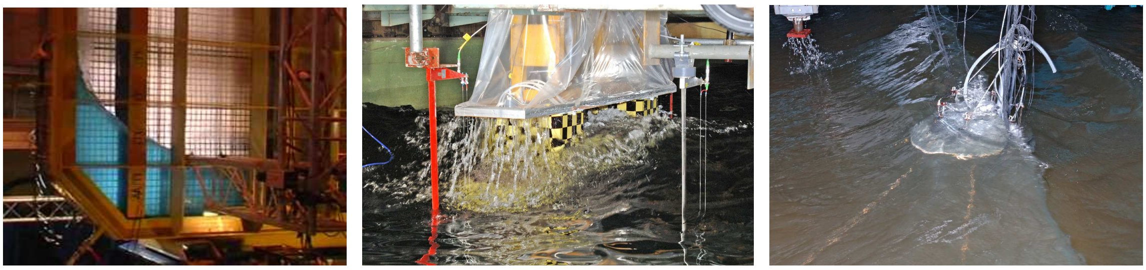
1.3 Experimental validation

Model experiments have been carried out by MARIN to provide validation material for the two-phase flow model and for the boundary condition implementation. Three experiments have been set up within the project:

- Large scale sloshing model tests
- Semi-submersible wave run-up tests
- Buoy model tests

Furthermore, a dambreak model experiment (to model green water flow on the deck of a ship) has been carried out within the preceding SAFEFLOW project [35].
The sloshing model tests investigate the fluid behaviour in partially-filled LNG tanks. The fluid

motion may lead to high impact loads on the tank walls, possibly causing damage to the insulation system. Furthermore, the global loads may affect the ship motions. The wave run-up tests examine the flow around a simplified semi-submersible two-column structure. Not only the pressure loads on, but also the wave propagation around the structure is of interest in this experiment. The buoy model tests measure the behaviour of a moored CALM buoy in extreme waves. The sloshing and wave run-up tests as well as the dambreak test are chosen for validation and are described in Chapter 5 with more detailed information about their geometry, flow conditions and the available measurement data. Figure 1.4 shows snapshots of each of these test cases. For every case, a number of parameters (like solid geometry, wave definition, initial



(a) Sloshing test (b) Wave run-up test (c) Buoy test

Figure 1.4: *Screenshots of the model experiments.*

liquid distribution) have been varied to generate an extensive set of validation data. Furthermore, all tests have been carried out for relatively long periods of time in order to have insight in the statistics and reproducability of the flow.

1.4 Simulation method

The numerical method has been incorporated in the computer program COMFLOW, which is based on the Navier-Stokes equations [34]. The two-phase flow model involving the modeling of both phases has been integrated in the existing one-phase flow version of COMFLOW. This one-phase version describes the motion of an incompressible, viscous fluid.

The numerical method has been developed originally to simulate liquid sloshing on board spacecraft [27]. For space applications, surface tension and an accurate description of the free surface are important, but gravity does not play a role. In the maritime area, the first applications being studied were sloshing in anti-roll tanks [54] and green water loading on the deck of a ship [25]. Later, the method has also been applied to floating objects, water entry of objects including ship launching [24] and wave loading on fixed structures [34].

By modeling two-phase flow, the free surface is no longer the boundary of the calculation domain but just the interface between both phases. On one hand this partly prevents difficulties with free surface boundary conditions, on the other hand it imposes new challenges as a discontinuity is introduced in the fluid properties across the interface.

1.4.1 Discretisation

The Navier-Stokes equations are solved for two phases on a fixed Cartesian grid. To this end, the aggregated-fluid approach is chosen, meaning that the behaviour of both fluids is described by one set of equations. This approach is chosen by many other two-phase flow methods [33, 43, 51, 64]. Only few other numerical methods use two sets of equations (one for each fluid), the segregated-fluid approach [29, 44, 47]. The fixed Cartesian grid is generated at the start of a simulation, with the cells being of rectilinear form. Given the presence of a solid geometry and/or moving bodies of non-rectilinear shape, grid cells can be cut by the solid geometry. Global stretching can be applied in every principal direction to have more resolution at locations where complex flow phenomena are expected.
Instead of using a Cartesian grid, another option is to construct an unstructured grid using a Lagrangian approach. However, aligning the grid with a moving interface results in a less transparent grid and is very difficult for highly distorted and rapidly moving free surfaces, as is the case in many offshore cases. Furthermore, unstructured grids need to be updated as soon as solid objects move through the grid. An alternative for grid-based methods could be the use of the Smoothed Particle Hydrodynamics (SPH) method. This meshless method puts a large number of particles in the flow, each with its own mass and velocity [14], which is however computationally expensive.
Due to its transparency and wide applicability, many two-phase flow methods use a fixed Cartesian grid [59]. The flow variables are staggered on the Cartesian grid as in the original Marker-and-Cell method [30], meaning that pressures and densities are defined in cell centers, while velocities are defined on cell faces. The control volumes for the mass conservation and momentum equation are located around cell centers and cell edges, respectively. The advantage of a staggered grid is that mass conservation can be applied easily in a cell, without the need of interpolations.
The conservation of kinetic energy gives constraints to the discretisation with respect to symmetry properties. For a symmetry-preserving discretisation, kinetic energy is dissipated only by diffusion [57]. Also, the compressibility of the second phase and the body force have their effect on the energy balance. Convection is discretised by a first-order or second-order upwind scheme, the former reducing the computational effort and the latter reducing the numerical viscosity inherent to upwind schemes. For the time discretisation the first-order Forward Euler method is adopted, which is replaced by the Adams-Bashforth method when the second-order upwind scheme is applied (see section 3.5.2). The Poisson equation for the pressure is solved implicitly by Successive Over-Relaxation [5] or by the preconditioned Krylov subspace method MILU [6].

1.4.2 Free surface and density

The free liquid surface present in offshore hydrodynamics can be regarded as an interface in the two-phase flow model. Various methods to describe the free surface can be found in literature, they describe the interface either in an Eulerian or in a Lagrangian way [45]. Using an

Eulerian method the free surface is described on a fixed grid, in contrast with the Lagrangian approach. The Level Set method [46] and Volume-of-Fluid method (used in this thesis) are the most popular ones, see figure 1.5. For the level-set formulation, the distance to the free surface is registrated, leading to an interface that is less 'sharp', thus making the simulation of a highly-distorted interface much more difficult [64]. Other options to describe the free surface are the Ghost Fluid Method [23], Immersed Boundary Method [37], particle methods and the use of a Riemann Solver [36, 44]. These methods are discussed in more detail in Chapter 4. The latter method is especially used when discontinuities like shock waves are present.
Using the Volume-of-Fluid method, a VOF function registrates the fractional volume of a grid

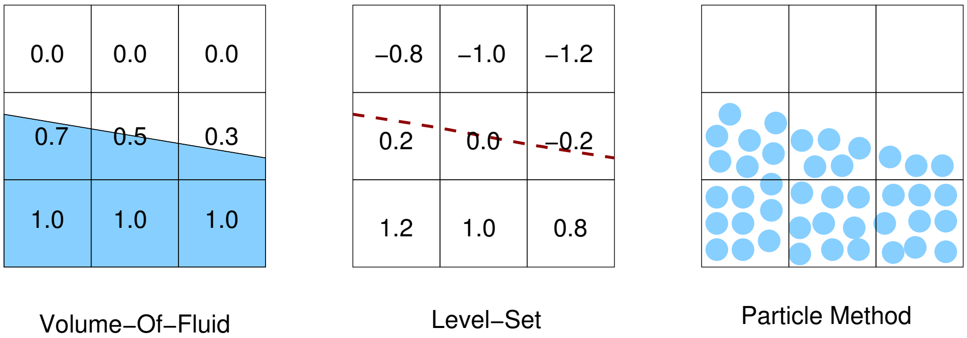


Figure 1.5: *Three methods to describe the free surface; Volume-Of-Fluid methods record the cell filling ratio, Level-Set methods calculate the distance to the free surface and Particle Methods fill the flow domain with liquid particles.*

cell filled with liquid. Based on the VOF function data, the free surface is first reconstructed and then displaced by the velocity field. For the reconstruction of the free surface, different options are available. Most used are the Simple Line Interface Calculation (SLIC) method of Hirt and Nichols [32] and the Piecewise Linear Interface Calculation (PLIC) method of Youngs [65]. Both have been implemented and to overcome mass conservation problems as well as flotsam and jetsam, a local height function has been introduced in [27, 55]. The local height function does not consider the VOF function of one grid cell only, but also the VOF function in the $3\times3\times3$-stencil around the grid cell.
By modeling the flow in both phases, the free surface is no longer the boundary of the calculation domain, so less boundary conditions are required, but on the other hand several variables have a large jump in their value across the interface. The density ratio between both phases can be up to a factor 1000, which is in particular visible in the time derivative and the convective term of the momentum equation. This may cause stability problems when the contribution of convective coefficients is amplified by the density ratio, resulting in the need for very small time steps. Aside from the density, also the pressure gradient shows a clear jump in its value across the free surface. By a careful discretisation of the terms in the Navier-Stokes equations containing densities (for example, by using the fact that there are clear jumps in density ρ and pressure gradient ∇p, but not in $\frac{1}{\rho}\nabla p$ across the interface), the computational costs due to the

density ratio effect are strongly reduced.
Given the staggered arrangement of variables on the grid and the requirements of mass conservation and conservation of kinetic energy, the density is defined in the cell centers, just as the pressure. However, for several terms in the momentum equation it is necessary to calculate the density along edges of grid cells. To determine these edge densities, a method to average the density of the adjacent cell centers is required. Different averaging methods have been studied; only by using a gravity-consistent averaging method (with an averaging stencil equal to a momentum cell) no spurious velocities are generated, see section 4.4.2.

1.4.3 Compressible air

The dynamical behaviour of entrapped air is described by means of a compressible flow model, which is especially important for violent flow conditions. In particular for larger air pockets with an oscillation period that cannot be neglected compared to the wave impact duration, a compressible flow model is needed [39].
During simulations, there are various situations in which air compressibility plays an important role. For simple 1D test cases it can be shown that incompressible air may hinder flow of the liquid phase, see section 5.2. For more complex geometries in two or three dimensions, an incompressible second phase especially causes problems for closed flow domains, such as tanks. The stiff incompressible air is frequently clustered near a liquid area under high pressure, leading to an unphysical pressure behaviour with pressure values strongly fluctuating in space and time. The oscillating pressure behaviour was also observed in [33], here it could be improved by modeling air compressibility. Including compressibility of the second phase allows an increase in density of the second phase in case of high pressures in the adjacent liquid, leading to a relaxation of the pressure by the 'cushioning' effect of the compressible air.
Thus far, only the requirement for entrapped air to be treated as compressible has been described. Entrained air, i.e. (groups of) small bubbles, cannot be reconstructed by the free surface algorithm, as the length scale of individual bubbles is typically smaller than the grid spacing. For some applications it is desirable to simulate the dynamics of these bubbles accurately as well. By recording all bubbles in a list, it is possible to advect them with the flow and to take buoyancy effects into account. The disadvantages of this bubble tracking method are the increased computational effort and the more intricate displacement algorithm, especially for complex geometries.

1.4.4 Boundary treatment

The flow domain is surrounded by boundaries, which are not always closed. Solid boundaries are subject to the no-slip boundary condition, meaning that there is zero velocity at the boundary.
As soon as boundaries are open for flow, the situation is getting more complex, since boundary conditions may have a disturbing effect on the fluid kinematics inside the flow domain. For simulations including waves, some of the wave components reflect against the outflow bound-

ary as long as a nonabsorbing boundary condition is applied. The disturbing effect of these 'wave reflections' can be minimized by putting the boundaries far away from the flow region of interest, but there are smarter methods that reduce the necessary amount of additional grid cells. One method that reduces the effect of reflections is the use of a numerical beach [34], however, this induces the need to extend the flow domain downstream of a studied object by at least one wavelength.
To keep the domain as small as possible while minimizing reflections at boundaries, an Absorbing Boundary Condition (ABC) is being developed [58]. Using this condition, the boundary can be located closer to the object of interest. In case of one-phase flow, additional difficulties arise near the outflow boundary since the velocity field is not continuous across the free surface. For two-phase flow, this difficulty vanishes as the velocity field is then calculated in all flow cells. Applying the Absorbing Boundary Condition reduces the required number of grid cells, making a finer resolution in the interesting flow areas possible.

1.5 Outline

The simulation method will be presented in more detail in the next chapters, together with its results for a number of test cases. Before describing the numerical discretisation, the Navier-Stokes equations and the free surface evolution are formulated in Chapter 2. In this chapter also the symmetry properties of the equations are investigated by means of a kinetic energy analysis. The 'basics' of the discretisation of the two-phase flow model are described in Chapter 3. Again, the Navier-Stokes equations are given, but now in their discrete form. The momentum equation can be formulated conservative or nonconservative, having a major impact on the stability of simulations, see section 3.2. The convective term, being part of the momentum equation, is formulated for different spatial discretisation schemes. In section 3.3, the convective term is given for both a first-order and a second-order upwind scheme. For the discretisation in time, two time integration methods are used. Together with the Poisson equation to calculate the pressure development in time they are described in section 3.5.
Chapter 4 is a follow up on the numerical discretisation, but in this chapter more attention is paid to the sensitive aspects of the two-phase flow model: the free surface and, directly related, the density. Some important aspects are the terms due to compressibility in the Poisson equation for the pressure (section 4.2), the mass conservation of both phases (section 4.3), the density averaging method (section 4.4) and the description of the free surface (section 4.5).
The numerical method has been validated for various test cases, and Chapter 5 shows the validation for test cases with different levels of complexity. The chapter starts with some test cases for which the numerical results are compared with analytical solutions, followed by test cases with more complex geometries and flows. Particular attention is paid to the validation with the experiments that have been carried out within the COMFLOW-2 project. Finally, some conclusions and suggestions for further research are given in Chapter 6.