Simulating Fluid-Solid Interaction Using Smoothed Particle Hydrodynamics Method

by

Kai Pan

B.S. Physics, University of Science and Technology of China (2008)
M.S. Physics, Massachusetts Institute of Technology (2012)

Submitted to the Department of Civil and Environmental Engineering
in partial fulfillment of the requirements for the degree of
Doctor of Philosophy in Computational Science and Engineering

at the

MASSACHUSETTS INSTITUTE OF TECHNOLOGY

February 2017

© Massachusetts Institute of Technology 2017. All rights reserved.
DISCLAIMER NOTICE

Due to the condition of the original material, there are unavoidable flaws in this reproduction. We have made every effort possible to provide you with the best copy available.

Thank you.

The images contained in this document are of the best quality available.
Simulating Fluid-Solid Interaction Using
Smoothed Particle Hydrodynamics Method

by
Kai Pan

Submitted to the Department of Civil and Environmental Engineering
on December 15, 2016, in partial fulfillment of the
requirements for the degree of
Doctor of Philosophy in Computational Science and Engineering

Abstract

The fluid-solid interaction (FSI) is a challenging process for numerical models since it requires accounting for the interactions of deformable materials that are governed by different equations of state. It calls for the modeling of large deformation, geometrical discontinuity, material failure, including crack propagation, and the computation of flow induced loads on evolving fluid-solid interfaces. Using particle methods with no prescribed geometric linkages allows high deformations to be dealt with easily in cases where grid-based methods would introduce difficulties. Smoothed Particle Hydrodynamics (SPH) method is one of the oldest mesh-free methods, and it has gained popularity over the last decades to simulate initially fluids and more recently solids. This dissertation is focused on developing a general numerical modeling framework based on SPH to model the coupled problem, with application to wave impact on floating offshore structures, and the hydraulic fracturing of rocks induced by fluid pressure.

An accurate estimate of forces exerted by waves on offshore structures is vital to assess potential risks to structural integrity. The dissertation first explores a weakly compressible SPH method to simulate the wave impact on rigid-body floating structures. Model predictions are validated against two sets of experimental data, namely the dam-break fluid impact on a fixed structure, and the wave induced motion of a floating cube. Following validation, this framework is applied to simulation of the impact of large waves on an offshore structure. A new numerical technique is proposed for generating multi-modal and multi-directional sea waves with SPH. The waves are
generated by moving the side boundaries of the fluid domain according to the sum of Fourier modes, each with its own direction, amplitude and wave frequency. By carefully selecting the amplitudes and the frequencies, the ensemble of wave modes can be chosen to satisfy a real sea wave spectrum. The method is used to simulate an extreme wave event, with generally good agreement between the simulated waves and the recorded real-life data.

The second application is the modeling of hydro-fracture initiation and propagation in rocks. A new general SPH numerical coupling method is developed to model the interaction between fluids and solids, which includes non-linear deformation and dynamic fracture initiation and propagation. A Grady-Kipp damage model is employed to model the tensile failure of the solid and a Drucker-Prager plasticity model is used to predict material shear failures. These models are coupled together so that both shear and tensile failures can be simulated within the same scheme. Fluid and solid are treated as a single system for the entire domain, and are computed using the same stress representation within a uniform SPH framework. Two new stress coupling approaches are proposed to maintain the stress continuity at the fluid-solid interface, namely, a continuum approach and stress-boundary-condition approach. A corrected form of the density continuity equation is implemented to handle the density discontinuity of the two phases at the interface. The method is validated against analytic solutions for a hydrostatic problem and for a pressurized borehole in the presence of in-situ stresses. The simulation of hydro-fracture initiation and propagation in the presence of in-situ stresses is also presented. Good results demonstrate that SPH has the potential to accurately simulate the hydraulic-fracturing phenomenon in rocks.

Thesis Supervisor: John R. Williams
Title: Professor of Information Engineering
Acknowledgments

This thesis marks the end of a challenging but fruitful journey. The experience in the past four years in every aspect of my life is profoundly memorable, and my heart is filled with nothing but gratitude.

First of all, I would like to express my deepest thanks to my advisor Professor John Williams for his continuous support, numerous encouragement and great guidance whenever I needed it during my doctoral study at MIT. Especially, I have to thank him for his patience with me and trust in me all the time. I would say Professor Williams has been an excellent mentor. He gave me enough freedom to pursue my own ideas towards research and shared his extensive knowledge in the field of numerical modeling and geo-mechanics with me. This work cannot be done without his help and support. It's my honor and fortune as well as pleasure to be his student.

I would also like to thank my committee members Professor Markus Buehler and Professor Nafi Toksoz for their effort, constructive comments, and helpful suggestions to improve the quality of my research work.

I am deeply thankful to other members in the geonumerics group, Bruce Jones, Ranjan Pramanik, Abdulaziz Albaiz, Thomas Douillet-Grellier, Seonkyoo Yoon, Samuel Raymond, Zeid Alghareeb, Justin Montgomery, Mohamad Sindi and others, who are like blood brothers to me and made my PhD life joyful and memorable. They have consistently encouraged me in the journey of realizing my dream.

I am grateful for the friendship with many people at MIT. Sharing life with them has been invaluable. I wish the best of luck to all of them.

Finally, my special acknowledgement is reserved for my parents, Heping Pan and Li Zhang, who love me all along and provide me with the motivation to pursue a PhD. They have been a constant source of spirit support and encouragement for me. Without their strongest support, I cannot carry out the difficult task of PhD studies.
Contents

List of Figures v

1 Introduction 1
   1.1 Dissertation outline ....................................... 3

2 SPH modeling of interaction between fluids and rigid-body solids 7
   2.1 Introduction .................................................. 7
   2.2 Methodology .................................................. 8
      2.2.1 The SPH method in brief ................................. 8
      2.2.2 SPH approximation of the governing equations .......... 10
      2.2.3 Equation of state ....................................... 11
      2.2.4 Boundary condition ...................................... 12
      2.2.5 Rigid-body dynamics ................................... 14
   2.3 Test cases ................................................... 15
      2.3.1 Three dimensional dam break flow against a tall structure .. 16
      2.3.2 Floating body dynamics under the action of induced wave .. 20
   2.4 Application: Wave impact on a off-shore oil platform .......... 24
   2.5 Conclusions ................................................ 34

3 SPH modeling of extreme waves 35
   3.1 Introduction ................................................ 35
   3.2 Methodology ................................................ 37
### 3.2.1 Generation of a single linear wave mode with a piston wave maker in 2D

---

### 3.2.2 Generation of a single linear wave mode in any direction with a curved wave maker in 3D

---

### 3.2.3 Generation of multiple wave modes

---

### 3.2.4 Wave spectra for realistic sea states

---

### 3.2.5 Extreme wave focusing

---

### 3.2.6 Implementation in SPH formalism

---

### 3.3 Numerical Examples of extreme wave modeling

---

### 3.4 Extreme wave on tension-leg platform

---

### 3.5 Conclusions

---

### 4 SPH for solid mechanics

#### 4.1 Introduction

---

#### 4.2 Methodology

---

##### 4.2.1 Constitutive model

---

##### 4.2.2 Governing equations and corrective terms in SPH

---

##### 4.2.3 Failure models

---

##### 4.2.4 Stress boundary conditions

---

##### 4.2.5 Velocity boundary conditions

---

#### 4.3 Numerical examples

---

##### 4.3.1 Line Crack

---

#### 4.4 Conclusions

---

### 5 SPH modeling of fluid and deformable-solid coupling

#### 5.1 Introduction

---

#### 5.2 Methodology

---

##### 5.2.1 Fluid model

---

##### 5.2.2 Solid model

---

---
List of Figures

2.1 Support domain of particle $a$ and weight function. .................. 8
2.2 Schematic diagram of the dam geometry (all dimensions in m) .......... 17
2.3 Evolution of the water collapse and interaction with the column at
 various time steps ........................................ 18
2.4 Force calculation in 3D dam break case; comparison between SPH
 simulation result and experimental data. ................................ 20
2.5 Sketch of the plan view of the central part of the wave flume with the
 floating breakwater set up. ....................................... 21
2.6 Wave flume test with a pile-moored floating breakwater. Image repro-
 duced from Manenti et. al [1]. ......................................... 22
2.7 Side view of SPH simulation of the wave flume test with a pile-moored
 floating breakwater at $t = 5$ s. ..................................... 22
2.8 Comparison between experimental data and numerical results for the
 floating breakwater .................................................. 23
2.9 Geometrical setup of three-dimensional simulation of an extreme wave
 hitting a platform. .................................................. 25
2.10 Close-up of the platform model. ....................................... 25
2.11 Total force from eight tension legs, as a function of the horizontal
 displacement of the platform, for the realistic platform geometry. .... 28
2.12 Six snapshots of the SPH simulation with a moving platform: 3D view.
 The fluid is colored by velocity magnitude. The solid side walls are not
 shown for better visualization. ......................................... 29
2.13 Six snapshots of the SPH simulation with a moving platform: Zoomed view of platform. ................................................................. 30

2.14 Results from the SPH simulation with a moving platform. (a) displacement of the platform’s center-of-mass in x-direction (Surge), y-direction (Sway) and z-direction (Heave). (b) Position of the platforms center-of-mass in the xz-plane during the simulation. .................. 32

2.15 Results from the SPH simulation with a moving platform. Left: Forces exerted by the fluid on the platform in x-direction, y-direction and z-direction. Right: Forces exerted by the tension legs on the platform in x-direction, y-direction and z-direction. .................. 33

2.16 Results from the SPH simulation with a moving platform: Platform rotation as a function of time. ................................................................. 33

3.1 A schematic diagram of progressive waves along any direction generated by a curved wave maker (red line, denoted by $x^*(y')$). .................. 39

3.2 ‘New Year Wave’ elevation profile. Black solid line is measured data. Red solid line is SPH simulation result. ................................. 46

3.3 Simulation geometry. ................................................................. 47

3.4 Measured wave spectrum (black) and fitted spectrum (red) for ‘New Year Wave’. ................................................................. 48

3.5 Top and side view of the SPH simulation of the New Year Wave at the focus point right in front of the platform at time $t=60s$. .................. 49

3.6 SPH simulation of an extreme wave hitting a tension-leg platform at various time. a) time = 52 s; b) time = 56 s; c) time = 60 s; d) time = 64 s. ................................................................. 50

3.7 Vertical displacement (“Heave”) and the horizontal displacement (“Surge”) of the tension-leg platform under the hit of an extreme wave event in SPH simulation. ................................................................. 52
4.1 Drucker-Prager model and the yield surface cone .......................... 57
4.2 Cohesion as a piecewise linear function of accumulated plastic strain. ... 59
4.3 Schematic stress boundary conditions with dummy particles .......... 62
4.4 Schematic stress boundary conditions without dummy particles ....... 63
4.5 Schematic of the line crack problem ........................................... 65
4.6 Truncation of the support domain for crack boundary particles and their neighbors. (a) Normal case, (b) Truncated case around the crack 67
4.7 SPH results of the line crack problem submitted to a constant compressive vertical pressure of 15MPa (a) Vertical Stress (Pa) (b) Vertical Displacement (mm) .......................................................... 68
4.8 Comparison of vertical stress between SPH result (x) and analytical solution (-) of the elastic line crack problem with in-situ pressure P = 15 MPa. ................................................................. 68
4.9 SPH result showing vertical stress distribution at various time steps in fracture propagation in the line crack problem with fracture pressure P = 50 MPa. ................................................................. 70
5.1 Schematic of fluid and solid domain in fluid-solid coupling. ................. 76
5.2 Schematic diagram of the hydrostatic validation model. ...................... 80
5.3 Hydro-static pressure in fluid phase (right) and geo-static stress in solid phase (left) of the simulation, compared with the analytical solution (solid line). Results from both continuum approach (x) and stress-boundary-condition approach (o) are plotted. ......................... 81
5.4 (a) Schematic of the circular opening problem. (b) Particle packing for fluid particles (red) inside hole and solid particles (blue) as solid plate. ....... 82
5.5 Horizontal stress $\sigma_{xx}$ (a) and vertical stress $\sigma_{yy}$ (b) along vertical axis ($x = 0$) of the simulation, compared with the analytical solution (-). Results from both continuum approach (o) and stress-boundary-condition approach (x) are plotted. ................................. 84
5.6  Schematic diagram of the hydraulic fracture model without in-situ stress. 85

5.7  Maximum positive principle stress at the fracture tip immediately before fracture propagation begins ($t=0.01s$) in the case of hydraulic fracturing simulation without in-situ stress. 86

5.8  Fracture propagation in a coupled fluid-solid model of a hydraulic fracture without in-situ stress. 87

5.9  Schematic diagram of the hydraulic fracture model with in-situ stress. 89

5.10 Maximum positive principle stress at the fracture tip immediately before fracture propagation begins ($t=0.012s$) in the case of hydraulic fracturing simulation with in-situ stress. 90

5.11 Fracture propagation in a coupled fluid-solid model of a hydraulic fracture with in-situ stress. 91
Chapter 1

Introduction

In fluid-structure interaction (FSI) problems, fluid flows interact with solid structures driving their motion and sometimes leading to failure if the forces exerted on the solid exceed the material strength. FSI problems arise in almost all fields of engineering and pose significant challenges. In most cases, analytical solutions to the model don’t exist owing to its strong nonlinearity and multidisciplinary nature [2–4]. Thus, it is necessary to employ numerical methods to investigate the fundamental physics involved in the complex interaction between fluids and solids.

Many numerical methods have been used and developed in the study of FSI problems. Among these methods, Arbitrary Lagrangian-Eulerian (ALE) method and Immersed Boundary (IB) method are the most commonly used. An ALE method allows arbitrary motion of grid/mesh points with respect to their frame of reference by taking the convection of these points into account as described in [5–7] and many works thereafter. In an ALE formulation, the finite element mesh need not adhere to the material to be fixed in space but may be moved arbitrarily relative to the material. Alternative to ALE technique, IB methods [8–10] consider a finite difference grid for the fluid domain with an immersed set of non-conforming boundary points that are mutually interconnected by an elastic law. The local body forces at the position of the solid points are computed to model the interaction between fluid and solid boundary.
The velocity and stress continuity at the interface are also maintained through the kinematic constraint imposed by this body force.

These methods can successfully predict the FSI process in one aspect or another [11]. However, these grid-based methods do not typically work well for simulation of large deformations, fractures, and fragmentation, especially if discontinuities occur in the solid failure process induced by fluid loads. The most widely used strategy to handle a moving discontinuity is to remesh the domain in each step such that the discontinuity is accurately represented in the simulation mesh and exists along the boundaries of mesh elements. However, this approach introduces numerical difficulties and can be computationally expensive. For example, due to the representation of cracks, artificial crack paths may emerge due to prescribed mesh linkages. In addition, the coupling of one numerical method for fluids and another for solids can introduce numerical errors from inconsistencies and prove difficult for code maintenance.

Using particle methods with no prescribed geometric linkages (such as in a mesh or a grid) allows high deformations to be modeled more easily in cases where grid-based methods would require expensive re-meshing. Smoothed Particle Hydrodynamics (SPH) method is one of the oldest mesh-free methods and it has gained popularity in recent decades to simulate both fluids and solids [12-22]. SPH is a particle method based on a Lagrangian discretization of partial differential equations. It was originally developed for simulating astrophysical problems [23] and was later applied to continuum solid and fluid mechanics [13]. SPH uses an updated Lagrangian formulation to evolve the state of the particles. In contrast to molecular dynamics where forces are determined by direct molecule to molecule potentials, SPH uses a continuum state equation to determine the internal stresses, which drive the evolution of the particle kinematics. Due to the Lagrangian nature of the method, particles are advecting according to its governing physics equation, which makes the simulation of large deformation and interface tracking relatively straightforward. For instance, in a free-surface flow problem, the fluid surface can be automatically captured without
special treatment. The same applies in the modeling of large solid deformation with fracture where multiple cracks emerge naturally from the failure model. For these reasons, the SPH method continues to gain traction in the field of FSI modeling. This dissertation is focused on developing a general numerical modeling framework based on SPH to model the coupled problem, with application to wave impact on floating offshore structures, and the hydraulic fracturing of rocks induced by fluid pressure.

1.1 Dissertation outline

In Chapter 2, a general SPH methodology is demonstrated and validated in modeling the interaction between fluid and rigid-body solids. For the interest of engineering application, a numerical modeling of real extreme sea waves and its impact on a tension-leg platform is presented in Chapter 3. Chapter 4 studies the SPH modeling of solid deformation which serves as the cornerstone of further more complicated modeling by coupling with fluid. Followed by that, a general numerical framework of coupling fluid and deformable solid with fracture initiation and propagation is presented in Chapter 5 along with its application in a hydraulic fracturing modeling. The following summary provides a brief outline of this dissertation:

- Chapter 2: This chapter investigates the interaction between fluid and rigid-body floating structures. Here, the fluid-structure interaction is considered using the weakly compressible smoothed particle hydrodynamics (SPH) method. To ensure the applicability of this method, numerical prediction of fluid forces and rigid-body motion is validated against two sets of experimental data which are, impact due to dam-break, and wave induced motion of a floating cube. For the dam break problem, the SPH method is used to predict impact forces on a rectangular column located downstream. In the second case of a floating cube, the SPH method simulates the motion of a buoyant cube under the action of in-
duced waves, where a wall placed upstream of the cube is displaced sinusoidally to induce waves. In both cases, the SPH framework implemented is able to accurately reproduce the experimental results. Following validation, the SPH modeling is applied to a simulation of a toy model of a tension leg platform upon impact of a large solitary wave. This analysis shows that the platform may be pulled into the water by stretched tension legs, where the extension of the tension legs also governs the rotational behavior of the platform. The result indicates that SPH has good potential in handling the simulation of fluid and floating rigid-body interaction.

- **Chapter 3**: In engineering application, an accurate estimate of forces exerted by real extreme sea waves on offshore structures is vital to assess potential risks to structural integrity. This chapter describes a method to simulate multi-modal and multi-directional sea waves with SPH. The waves are generated by moving the side boundaries of the fluid domain according to the sum of random Fourier modes, each with its own direction, amplitude and wave frequency. By carefully selecting the amplitudes and the frequencies, the ensemble of wave modes can be chosen to satisfy a standard sea wave spectrum. The method is used to simulate an extreme wave event, with generally good agreement between the simulated waves and the recorded real-life data. The potential of the method for practical situations is illustrated by a simulation of the impact of an extreme wave on a tension-leg platform.

- **Chapter 4**: SPH simulation of the initiation and propagation of pressure-driven fractures in brittle solids is presented in this chapter. An elasto-plastic damage model is employed to model the solid. Drucker-Prager plasticity model is used to predict material shear failures due to plastic deformation. For tensile failure, the Grady-Kipp damage model is used. These models are coupled together so that both shear and tensile failures can be simulated within the same scheme. A new method of applying stress boundary condition and velocity boundary
Chapter 1. Introduction

condition both with and without dummy particles is developed and validated in the work. Results show that SPH is able to correctly predict the evolution of fracture in brittle solids. The model is also applied to the solution of crack propagation in a "line crack" problem. The influence of initial in-situ stresses is also accounted for. Comparison of SPH results for these cases to analytical solutions shows that SPH may be applied to accurately simulate the evolution of fluid-driven fractures in brittle solids.

• Chapter 5: The modeling of solid deformation induced by fluid loads is another critical issue in the simulation of FSI problems. SPH method is suitable for simulating this problem as the method has advantages in handling large structure deformation under a high pressure load thanks to its Lagrangian nature. This chapter devises a general SPH numerical modeling framework to model the interaction between fluids and brittle solids with highly nonlinear deformation and dynamic fracture initiation and propagation. SPH modeling of fluid and elastoplastic-solid coupling is presented. Fluid and solid are treated as a single system for the entire domain, and are computed using the same stress representation within a uniform SPH framework. Two stress-coupling approaches are proposed to maintain the stress continuity at the fluid-solid interface, namely, continuum approach and stress-boundary-condition approach. Both approaches allow interactions between particles from different materials when solving the momentum equation. A corrected SPH approximation form of the density continuity equation is implemented to handle the density discontinuity of the two phases at the interface. The method is applied to simulate rock deformation in a variety of rock mechanics test cases, including a hydrostatic problem and a fluid-pressurized borehole case in the presence of in-situ stresses where analytic solutions are available. The simulation of fracture initiation and propagation in the hydrofracturing case with and without in-situ stresses are also presented as a real world application. Good results demonstrate that SPH has the potential
to accurately simulate the coupling of fluid and deformable solids.
Chapter 2

SPH modeling of interaction between fluids and rigid-body solids

2.1 Introduction

Smoothed Particle Hydrodynamics (SPH) is a mesh-free, fully Lagrangian method for numerical simulation of fluid flow [12-15]. In the SPH method, a computational domain is discretized by a set of points, or particles, and a meshless discretization scheme is used to represent a scalar or vector field in terms of its values at these points. Since the particles are advected with the flow field, the SPH is ideally suited to problems of free surface flow, where computationally expensive surface tracking, typically required by continuum approaches, is not necessary [12].

Before applying the SPH method to real world problems, the SPH simulation is demonstrated to accurately predict the impact force on fixed structures and rigid-body motion. A three-dimensional dam break case is simulated and the calculated load on a tall column is compared against those measured during an experiment by Yeh & Petroff [24]. This configuration is proved to be a popular test case for bench-
marking free surface flow algorithms, and appears as a validation case to indicate the
ability for the SPH method to capture loading characteristics in violent gravity driven
fluid-structure interactions [25, 26]. Following analysis of fixed structure impacts, the
motion of a floating body, under the action of induced waves [1, 27] is considered.
The SPH method accurately reproduces the experimental data, demonstrating the
advantages of the SPH in simulating free-surface-flow and floating structure interac-
tion. Finally, as a proof of concept, our SPH framework is applied to simulate the
effect of a solitary wave on a tension-leg platform. The force acting on the platform
is calculated and analyzed under the combined effect of a large wave and stretched
tension legs. This simulation can serve as a blueprint for more advanced simulations
in the future where the platform model may include more detail, and the initial state
of the fluid phase may be more representative of conditions observed in oceans.

2.2 Methodology

2.2.1 The SPH method in brief

![Diagram showing smoothing kernel W, current particle a, and support domains.]

Figure 2.1: Support domain of particle a and weight function.
Chapter 2. *SPH modeling of interaction between fluids and rigid-body solids*  

Through the SPH technique, the continuum is represented as a set of discrete particles, characterized by their own physical properties such as mass, density, and pressure [12–13]. In SPH, the fundamental principle is to approximate any field $A(r)$ by an integral interpolant based on a weight function or smoothing function or smoothing kernel as:

$$A(r) = \int A(r')W(r - r', h)dr'$$  \hspace{1cm} (2.1)

where $W(r - r', h)$ is the weighting function or kernel and $h$ is the weighting function smoothing length. This weighting function required to be continuous and differentiable and satisfy the normalization, delta function and compactness properties.

Each particle has a support domain, $\Lambda_a, \forall a \in \Omega$, specified by the smoothing length $h$ (Fig. 2.1). The value of a function at a typical particle is obtained by interpolating values of that function at all particles in his support domain weighted by smoothing function. In discrete notation, this becomes,

$$A(x_a) = \sum_{b \in \Lambda_a} \frac{m_b}{\rho_b} A_b W(x_a - x_b, h)$$  \hspace{1cm} (2.2)

where the summation is over all particles, $b$, within $\Lambda_a$ the compact support domain of the kernel function $W$ at particle $a$. $m_b$ and $\rho_b$ are respectively the mass and the density of particle $b$.

The gradient of the function $A$ at the position of the particle $a$ is given by differentiating kernel $W$ in Eq. (4.5)

$$\nabla A(x_a) = \sum_{b \in \Lambda_a} A_b \frac{m_b}{\rho_b} \nabla W(x_a - x_b, h).$$  \hspace{1cm} (2.3)

where

$$\nabla W(x_a - x_b, h) = \frac{\partial W_{ab}}{\partial x_a} = \left(\frac{x_a - x_b}{r}\right) \frac{\partial W_{ab}}{\partial r}$$  \hspace{1cm} (2.4)

with $r$ is the relative distance between particle $a$ and $b$ and its defined as $r = |x_a - x_b|$.
There are many different interpolating kernel or smoothing functions available in SPH. To give an example, the quintic spline kernel function following Morris [28] is given below,

\[
W_{ab} = \alpha_d \times \begin{cases} 
(3 - R)^3 - 6(2 - R)^2 + 15(1 - R)^5 & 0 \leq R \leq 1; \\
(3 - R)^3 - 6(2 - R)^2 & 1 \leq R \leq 2; \\
(3 - R)^3 & 2 \leq R \leq 3; \\
0 & 3 \leq R.
\end{cases} \tag{2.5}
\]

where \( R = |\mathbf{r}_i - \mathbf{r}_j|/h \) and \( \alpha_d = 120/h, \alpha_1 = 7/(478\pi h^2), \alpha_2 = 3/(359 h^3) \) in 1,2 and 3 dimensions respectively.

### 2.2.2 SPH approximation of the governing equations

Determination of particle velocity is achieved through discretization of the Navier-Stokes conservation of linear momentum equation,

\[
\frac{D\mathbf{v}_a}{Dt} = -\frac{1}{\rho_a} \nabla p + \mathbf{g} + \Theta \tag{2.6}
\]

where \( p \) is the pressure, \( \mathbf{g} \) is the gravity acceleration and \( \Theta \) is the dissipative term.

In SPH notation, Eq.2.6 can be written as,

\[
\frac{D\mathbf{v}_a}{Dt} = -\sum_b m_b \left( \frac{p_a}{\rho_a^2} + \frac{p_b}{\rho_b^2} + \Pi_{ab} \right) \nabla_a W_{ab} + \mathbf{g} \tag{2.7}
\]

in which \( p_a \) and \( \rho_a \) are respectively the pressure and density of particle \( a \) (the same applies to particle \( b \)). \( \Pi_{ab} \) represents an artificial viscosity model, where we use the model proposed by Monaghan [12] which is defined as,

\[
\Pi_{ab} = \begin{cases} 
\frac{\alpha_{cd}}{\rho_b} \frac{\mathbf{v}_{ab} \cdot \mathbf{r}_{ab}}{r_{ab}^2 + \epsilon h^2} & \mathbf{v}_{ab} \cdot \mathbf{r}_{ab} < 0; \\
0 & \mathbf{v}_{ab} \cdot \mathbf{r}_{ab} \geq 0.
\end{cases} \tag{2.8}
\]

where \( \mathbf{r}_{ab} = \mathbf{r}_a - \mathbf{r}_b, \mathbf{v}_{ab} = \mathbf{v}_a - \mathbf{v}_b \); being \( \mathbf{r}_k \) and \( \mathbf{v}_k \) the position and velocity corresponding to particle \( k \) (a or \( b \)); \( \bar{c}_{ab} = (c_a + c_b)/2, \epsilon = 0.01 \). The artificial
viscosity coefficient, $\alpha$, has the main purpose of preventing instability and spurious oscillations in the numerical scheme [26,29,30]. The value of $\alpha = 0.1$ is adopted in this work unless stated otherwise. Eq. (2.7) is used to update the accelerations of fluid particles.

Changes in the fluid density are calculated by means of discretizing the mass continuity equation,

$$\frac{D\rho}{Dt} = -\rho \nabla \cdot \mathbf{v}$$

which in SPH formalism gives:

$$\frac{D\rho_a}{Dt} = \sum_b m_b (\mathbf{v}_a - \mathbf{v}_b) \cdot \nabla_a W_{ab}$$

where $\nabla_a W_{ab}$ is the gradient of the kernel function. Compared to another density computation approach using weighted summation of mass terms [31], the method described in Eq (2.10) is more suitable for free surface flow simulation as it prevents artificial density decrease near boundaries and near free surfaces [12,29].

### 2.2.3 Equation of state

Acceleration depends on the pressure, which, for a weakly compressible fluid, is usually specified by an equation of state [32] of the form,

$$P = \frac{c_0^2 \rho_0}{\gamma} \left[ \left( \frac{\rho}{\rho_0} \right)^\gamma - 1 \right]$$

where $\rho_0$ is the reference density of the fluid and $c_0$ is the speed of sound at the reference density, $p$ is the pressure. In the calculations to be described here, $\gamma = 7$ [12]. In the numerical scheme the speed of sound $c_0$ is is set equal to at least 10 times the maximum velocity expected in order to obtain reasonable time steps and also keep the density variation within 1% compared to the reference density [13].

Time-step $\Delta t$ is calculated according to the Courant-Friedrichs-Lwey condition.

The numerical simulation are updated in time using a verlet time integration scheme [29].
2.2.4 Boundary condition

In the SPH literature, various methods exist for the discretization of solid boundary conditions [33]. A brief overview of these methods are listed below:

- **Dynamic boundary condition**: In the dynamic boundary condition method [34], boundary particles satisfy the same equation of continuity and of state as the fluid particles, but their position remains unchanged (fixed boundaries) or is externally imposed (moving objects like wavemakers). This method allows to easily discretize complex domains and leads to an acceptable compromise between accuracy and computational costs. This is also the method which is adopted in the simulation of fluid and rigid-body solid interaction.

- **No slip**: The wall boundary conditions are modeled by introducing additional SPH particles on the position of the boundary. The boundary particles are regularly distributed on the boundaries (to satisfy the impermeability condition) and have a zero velocity to impose the no-slip condition. To prevent the inconsistency between the density of inner particles and that of the wall, additional layers of dummy particles are placed outside the domain.

  The velocity of the dummy particle $v$ is obtained as opposite (negative) of the extrapolated velocity of interior regular particles in order to prevent penetration of regular particles to the described boundary.

  \[
  \vec{v}_b = -\frac{\sum_{a \in \Lambda_b} \frac{m_a}{\mu_a} \vec{v}_a W_{ab}}{\sum_{a \in \Lambda_b} \frac{m_a}{\mu_a} W_{ab}} \quad (2.12)
  \]

  \[
  \vec{v}_b = \frac{\sum_{a \in \Lambda_b} \frac{m_a}{\mu_a} \vec{v}_a W_{ab}}{\sum_{a \in \Lambda_b} \frac{m_a}{\mu_a} W_{ab}} \quad (2.13)
  \]

  The pressure of the dummy particles is evaluated in the similar way according
to the following relation:

\[
\bar{p}_b = \frac{\sum_{a \in \Lambda_b} \frac{m_a}{\rho_a} p_a W_{ab}}{\sum_{a \in \Lambda_b} \frac{m_a}{\rho_a} W_{ab}} \tag{2.14}
\]

\[
\bar{p}_b' = \frac{\sum_{a \in \Lambda_b} \frac{m_a}{\rho_a} W_{ab}'}{\sum_{a \in \Lambda_b} \frac{m_a}{\rho_a} W_{ab}} \tag{2.15}
\]

- **Free slip:** The velocity of the dummy particle normal to the boundary \( \mathbf{v}^n \) is obtained as opposite (negative) of the extrapolated normal velocity of interior regular particles in order to prevent penetration of regular particles to the described boundary. Tangential component of velocity \( \mathbf{v}' \) to the boundary is set equal to extrapolated velocity in order to simulate free-slip boundary condition.

\[
\mathbf{v}_b^n = -\frac{\sum_{a \in \Lambda_b} \frac{m_a}{\rho_a} \mathbf{v}_a' W_{ab}}{\sum_{a \in \Lambda_b} \frac{m_a}{\rho_a} W_{ab}} \tag{2.16}
\]

\[
\mathbf{v}_b' = \frac{\sum_{a \in \Lambda_b} \frac{m_a}{\rho_a} \mathbf{v}_a' W_{ab}}{\sum_{a \in \Lambda_b} \frac{m_a}{\rho_a} W_{ab}} \tag{2.17}
\]

The pressure of the dummy particles is evaluated in the similar way according to the following relation:

\[
\bar{p}_b = \frac{\sum_{a \in \Lambda_b} \frac{m_a}{\rho_a} p_a W_{ab}}{\sum_{a \in \Lambda_b} \frac{m_a}{\rho_a} W_{ab}} \tag{2.18}
\]

- **Moving Boundary Particles** Moving structures can be simulated by creating a collection of boundary particles that have a prescribed, non-zero, velocity. As these particles move according to some predefined equation, they will push any particles in their vicinity.

- **Repulsive Boundary Particles** In order to prevent any penetration of the boundary particles by particles, a short-range force can be added to the normal equation of motion of particles. The additional force, which applies only if the particle \( a \) is closer to a wall boundary particle \( b \) than the small distance \( r_0 \),
Chapter 2. SPH modeling of interaction between fluids and rigid-body solids

reads:

\[
F_{ab} = \begin{cases} 
0 & |x_{ab}| > r_0 \\
D \left[ \left( \frac{r_0}{|x_{ab}|} \right)^{n_1} - \left( \frac{r_0}{|x_{ab}|} \right)^{n_2} \right] & |x_{ab}| \leq r_0
\end{cases}
\]  

(2.20)

where \( n_1 \) and \( n_2 \) are model parameters typically chosen equal to 12 and 4, respectively. The parameters \( r_0 \) (dimension: length) and \( D \) (dimension: velocity squared) are case-dependent parameters; \( r_0 \) is often taken the same as \( h \) whereas \( D = agH \) in free-surface flows, where \( H \) is the initial liquid depth and \( a \) is between 1 and 10.

2.2.5 Rigid-body dynamics

In this work we use equations describing rigid body dynamics in generic 3D case by adopting the rotation matrix approach. Since this theory is explained in many textbooks on Newtonian dynamics [35], we just directly present the major equations here. In the mathematical description of rigid body motion, it is convenient to work with two different reference frames: a world frame, which remains fixed in time and an object frame, which moves and rotates with the rigid body. We denote coordinates in the world frame by \( x = (x, y, z) \), whereas the coordinates in the object frame are denoted by \( \xi = (\xi, \eta, \zeta) \). The origin of the object frame is kept equal to the centre-of-mass (COM) of the floating object. Any coordinate in the object frame, \( \xi \), can be translated into world frame coordinates \( x \) as follows,

\[
x = x_{com} + R\xi
\]  

(2.21)

where \( x_{com} \) is the displacement of COM of a rigid body and \( R \) is the so-called rotation matrix. The rotation matrix is composed of the projections of the unit vectors \( e_\xi \), \( e_\eta \), \( e_\zeta \) onto the world coordinate axes \( e_x \), \( e_y \), and \( e_z \). The motion of \( x_{com} \) is given by:

\[
\frac{dx_{com}}{dt} = v_{com}, \quad \frac{dv_{com}}{dt} = F/m
\]  

(2.22)
where $v_{com}$ is the velocity of COM, $\mathbf{F}$ is the total force acting on the floating object and $m$ is the mass of the floating object. The rotation matrix $\mathbf{R}$ evolves according to,

$$\frac{d}{dt} \mathbf{R} = \mathbf{R} \Omega$$  \hspace{1cm} (2.23)

where $\Omega$ is a matrix filled with components of the angular velocity $\omega$ which is defined in the object frame [35].

The change of angular velocity $\omega$ in time is given by,

$$\frac{d\omega}{dt} = \mathbf{I}^{-1} \mathbf{R}^{-1} (\mathbf{T} - \mathbf{R} \Omega \mathbf{I} \omega)$$  \hspace{1cm} (2.24)

where $\mathbf{I}$ is the moments-of-inertia tensor in the object frame whose entries can be calculated from the geometry in the object frame, $\mathbf{T}$ is the total external torque calculated in the world frame. In the simulation, we first compute the total force and torque on the floating object, $\mathbf{F}$ and $\mathbf{T}$, and then update the floating object position according to Eq. (2.22) – Eq. (2.24). The position of any rigid body particle is then computed from Eq. (2.21).

2.3 Test cases

In this section we show two test cases, the first one is devoted to SPH simulation of wave impact force on a fixed structure [24] while the second to floating body dynamics under the action of induced wave [27]. The accurate prediction of both cases is critical for the more complex simulation of an off-shore platform later.
Chapter 2. SPH modeling of interaction between fluids and rigid-body solids

2.3.1 Three dimensional dam break flow against a tall structure

In this section we present the validation of the SPH method for a typical three-dimensional dam break problem. Our purpose is to simulate a dam break experiment, which was carried out by Yeh & Petroff [24]. The experiment provides data for the force of an impacting fluid surge, which can be compared with the force predicted by the SPH simulation. The experiments utilized a rectangular tank, 0.6 m wide, 1.6 m long, and 0.75 m high. A centrally located vertical column with a square 0.12 m profile is located 0.9m from one end of the tank. A sketch of the experimental geometry is depicted in Fig. 2.2, the fluid used in experiment is water at ambient conditions. Water is held at a height of 0.3 m behind a gate which is raised to begin the experiment. Since it was not possible to completely evacuate the water from the tank, a 1 cm layer of water was present on the tank floor prior to the experiment.

As the fluid surge impacts upon the vertical column, attached load gauges report the impact force experienced by the column. Using an SPH model of this experiment, we compare the horizontal component of impact force predicted by the SPH method to that measured in the experiment.

A sequence of snapshots of the particle positions is shown in Fig. 2.3, where particles are colored by velocity magnitude. The simulation shown is carried out with an initial particle spacing of 0.5 mm, which corresponds to a total number of approximately 800,000 fluid particles and 1,000,000 boundary particles. The sequence is as follows, first the fluid particles fall under the effect of gravity and flow with relatively high velocity towards the vertical column. Then, at approximately 0.3 seconds, they reach the vertical column. Some fluid particles impact upon the column before falling back. Whereas the particles to the side of the tank will move past the column. After 1 second of simulation, a secondary impact upon the column occurs as the initial wave is reflected by the tank wall to the rear of the column.

The experimentally reported force data, which is shown in Fig. 2.4, reflect what
Figure 2.2: Schematic diagram of the dam geometry (all dimensions in m)
Figure 2.3: Evolution of the water collapse and interaction with the column at various time steps
can be observed visually in Fig. 2.3. In the first 0.3 seconds, the force is negligible because there is hardly any fluid touching the column. Then the first water surge arrives at the column, at 0.3 s, part of the fluid is decelerated and transfers its force onto the column. When the reflected wave passes the column, at approximately 1.3 seconds, a negative horizontal force is measured on the column. Finally the net horizontal force on the column settles around 0 N when the fluid is slowly coming to a standstill.

The horizontal force on the vertical column is plotted in Fig. 2.4 where a good agreement is obtained. Until 0.3 seconds, the force is practically 0 N, as expected. The height of the first peak is predicted rather well by the SPH simulation. This is in contrast with some other SPH simulation results reported in the literature [25], for example, had to use different filtering techniques to show agreement between their SPH results and the experimental data. Our SPH results, on the other hand, are generally quite close to the experimental data without any post-processing.

At 1.5 seconds, the secondary impact occurs roughly 0.25 seconds early in the SPH simulation. This is observed in other literature as well [24–26]. Authors in [24], who compared the experimental data with their own finite-volume simulation technique, point out that small bubbles are created in the experiment (presumably in regions of low pressure in the fluid), which in turn may slow down the speed of the reflected wave. Since these bubbles are not modeled in their numerical simulation method, the deceleration of the reflecting wave is not observed. Since our SPH simulation technique does not incorporate a model for bubble creation at low pressures, it does not pick up the same phenomenon, and the explanation in [24] is expected to be relevant for our simulation technique as well. In spite of this, our SPH result still shows better agreement with experimental data than some other results in the literature [26], for example, overpredicted the force on the obstacle after the initial wave impact.
2.3.2 Floating body dynamics under the action of induced wave

To fully capture the motion of offshore platforms we must ensure that we not only correctly capture the hydrodynamic loads on the structure, but that the motion of this structure is also correctly resolved and accounted for in the flow solution. As a validation case, we consider the experimental setup reported by Ruol *et. al* [27] and Manenti *et. al* [1]. In the work of Ruol *et. al* they carried out an experiment of a buoyant cube (representative of a waterbreak) secured by pile-mooring. Under the action of a periodic wave motion, both the vertical position of the cube, and upstream fluid surface height were recorded.

This experiment was designed to be Quasi-2D because the width of the waterbreak is approximately equivalent to the width of the tank. Manenti *et. al* also simulated this experiment in a 2D SPH model showing reasonable agreement. However, we note that in the experiment the waterbreak was allowed to roll to some extent, where this degree of freedom was no considered by Manenti *et. al*. We choose to include this
extra degree of freedom in this work by applying the rigid-body-motion equations with the rotation matrix approach described in Eq. (2.21) – Eq. (2.24).

Figure 2.5: Sketch of the plan view of the central part of the wave flume with the floating breakwater set up.

To illustrate the situation, the experimental configuration is shown in Fig. 2.5 and Fig. 2.6. The piston wave maker is located to the left of the domain, where it generates regular surface waves. The floating object is initially located 2 m downstream of the wave maker. The fluid used in the experiment is water with a density of 1000 kg/m³. The material and internal geometry of the floating object were chosen such that the effective density is 500 kg/m³, so that it is only half submerged while the system is at rest. Be advised that in the experiment, the floating object was moored with piles
such that it can not move horizontally. This experimental condition is reproduced in the SPH simulation by fixing the horizontal position of the center of mass of the floating object while keeping all other degree of freedom (DOF) unrestricted, such as vertical displacement and rotation. The flat piston wave maker describes a sinusoidal motion in the horizontal direction with a frequency of 1.14 Hz and stroke of 0.05 m. Fig. 2.7 shows a side-view frame of the simulation at $t = 5$ s.

Figure 2.7: Side view of SPH simulation of the wave flume test with a pile-moored floating breakwater at $t = 5$ s.

For quantitative comparison between the SPH model and laboratory results, the water surface elevation at gauge number 3 (Fig. 2.5) and heave displacements of the floating object are considered at this stage of the analysis. Fig. 2.8(a) shows SPH calculated oscillations of the free surface at wave gauge number 3. The compari-
son with the experimental measures at steady condition shows good agreement of the signals frequency once the model achieves a steady state of periodic oscillation. Fig. 2.8(b) compares numerically calculated and measured heave oscillations of the floating breakwater. Phase angle and frequency are predicted with good accuracy as the model approaches a steady oscillation.

![Graph](image)

(a) Upstream elevation at gauge #3

(b) Heave of the floating breakwater

Figure 2.8: Comparison between experimental data and numerical results for the floating breakwater

We believe the better agreement observed in our work is due to that, firstly, we include the modeling of the pile-mooring which prevents the floating from moving
horizontally, and secondly, the buoyant cube is allowed to roll slightly about the horizontal axis as in the original experiment. If the rotational degree of freedom is neglected it is not possible to fully capture the behavior of the waterbreak driven by the induced wave, resulting in a discrepancy between the model and experimental results.

2.4 Application: Wave impact on a off-shore oil platform

As a proof-of-concept, we will show a realistic application of the floating object model in this section. The objective is to study the effect of a solitary extreme wave impacting upon a tension-leg platform (TLP).

The geometrical configuration of the present simulation is shown in Fig. 2.9. The fluid basin is 50 m deep and stretches over a longitudinal distance of 800 m, and a transverse width of 150 m. The platform's center-of-mass is situated at a distance of 426 m from the initial position of the moving boundary particles used to generate the solitary wave, i.e. practically in the middle of the domain. The beach downstream of the platform, which is intended to prevent reflection of waves, starts at 600 m from the initial position of the moving boundary particles and has an angle of 14° with the horizontal. In the transverse direction, the fluid is contained by rigid side walls.

A close-up of the SPH model of a floating platform is shown in Fig. 2.10. It consists of four vertical cylinders that, in reality, are mostly filled with air and keep the platform afloat. The cylinders are connected by pontoons. The top deck is modeled by a thin layer of particles. In principle one could add equipment at the top of the platform, but it is left out here for the sake of simplicity. Nevertheless, all the critical parameters for the motion of the floating platform, namely the center-of-mass, the mass and the moments-of-inertia, are taken from a real platform. Therefore, the actual motion of the platform is still expected to be predicted fairly accurately, despite
Chapter 2. *SPH modeling of interaction between fluids and rigid-body solids*  

Figure 2.9: Geometrical setup of three-dimensional simulation of an extreme wave hitting a platform.

Despite the lack of geometrical detail.

The platform is connected to the seabed by eight tension legs with two on each vertical cylinder of the platform. In the SPH simulation, tension legs are modeled as springs. Each of the springs is connected to an anchor point at the seabed at 500 m depth; although the sea bed is well outside the flow domain, this does not affect the
validity of the spring equations. The points where the tension legs are attached to the platform are allowed to move in all three directions.

Each tension leg acts like a spring, with a spring constant, \( k_0 \), and a nominal length of zero tension, \( L_0 \). Each tension leg \( i \) provides the following force:

\[
F_{\text{TensionLeg},i} = -k_{0,i} \left( |x_{\text{attach},i} - x_{\text{anchor},i}| - L_{0,i} \right) \frac{r_i}{|r_i|} \tag{2.25}
\]

where \( x_{\text{attach},i} \) denotes the position of the attachment point of the tension leg at the platform and \( x_{\text{anchor},i} \) denotes the location of the anchor at the sea bed. The force is precisely directed along the vector separating the attachment point and the anchor point with

\[
r_i = x_{\text{attach},i} - x_{\text{anchor},i} \tag{2.26}
\]

The anchor points at the sea bed are aligned with the attachment points at the platform, so that the tension legs are perfectly vertical in the equilibrium situation. With the geometry chosen for the platform, the tension legs are slightly stretched in the equilibrium situation.

Because of the large spring constant, \( k_0 \), a TLP is designed to have small vertical displacement under the wave impact as vertical displacement can be hazardous. Given this, it is expected that the horizontal displacement of the platform is relatively larger than its vertical displacement. Therefore, it's useful to investigate the effect of tension legs when the platform is displaced horizontally. We first analyzed this problem by assuming that the platform has moved by a distance \( \Delta x \), so that \( r_i \) is equal to:

\[
r_i = z_0 e_z + \Delta x e_x \tag{2.27}
\]

Inserting Eq. (2.27) into Eq. (2.25) gives:

\[
F_{T,i} = \frac{r_i}{|r_i|} - k_{0,i} \left( |r_i| - L_{0,i} \right) \frac{r_i}{|r_i|} = -k_{0,i} \left( |z_0 e_z + \Delta x e_x| - L_{0,i} \right) \frac{z_0 e_z + \Delta x e_x}{|z_0 e_z + \Delta x e_x|} \tag{2.28}
\]
Chapter 2. *SPH modeling of interaction between fluids and rigid-body solids*

Fig. 2.11 shows the force components due to the eight tension legs in the platform geometry used in the SPH simulation.

To highlight this effect in some more detail, we expand Eq. (2.28) into a Taylor series up to third order. We investigate what happens if there is a horizontal displacement $\Delta x$. We can calculate the length of $r_i$ as:

$$|r_i| = \sqrt{z_0^2 + \Delta x^2} = z_0 \left(1 + \frac{1}{2} \epsilon^2 + \ldots\right)$$

(2.29)

where $\epsilon$ is introduced as: $\epsilon = \Delta x / z_0$. With this, the horizontal forces as given by Eq. (2.28) becomes:

$$F_{T,i} \cdot e_x = -k_{0,i} \left(z_0 + \frac{1}{2} z_0 \epsilon^2 - L_{0,i}\right) \left(\frac{\epsilon}{1 + \frac{1}{2} \epsilon^2}\right)$$

(2.30)

and the vertical force is:

$$F_{T,i} \cdot e_z = -k_{0,i} \left(z_0 + \frac{1}{2} z_0 \epsilon^2 - L_{0,i}\right) \left(\frac{1}{1 + \frac{1}{2} \epsilon^2}\right)$$

(2.31)

The estimates above are plotted in Fig. 2.11 which shows good agreement with the exact solution obtained from Eq. (2.28) given small horizontal displacement. Thus, in our simulation, the horizontal force is approximately linear in $\epsilon$, with an important third-order term, whereas the magnitude of the vertical force is non-zero even if $\epsilon = 0$, and increases with a quadratic term in $\epsilon$. Hence, any horizontal displacement is expected to cause a higher resulting force from the tension legs in the vertical direction than in the horizontal direction.

In summary, if the platform moves in the horizontal direction, the tension legs are stretched. In the horizontal direction, they tend to pull the platform back into position, but also they pull the platform down with even larger vertical force downward than its horizontal force component. This is a potential risk to a tension leg platform as it may end up being submerged by a forceful high sea wave. In our SPH simulations, we will investigate whether this effect is compensated by the buoyant force exerted by the fluid.
Figure 2.11: Total force from eight tension legs, as a function of the horizontal displacement of the platform, for the realistic platform geometry.

We prescribe the displacement of the wave maker as a piston-like motion which follows a hyperbolic tangent function as:

$$\frac{x_{\text{piston}}(t) - x_{\text{piston}}(0)}{S} = \frac{1}{2} \left\{ 1 + \tanh \left( 7.6 \left( \frac{t}{\tau} - \frac{1}{2} \right) \right) \right\}$$  \hspace{1cm} (2.32)

where $S = 80m$ is piston stroke. Goring [36] derives that, for a perfect solitary wave, the time scale $\tau(s)$ should be:

$$\tau = \frac{(3.8 + \frac{3}{16}(S/d)^2)}{\sqrt{(1 + \frac{3}{16}(S/d)^2) \left( \frac{3}{64}(S/d)^2 \right)}} \sqrt{d/g}$$ \hspace{1cm} (2.33)

where $d = 50m$ is the water depth and $g$ is gravitational acceleration. The resulting free-surface wave in the fluid is then a solitary wave of elevation according to reference [36–38].

Snapshots of the particle positions during the SPH simulation are shown in Fig. 2.12, at six instants of time. To illustrate better the flow about the platform, these results are presented in a zoomed view in Fig. 2.13. The fluid is colored by the local velocity magnitude. At the start of the simulation, $t = 0$ s, the fluid is at rest, but a solitary
wave is formed within the first 20 seconds by moving the left boundary. The solitary waves travels with a velocity of approximately 25 m/s and reaches the platform around $t = 30$ s. Between $30 \text{s} < t < 40 \text{s}$, some water is flowing on the platform deck, while the platform is displaced towards the right. At $t = 40 \text{s}$, the solitary wave reaches the beach and dissipates, the fluid eventually comes to rest again, with the platform slowly moving back to its equilibrium position.

Figure 2.12: Six snapshots of the SPH simulation with a moving platform: 3D view. The fluid is colored by velocity magnitude. The solid side walls are not shown for better visualization.

The displacement of the platform in the course of time is captured in Fig. 2.14. On the left-hand-side, we plot the three-dimensional motion as a function of time. It is clear that the highest displacement takes place in the x-direction (“Surge”) between
Figure 2.13: Six snapshots of the SPH simulation with a moving platform: Zoomed view of platform.
30 and 40 seconds of simulation time. This coincides with the moment the platform is hit by the solitary wave. After the wave has passed, the platform moves back towards its initial position but it overshoots a bit, so that it is located some 50 m left of its original position after 75 s. The overshooting is a characteristic of an under-damped mass-spring-damper system: the platform represents the mass here, the tension legs the springs and the fluid the damping mechanism of the motion.

The vertical displacement ("Heave") increases between 20 and 25 seconds, as the platform is lifted up by the arriving wave. Quickly afterwards, however, the vertical displacement becomes negative, indicating that the platform is pulled into the water. This correlates with the horizontal displacement which causes the tension legs to pull the platform into the water, as explained in Eq. (2.31).

The trajectory of the platform, plotted in the right graph of Fig. 2.14, confirms the relation between displacement and vertical displacement. The trajectory is similar to a parabola, with the vertical displacement approximately proportional to the horizontal displacement squared. This is in agreement with the analysis described above which is indicated by Fig. 2.11.

Since the total force on the platform is equal to the sum of the force by the tension legs plus the force by the fluid plus gravity, it is now also possible to extract information on the force exerted by the fluid on the platform; it is shown in the left graph of Fig. 2.15. The right graph of Fig. 2.15 shows the force from the tension legs. The result suggests that the fluid exerts a much larger force on the platform in the upward direction (buoyant force) than in the horizontal direction, even when the wave hits the platform. The main contribution of the fluid is thus to keep the platform afloat. Comparison between the left graph and right graph of Fig. 2.15 shows that the fluid provides upward lift when the tension legs pull the platform into the water.

Finally, the rotation of the platform during the simulation is shown in Fig. 2.16. The pitch (rotation around the y-axis) is highest precisely when the wave arrives around $t = 25$ s, as could be expected. The actual pitch rotation, however, is ex-
Figure 2.14: Results from the SPH simulation with a moving platform. (a) displacement of the platform’s center-of-mass in x-direction (Surge), y-direction (Sway) and z-direction (Heave). (b) Position of the platforms center-of-mass in the xz-plane during the simulation.

extremely small, it has a maximum of 0.6°. The reason for this extremely low value is because a stretched tension leg exerts a larger force than a tension leg in its equilibrium position, any rotation in the x-direction (roll) or y-direction (pitch) is directly counteracted. Therefore, it is no coincidence that the largest rotation angles are observed in the z-direction (yaw), it is the only direction where the rotation is not efficiently counteracted by the spring-like tension legs.

This model shows that the result of rogue wave impact may be counter intuitive. Though the platform rose upon initial impact, it quickly became submerged. It also shows that a tension-leg platform is very unlikely to topple over during the arrival of an extreme wave, at least for the design parameters used during the present simulation. A model such as this can serve as a stepping stone for more advanced simulations in the future, where the platform can be modeled in more detail and the sea can be modeled in better agreement with real oceanographic data.
Figure 2.15: Results from the SPH simulation with a moving platform. Left: Forces exerted by the fluid on the platform in x-direction, y-direction and z-direction. Right: Forces exerted by the tension legs on the platform in x-direction, y-direction and z-direction.

Figure 2.16: Results from the SPH simulation with a moving platform: Platform rotation as a function of time.
2.5 Conclusions

Two validation cases and a realistic design experiment were performed using the SPH method. Validation of dam-break flow with a rectangular column located downstream showed that results were in good agreement with the experimental data. Oscillations in the column forces after both the first and the secondary impact were well captured. The simulations demonstrated the ability of SPH to reproduce the complex transient loading characteristic of inertial and gravity driven flow on fixed structures. The SPH simulation was also validated against an experiment which measures the vertical displacement of a rectangular object forced by regular waves. It showed that the heave of the floating object and the wave elevation was in good agreement with experimental data. Finally, the SPH simulation was applied on a real world application to study the effect of a single solitary wave hitting a tension-leg platform. The impact force due to the wave and the reaction force exerted by the tension legs were analyzed separately in detail. Also studied, was the horizontal and vertical displacement of the platform under the effect of a sea wave and tension legs. This analysis showed that the platform may be pulled into the water by stretched tension legs, where the extension of the tension legs also governed the rotational behavior of the platform. This simulation can serve as a blueprint for more advanced simulations in the future where the platform model may include more detail, and the initial state of the fluid phase may be more representative of conditions observed in oceans.
Chapter 3

SPH modeling of extreme waves

3.1 Introduction

In offshore hydrocarbon production, a platform from which drilling and production activities occur is placed in one location for many years, typically decades. These structures must survive all weather types, including extreme waves. Extreme waves are defined as waves that have two times higher amplitude than surrounding waves. Although they are rare, extreme waves have been observed on several locations in recent years [39]. The first extreme wave to be detected by a measuring instrument was the “New Year wave”, occurring at the Draupner platform in the North Sea off the coast of Norway on 1 January 1995. In an area with significant wave height of approximately 12 m, an extreme wave with a maximum wave height of 25.6 m occurred. The peak elevation above the nominal sea level was 18.5 m. In recent work, the European space agency also identified more than 10 rogue waves, over 25 m high, in a period of just three weeks [40]. All these observations demonstrate that such waves are a real danger to shipping and other offshore industries. In particular, the highly non-linear impact of a rogue wave on a floating, moored offshore structure is a problem that has significant practical application in the safety of offshore oil and gas production.
Thus an accurate knowledge of response to rogue waves is therefore a fundamental component of offshore structure design. Lab tests with scale models are frequently used to determine hydrodynamic forces, but they are both expensive and time consuming. As such, the behavior of offshore structures subject to extreme waves cannot be studied by small-scale model experiments easily. So computational tools for the prediction of local wave impact loads and overall motion of the structure are desirable.

Smoothed Particle Hydrodynamics (SPH) is a mesh-free, fully Lagrangian method for numerical simulation of fluid flow [12-15]. In the SPH method, a computational domain is discretized by a set of points, or particles, and a meshless discretization scheme is used to represent a scalar or vector field in terms of its values at these points. Since the particles are advected with the flow field, the SPH is ideally suited to problems of free surface flow, where computationally expensive surface tracking, typically required by continuum approaches, is not necessary [12]. Although several studies have been conducted on studying the fluid-structure interactions of offshore ships and platforms [41-43], all studies do not capture three-dimensional characteristics of real sea waves consisting of multiple wave modes of different frequencies and different directions.

In this work, a new numerical model is devised to directly generate realistic three-dimensional sea waves in SPH. The underlying theory of the method follows what describes in Schäffer [44]. The waves are generated by moving the side boundaries of the fluid domain according to the sum of random Fourier modes, each with its own direction, amplitude and wave frequency. If the amplitudes and the frequencies of each single wave mode is chosen according to a sea wave spectrum, the same sea wave state can be generated as the ensemble of all single wave modes. This method can be used to generate any random sea wave state as long as the wave spectrum is available.
3.2 Methodology

3.2.1 Generation of a single linear wave mode with a piston wave maker in 2D

Havelock [45] first derived the potential flow solution of progressive free-surface waves generated by a wave maker. Following Benz & Asphaug [46], a mathematical model, based on the linearized governing equations, is used for the particular problem of the waves generated by a sinusoidally moving piston-type wave maker starting from rest at some arbitrary point \( x^* \) with frequency \( \omega_0 \), velocity amplitude \( U = \frac{1}{2} S \omega_0 \) where \( S \) is the stroke of the piston. Following the same approach as Havelock [45] we arrive at the following fluid potential:

\[
\Phi(x, z, t) = -B_0 k_0^{-1} \cosh(k_0 [d + z]) \sin(k_0 [x - x^*] - \omega_0 t + \varphi^*) \\
+ \cos(\omega_0 t + \varphi^*) \sum_{n=1}^{\infty} B_n k_n^{-1} e^{-k_n [x - x^*]} \cos(k_n [d + z])
\]

(3.1)

where the constants \( B_n \) are given as

\[
B_0 = \frac{2 \omega_0 S \sinh(2k_0 d)}{2k_0 d + \sinh(2k_0 d)} \\
B_n = \frac{2 \omega_0 \sin(2k_n d)}{2k_n d + \sin(2k_n d)} \quad \forall n \geq 1.
\]

(3.2)

In these equations, \( k_0 \) is the real positive root of \( \omega_0^2 = g k_0 \tanh(k_0 d) \) and \( k_n(n \geq 1) \) are real positive roots of \( \omega_0^2 = -g k_n \tanh(k_n d) \). \( \varphi^* \) is an arbitrary phase lag. The corresponding wave elevation profile downstream of the wave maker is then

\[
\zeta(x, t) = \frac{B_0 \omega_0}{g k_0} \cosh(k_0 d) \cos(k_0 [x - x^*] - \omega_0 t + \varphi^*) \\
- \sin(\omega_0 t + \varphi^*) \sum_{n=1}^{\infty} \frac{B_n \omega_0}{g k_n} e^{-k_n [x - x^*]} \cos(k_n d)
\]

(3.3)

Benz & Asphaug [46] notes that \( k_n d \geq \frac{1}{2} \) for all \( n \geq 1 \), so that the terms under the summation of Eq. (3.3) decrease rapidly with \( x \). At a distance of \( 3d \) from the
wave maker, all these terms are negligible. Thus, the wave profile far downstream of the wave maker is [44]:

\[
\zeta(x \geq 3d, t) = S \left[ \frac{\cosh(2k_0d) - 1}{2k_0d + \sinh(2k_0d)} \right] \cos(k_0[x - x^*] - \omega_0t + \varphi^*)
\]  

(3.4)

where we have simplified the first term, using \( \omega_0^2 = gk_0 \tanh(k_0d) \) and \( 2\sinh^2(q) = \cosh(2q) - 1 \). If we denote wave height H (crest to trough) as two times the wave amplitude, then the theoretical ratio between the wave height H and the wave maker stroke S, say \( \chi = H/S \), is:

\[
\chi_{\text{theory}} = \frac{2[\cosh(2k_0d) - 1]}{2k_0d + \sinh(2k_0d)}.
\]  

(3.5)

Ursell et. al [47] found in the experiment that the observed wave height was slightly lower than the theoretical prediction of potential flow theory due to the presence of viscosity. The experimental data suggests that the actual value \( \chi \), i.e. the ratio between the wave height and piston stroke, can be approximated by:

\[
\chi_{\text{experiment}} \approx \tanh \left( \frac{k_0d}{2} \right).
\]  

(3.6)

### 3.2.2 Generation of a single linear wave mode in any direction with a curved wave maker in 3D

Until now we focused on waves in the x-direction with a plane wave maker in 2D. We now generalize this to waves in any direction in the \((x, y)\) plane with a curved wave maker in 3D. A schematic diagram of wave propagating in any direction is shown in Fig. 3.1.

Each boundary element at \( x^*(y') \) moves harmonically in the \( k_0 \) direction where \( k_0 \) is the wave number vector. By setting the phase \( \varphi^* = k_0 \cdot x^*(y') \), the wave maker velocity, which is in the direction of the wave number \( k_0 \), gives the boundary condition at each element:

\[
u_{MB}(t)_{x=x^*(y')} = \frac{1}{2} S \omega_0 \cos(\omega_0t + k_0 \cdot x^*(y')).
\]  

(3.7)
Chapter 3. SPH modeling of extreme waves

Figure 3.1: A schematic diagram of progressive waves along any direction generated by a curved wave maker (red line, denoted by $x^*(y')$).
From a simple analogy of Eq. (3.3), the corresponding wave elevation, far downstream of the wave maker, is:

$$
\zeta(x, t) = S \left[ \frac{\cosh(2k_0d) - 1}{2k_0d + \sinh(2k_0d)} \right] \cos(k_0 \cdot x - \omega_0 t), \quad \text{if} \quad k_0 \cdot [x - x^*(y')] > 3k_0d. \quad (3.8)
$$

Following Eq. (3.7), the equation of motion of an element in the moving boundary is given by:

$$
\frac{\partial x_{MB}}{\partial t} = u_{MB}; \quad u_{MB}(t) = \frac{S\omega_0k_0}{2k_0} \cos(\omega_0 t + k_0 \cdot x_{MB,0}(y') + \varphi_0), \quad (3.9)
$$

where $x_{MB,0}$ denotes the initial position of the boundary at coordinate $y'$.

### 3.2.3 Generation of multiple wave modes

According to linear wave theory, any arbitrary wave can be represented as an assemble of $N_{\text{modes}}$ different wave modes with selected wave amplitude, frequency, direction and phase:

$$
\zeta(x, t) = \sum_{n=1}^{N_{\text{modes}}} A_i \cos(k_i \cdot x - \omega_i t + \varphi_i), \quad (3.10)
$$

where $A_i$ denotes the amplitude of the $i$-th wave mode, $\omega_i$ is the frequency of the $i$-th wave mode, and $\varphi_i$ is the phase of the $i$-th wave mode. For each mode, the dominant wave number $k_i$ is related to the wave frequency by:

$$
\omega_i^2 = gk_i \tanh(k_id), \quad (3.11)
$$

where $k_i = |k_i|$. To leading order, the potential describing the flow is:

$$
\Phi(x, t) = -\sum_{n=1}^{N_{\text{modes}}} A_i \omega_i k_i^{-1} \frac{\cosh(k_i[d + z])}{\sinh(k_id)} \sin(k_i \cdot x - \omega_i t + \varphi_i). \quad (3.12)
$$

The potential flow derived above can be generated by moving the wave maker according to the sum of the wave maker motion used to generate each single wave as

$$
u_{MB} = \sum_{n=1}^{N_{\text{modes}}} A_i \omega_i k_i \cos(k_i \cdot x_{MB,0} - \omega_i t + \varphi_i); \quad \frac{dx_{MB}}{dt} = u_{MB}. \quad (3.13)$$
which is a generalization of Eq. (3.9) to the case of $N_{\text{modes}}$ different wave modes. Note that $x_{\text{MB},0}$ in Eq. (3.13) denotes the initial position of the boundary particle: $x_{\text{MB},0} = x_{\text{MB}}(t = 0)$.

The simple summation of wave modes, as described in Eq. (3.12), does not incorporate any effect that one mode may have on the motion of the others. Higher-order methods exist where the mutual interaction is taken into account, within the framework of potential flow theory [48]. For the sake of simplicity, we neglect those interactions here. They could be included in a future refinement of the method; there is no conceptual limitation that forces us to look at linear wave interactions here.

3.2.4 Wave spectra for realistic sea states

Parameters in wave elevation profile Eq. (3.10) and equation-of-motion of wave maker Eq. (3.13) have to be carefully chosen in order to reproduce a realistic sea state. In particular, the following parameters need to be chosen for each wave mode: amplitude $A$, frequency $\omega$, wave number vector $k$, and phase lag $\varphi$.

A realistic sea state consists of a large number of wave modes, with $N_{\text{modes}} > 100$ being common [39]. The statistical distribution of wave amplitude $A$ is usually described in terms of a wave frequency through a wave spectrum. The wave spectrum data obtained through the onsite measurement is discrete, therefore a mathematical model has to be used to fit the measured data before using. Typical mathematical models of wave spectrum are the JONSWAP [49] or Pierson-Moskowitz spectrum [50]. For example, the Pierson-Moskowitz spectrum has the following functional form:

$$S(\omega) = \frac{S_{\text{peak}}}{\exp(-5/4)} \left(\frac{\omega_{\text{peak}}}{\omega}\right)^5 \exp\left[-\frac{5}{4} \left(\frac{\omega}{\omega_{\text{peak}}}\right)^4\right]$$

(3.14)

where $S_{\text{peak}}$ denotes the maximum value of the wave spectrum and $\omega_{\text{peak}}$ the wave frequency at which the spectrum is maximum. The spectrum function $S(\omega)$ describes the power spectrum of the free-surface elevation. The significant wave height (SWH or $H_s$) is defined as four times the square root of the zeroth-order moment of the wave.
Chapter 3. SPH modeling of extreme waves

A realistic sea state consists of single waves with multiple directions. The wave direction is determined by the wave number \( \mathbf{k} \). The magnitude of the wave number vector, \( \mathbf{k}_i \), is computed from the dispersion relation, Eq. (3.11) depending on wave frequency \( \omega_i \). In order to describe the random directionality, the direction \( \theta_i \) of each wave vector \( \mathbf{k}_i \) can be obtained by sampling \( \theta_i \) from a normal distribution with mean \( \langle \theta_i \rangle = \frac{\pi}{2} \) and variance \( \sigma^2 = 0.25 \).

A realistic sea state consists of single waves with multiple directions. The wave direction is determined by the wave number \( \mathbf{k} \). The magnitude of the wave number vector, \( \mathbf{k}_i \), is computed from the dispersion relation, Eq. (3.11) depending on wave frequency \( \omega_i \). In order to describe the random directionality, the direction \( \theta_i \) of each wave vector \( \mathbf{k}_i \) can be obtained by sampling \( \theta_i \) from a normal distribution with mean \( \langle \theta_i \rangle = \frac{\pi}{2} \) and variance \( \sigma^2 = 0.25 \).
Chapter 3. *SPH modeling of extreme waves*  

$\theta_0$ and variance $\sigma_\theta$. Thus, if the waves are spread out over a range of directions, the wave spectrum, for instance the Pierson-Moskowitz spectrum, has the form of a joint probability distribution in the frequency and the direction:

$$S(\omega, \theta) = \frac{S_{peak}}{\sigma_\theta \sqrt{2\pi}} \left( \frac{\omega_{peak}}{\omega} \right)^5 \exp \left[ -\frac{5}{4} \left( \frac{\omega}{\omega_{peak}} \right)^4 - \frac{(\theta - \theta_0)^2}{2\sigma_\theta^2} \right]$$  \hspace{1cm} (3.20)

### 3.2.5 Extreme wave focusing

Many of the independent wave modes peak at one point in space at the same time resulting in an extreme wave event. In a simulation or an experiment, one could in principle wait for long time to let all important wave modes focus at one point to generate the extreme wave. However, on one hand this may take a lot of time, and on the other hand the extreme wave event does not necessarily occur at a location of interest, which is usually the position where the measurement equipment is located in an experiment.

An extreme wave event can be generated more efficiently by judiciously choosing the phase of each wave mode $[51]$. As a first step, we can rewrite Eq. (3.10) for the elevation profile as:

$$\zeta(x, t) = \sum_{n=1}^{N_{\text{modes}}} A_n \cos[k_i \cdot (x - x_f) - \omega_i (t - t_f) + \tilde{\varphi}_i],$$  \hspace{1cm} (3.21)

where $\tilde{\varphi}_i$ is a modified phase, and $x_f$ and $t_f$ are the spatial coordinate and the time instant of the extreme wave event, respectively. Note that both $\omega_i t_f$ and $k_i \cdot x_f$ only provide a constant phase shift in the cosine.

In order to generate an extreme wave, it is necessary to choose $\tilde{\varphi}_i$ within a restricted range close to 0 for all wave modes so that at the place and time of the extreme wave event, $x_f$ and $t_f$, all wave modes give a positive contribution to the total elevation profile in Eq. (3.10). Thus, a high free-surface elevation can be expected at the focus point $x_f$ at time $t_f$. In the present study, we choose each $\tilde{\varphi}_i$ as a random
number, drawn from a uniform probability distribution between $-\pi/2$ and $\pi/2$:

$$\frac{-\pi}{2} \leq \varphi_i \leq \frac{\pi}{2}, \quad \forall i.$$  \hfill (3.22)

Substituting \( \varphi_i = -k_i \cdot x_f + \omega_i t_f + \tilde{\varphi}_i \) into Eq. (3.12), we obtain the potential as:

$$\Phi(x, t) = - \sum_{n=1}^{N_{\text{modes}}} A_i \omega_i k_i^{-1} \frac{\cosh(k_i[d + z])}{\sinh(k_i d)} \sin(k_i \cdot (x - x_f) - \omega_i (t - t_f) + \tilde{\varphi}_i).$$  \hfill (3.23)

The velocity of the boundary to get the desired profile is obtained from Eq.(3.13), again by substituting \( \varphi_i = -k_i \cdot x_f + \omega_i t_f + \tilde{\varphi}_i \):

$$\mathbf{u}_{\text{MB}} = \sum_{n=1}^{N_{\text{modes}}} \frac{A_i \omega_i k_i}{\chi_i k_i} \cos[k_i \cdot (x_{\text{MB},0} - x_f) - \omega_i (t - t_f) + \tilde{\varphi}_i], \quad \frac{dX_{\text{MB}}}{dt} = \mathbf{u}_{\text{MB}}.$$  \hfill (3.24)

### 3.2.6 Implementation in SPH formalism

As derived in previous sections, a multi-directional multi-modal wave spectrum can be generated by moving the wave maker particle (moving boundary) according to Eq.(3.24). The moving boundary area usually starts from one side of the simulation domain and ends at the focusing point. However, this can generate discontinuity between “moving boundary” particles and “fixed boundary” particles, leading to fluid leaking. In order to maintain the continuity in the transition region, a spatial smoothing can be applied between “moving boundary” particles and “fixed boundary” particles. The spatial smoothing factor has the following form:

$$f_x = \frac{1}{2} \left\{ 1 + \tanh \left[ 7.6 \left( \frac{x_{\text{limit}} - x_{\text{MB},0}}{L_x} - \frac{1}{2} \right) \right] \right\},$$  \hfill (3.25)

where \( L_x \) is a smoothing length and \( x_{\text{limit}} \) denotes the separation between “moving boundary” particles and “fixed boundary” particles.

In addition to the spatial smoothing, a smoothing in the time domain is also necessary. This is because fluid particles need time to settle down, therefore the wave maker cannot start moving according to Eq. (3.24) exactly at time \( t = 0 \) s. A smoothing in time domain is applied to gradually initiate the motion of the wave
maker which will eventually approach to the final equation of motion Eq. (3.24) at large time \( t \). The smoothing factor in time domain has the following form:

\[
f_t = \frac{1}{2} \left\{ 1 + \tanh \left[ 7.6 \left( \frac{t}{t^*} - \frac{1}{2} \right) \right] \right\},
\]

where \( t^* \) is a smoothing time scale. In the present simulations we choose \( t^* \) equal to 30 s.

As a summary, the final equation of motion of moving particles now becomes:

\[
\mathbf{u}_{MB} = \sum_{n=1}^{N_{\text{modes}}} \frac{A_i \omega_i k_i}{\chi_i k_i} \cos[k_i \cdot (\mathbf{x}_{MB,0} - \mathbf{x}_f) - \omega_i (t-t_f) + \bar{\varphi}_i], \quad \frac{d \mathbf{x}_{MB}}{dt} = \mathbf{u}_{MB}. \tag{3.27}
\]

In the simulations presented in this report, the phase \( \bar{\varphi}_i \) follows a uniform distribution over a range described in Eq. (3.22). The wave direction \( \theta_i \) follows a normal distribution with mean \( \theta_0 \) and variance \( \sigma_\theta \) depending on each specific application.

### 3.3 Numerical Examples of extreme wave modeling

The extreme wave event occurred on Jan, 1st, 1995 at the Draupner platform in the North Sea, which was named as ‘New Year Wave’, was used to validate the proposed numerical scheme. The wave elevation measured at the platform is shown as the solid black curve in Fig. 3.2. The wave crest and trough is is 20m and -5m respectively which leads to a 25m-high wave amplitude while the surrounding waves are only 5m high, indicating that this is really an extreme wave event.

The simulation setup is shown in Fig. 3.3. The numerical tank has a length of 1000m, width of 150m, depth of 50m and an inclined beach with length of 400m. The length of the transition zone, \( L_x \), is set to 100 m. The ‘New Year Wave’ spectrum is shown as the black curve in Fig. 3.4. A Pierson-Moskowitz spectrum [50] is used to fit the measured spectrum and is plotted as the red curve in Fig. 3.4. The wave spectrum is discretized into 100 independent modes with cutting-off frequencies ranging from
Figure 3.2: ‘New Year Wave’ elevation profile. Black solid line is measured data. Red solid line is SPH simulation result.
0.2 rad/s to 2 rad/s. The direction of each wave mode $\theta_i$ is taken from a normal distribution with mean $\theta_0 = 0$ and variance $\sigma_\theta = 15$ degree. Each $\varphi_i$ is chosen a random number, drawn from a uniform probability distribution between $-\pi/2$ and $\pi/2$.

![Simulation geometry](image)

Figure 3.3: Simulation geometry.

The SPH simulation result of the wave elevation at the focus point ($x_f = 390$ m, $y = 0$ m, right in front of the platform) is plotted as the solid red curve in Fig. 3.2. The amplitude of the highest wave peak matches the measured data very well. The peak occurs at $x = 350$ m at $t = 55$ seconds, which is slightly different from the designed focus point and time $(x_f, t_f) = (x = 390$ m, $y = 0$ m; $t = 60$ s). This may be due to the non-linear wave effect and the fluid viscosity which are not taken into account in the first-order linear potential flow theory. Despite of that, the result shows good agreement between measured data and SPH simulation at the wave peak region. This
demonstrates the good potential of using proposed SPH numerical scheme to generate the extreme wave event.

A snapshot of the SPH simulation at the moment that the highest wave is created is shown in Fig. 3.5. As designed, the extreme wave is generated in the centre of the basin in front of the platform, at $x_f \sim (390m, 0m)$ and $t_f \sim 60s$.

3.4 Extreme wave on tension-leg platform

As a proof-of-concept, a real-life application of the extreme wave hitting the tension-leg platform has been studied. In the simulation as shown in Fig. 3.3, a tension-leg platform is located in the middle of the tank. The initial position of centre-of-mass is at $x = 426m$, $y = 0m$ and $z=9.05m$. The front of the platform is located at $x = 390m$ which is also the focus point of the modeled extreme wave. The mass of the platform is $3.7 \times 10^7 \text{kg}$. There are eight tension legs connecting the platform and
Figure 3.5: Top and side view of the SPH simulation of the New Year Wave at the focus point right in front of the platform at time t=60s.
the sea bed 500m below the sea level. A tension leg is essentially a steel pipe and is modeled as an ideal springs following Hooke's law. The spring constant is 42 MN/m and the tension-legs are under a pre-tension of 14.5 MN when the platform is in the equilibrium position.

Figure 3.6: SPH simulation of an extreme wave hitting a tension-leg platform at various time. a) time = 52 s; b) time = 56 s; c) time = 60 s; d) time = 64 s.

Snapshots from the simulation at various times are shown in Fig. 3.6. At 52 s (Fig. 3.6(a)), the wave builds up and approaches to the platform. At 56 s (Fig. 3.6(b)) which is slightly earlier than the designed focus time t=60s, all wave modes focus at the focus point forming an extreme wave event in front of the platform. After 60 s (Fig. 3.6(c)), wave hits the platform. Some part of the wave moves around
the platform, but a large portion of wave hits the deck and pushes the platform. The tension legs are stretched and the motion of the platform is determined by the combined effect of fluid impact force and spring force. At 64 s (Fig. 3.6(d)), wave passes the platform and the platform is pulled back into its equilibrium position under the force of tension legs.

The SPH prediction for the displacement of the tension-leg platform is shown in Fig. 3.7. The maximum displacement is approximately 31m. A large excitation starts at 40s, which is followed by an even larger displacement after 60 s. The horizontal displacement is much larger than the vertical displacement. This is because the toughness of the system is larger on the vertical direction than the horizontal direction, which gives more restriction on the vertical direction. The horizontal displacement peaks at $t = 63s$ and then decreases towards its equilibrium position at $0m$ thanks to the tension-leg force. The relative small vertical displacement shows that the platform is still under safe condition under the impact of the extreme wave simulated in our model.

3.5 Conclusions

In this work, a new numerical method was developed to generate multi-modal and multi-directional sea waves within the framework of SPH. A deforming wall was used to create extreme waves according to the theory of diffractive focusing of multi-directional waves. The proposed SPH scheme was validated by generating an extreme wave for which real-life data is available. The result shows good agreement between measured data and simulation data. As a proof-of-concept, the method was used to simulate the motion of a tension-leg platform hit by an extreme wave. The result showed that SPH can be used to accurately simulate realistic extreme waves and their impact on offshore structures.
Figure 3.7: Vertical displacement ("Heave") and the horizontal displacement ("Surge") of the tension-leg platform under the hit of an extreme wave event in SPH simulation.
Chapter 4

SPH for solid mechanics

4.1 Introduction

SPH simulation of the initiation and propagation of pressure-driven fractures in brittle solids is presented in this chapter. An elasto-plastic damage model is employed to model the solid. Drucker-Prager plasticity model is used to predict material shear failures due to plastic deformation. For tensile failure, the Grady-Kipp damage model is used. These models are coupled together so that both shear and tensile failures can be simulated within the same scheme. Results show that SPH is able to correctly predict the evolution of fracture in brittle solids. The model is applied to the solution of crack propagation in a "line crack" problem. The influence of initial in-situ stresses is also accounted for. Comparison of SPH results for these cases to analytical solutions shows that SPH may be applied to accurately simulate the evolution of fluid-driven fractures in brittle solids.
4.2 Methodology

The density continuity equation and momentum conservation equation in a Lagrangian system are given as

\[
\frac{D\rho}{Dt} = -\rho \nabla \cdot \mathbf{v} \quad (4.1)
\]
\[
\frac{D\mathbf{v}}{Dt} = \frac{1}{\rho} \nabla \cdot \mathbf{\sigma} + \mathbf{g} \quad (4.2)
\]

where \(\rho\) is the density, \(\mathbf{v}\) is the velocity; \(\mathbf{\sigma}\) is the stress tensor; \(\mathbf{g}\) is the external body force per unit mass and \(D/Dt\) denotes the material derivative.

In order to close the system of equations given by equations (4.1)–(4.2) a constitutive model for evaluation of the stress tensor is needed.

4.2.1 Constitutive model

The strain-stress relation follows the general Hooke’s law as

\[
\dot{\mathbf{\sigma}} = 2G\dot{\mathbf{\varepsilon}}_d + K\dot{\mathbf{\varepsilon}}_v \quad (4.3)
\]

where \(\dot{\mathbf{\sigma}}\) is the stress rate, \(G\) and \(K\) are the shear and bulk modulus, \(\dot{\mathbf{\varepsilon}}_d\) and \(\dot{\mathbf{\varepsilon}}_v\) are the deviatoric and volumetric parts of the elastic strain rate tensor respectively. The total strain rate tensor can be computed as

\[
\dot{\mathbf{\varepsilon}} = \frac{1}{2} (\nabla \otimes \mathbf{v} + (\nabla \otimes \mathbf{v})^T) \quad (4.4)
\]

which can be decomposed into the hydrostatic and deviatoric part as

\[
\dot{\mathbf{\varepsilon}} = \left( \dot{\mathbf{\varepsilon}} - \frac{tr(\dot{\mathbf{\varepsilon}})}{3} \mathbf{I} \right) + \frac{tr(\dot{\mathbf{\varepsilon}})}{3} \mathbf{I} \quad (4.5)
\]

\[
= \dot{\mathbf{\varepsilon}}_d + \dot{\mathbf{\varepsilon}}_v
\]

A yield function, \(F(\mathbf{\sigma}, \mathbf{q})\), defines the elastic deformation region as

\[
\chi = \{ \mathbf{\sigma} | F(\mathbf{\sigma}, \mathbf{q}) < 0 \} \quad (4.6)
\]
where \( q \) is the component of internal variables associated with the hardening or softening phenomenon. The flow vector, \( \mathbf{N} \), is computed as

\[
\mathbf{N} = \frac{\partial Q(\sigma, q)}{\partial \sigma}
\]  

(4.7)

where \( Q(\sigma, q) \) is the flow (or plastic) potential. If \( Q(\sigma, q) \) equals to \( F(\sigma, q) \), it is called the associated flow rule; otherwise, it is called non-associated flow rule.

For non-associated flow rule, the elastic strain rate tensor \( \dot{\varepsilon}_e \) and the plastic strain rate tensor \( \dot{\varepsilon}_p \) are given by

\[
\dot{\varepsilon}_p = \dot{\lambda} \mathbf{N}
\]  

(4.8)

\[
\dot{\varepsilon}_e = \dot{\varepsilon} - \dot{\varepsilon}_p
\]  

(4.9)

\[
\dot{\mathbf{q}} = -\dot{\lambda} \frac{\partial Q(\sigma, q)}{\partial q}
\]  

(4.10)

where \( \lambda \) is the plastic consistency parameter or plastic multiplier. The loading-unloading conditions reads

\[
\dot{\lambda} \geq 0, \quad F \leq 0, \quad \dot{\lambda}F = 0
\]  

(4.11)

The hardening internal variables and the plastic parameter are computed from the above initial value problem (4.9), (4.10) and (4.11). This procedure will be discussed in details in the following sections.

4.2.2 Governing equations and corrective terms in SPH

SPH approximation of the density continuity in solids follows the same form as that for fluids in (2.10). In this paper, the most popular SPH approximation of the momentum equation for solid is used and is given below [19]

\[
\frac{Dv_i^\alpha}{Dt} = \sum_j m_j \left( \frac{\sigma_i^\alpha}{\rho_i} + \frac{\sigma_j^\alpha}{\rho_j} - \Pi_{ij}^\delta \delta^{\alpha\delta} + (R_{ei}^\alpha + R_{ej}^\alpha) f_{ij} \right) \frac{\partial W_{ij}}{\partial x_i^\delta} + g_{i}^\alpha
\]  

(4.12)
where, $\Pi_{ij}$ is the artificial viscosity term required to resolve unphysical oscillations associated with shock [53]; $R_{\epsilon i}^{\alpha,\beta}$ and $R_{ij}^{\alpha,\beta}$ are the artificial stress terms proposed for tensile instability correction [54]; $n$ is an exponent dependent on the smoothing kernel; $f_{ij}$ is defined as

$$f_{ij} = \frac{W_{ij}}{W(\Delta d, h)}$$

(4.13)

where $\Delta d$ is the initial particle spacing. In this study, $h$ is assumed to be $1.3\Delta d$ for the cubic spline kernel. For the problems described herein, the exponent $n$ and the correction parameter $\epsilon$ are chosen to be 4 and 0.5 respectively, to correct for the tensile instability.

The rate of strain is approximated in the SPH formulation as follows

$$\dot{\epsilon}_i^{\alpha,\beta} = \frac{1}{2} \left[ \sum_j^N \frac{m_j}{\rho_j} (v_j^{\alpha} - v_i^{\alpha}) \frac{\partial W_{ij}}{\partial x_i^{\beta}} + \sum_j^N \frac{m_j}{\rho_j} (v_j^{\beta} - v_i^{\beta}) \frac{\partial W_{ij}}{\partial x_i^{\alpha}} \right]$$

(4.14)

The XSPH correction [31] to the velocity of a particle is preferred for problems involving tension and fracture propagation, so that particle $i$ is moved according to an average velocity given by

$$\frac{Dx_i^{\alpha}}{Dt} = v_i^{\alpha} + \bar{\epsilon} \sum_j^N \frac{m_j}{\rho_{ij}} (v_j^{\alpha} - v_i^{\alpha}) W_{ij}, \quad \bar{\epsilon} \in [0, 1]$$

(4.15)

### 4.2.3 Failure models

Drucker-Prager plasticity model is used to predict material shear failures due to plastic deformation. For tensile failure, the Grady-Kipp damage model [16–19, 22] is used. These models are coupled together so that both shear and tensile failures can be simulated within the same scheme [18, 19, 22].

#### 4.2.3.1 Drucker-Prager model

The Drucker-Prager yield criterion is used to model the shear failure of brittle materials. The yield criterion $F(\sigma, c)$ represents the surface of a circular cone (Fig. 4.1)
and is given by

\[ F(\sigma, c) = \sqrt{J_2(s)} + \eta I_1(\sigma) - \xi c \]  \hspace{1cm} (4.16)

where \( c \) is the cohesion of the material, \( J_2(s) \) is the second principal invariant of the deviatoric stress tensor, \( s \) is the deviatoric stress tensor, and \( I_1(\sigma) \) is one third of the first invariant of the stress tensor. The parameters \( \xi \) and \( \eta \) are given as

\[ \eta = \frac{6 \sin \phi}{\sqrt{3(3 - \sin \phi)}} \quad \xi = \frac{6 \cos \phi}{\sqrt{3(3 - \sin \phi)}} \]  \hspace{1cm} (4.17)

where \( \phi \) is the friction angle.

The plastic potential function \( Q(\sigma) \) has the following form

\[ Q(\sigma) = \sqrt{J_2(s)} + \bar{\eta} p(\sigma) \]  \hspace{1cm} (4.18)

where the parameter \( \bar{\eta} \) depends on the dilatancy angle \( \psi \) and is given by

\[ \bar{\eta} = \frac{6 \sin \psi}{\sqrt{3(3 - \sin \psi)}} \]  \hspace{1cm} (4.19)
For non-associative flow rule, dilatancy angle \( \psi \) is different from the friction angle \( \phi \). For associative flow rule, they are equivalent which leads to \( \bar{n} = n \). Substituting equation (4.18) into equation (4.9), the plastic strain of a particle takes the form

\[
\dot{\varepsilon}_p = \dot{\lambda} N = \dot{\lambda} \left( \frac{1}{2\sqrt{J_2(s)}} s + \frac{\bar{n}}{3} J_3 \right)
\]  

(4.20)

In the present study, associative isotropic strain softening is included by letting the cohesion, \( c \) be a piecewise linear function of the accumulated plastic strain (Fig. 4.2)

\[
c = c(\varepsilon_p) = c_0 + \kappa(\varepsilon_p)
\]  

(4.21)

where the scalar, \( \kappa \), is associated to isotropic softening. The accumulated plastic strain is then estimated from Eq. (4.10) by considering an associative softening law as

\[
\dot{\varepsilon}_p = -\lambda \frac{\partial F}{\partial \kappa} = \dot{\lambda} \xi
\]  

(4.22)

**Numerical Implementation** The integration algorithm for the Drucker-Prager model is implemented based on elastic-predictor and plastic-corrector steps [55].

According to Eq. (4.3), the stress rate can be computed from the strain rate. After obtaining the stress rate, a trial stress tensor, \( \sigma^{tr} \) can be computed using typical time integration algorithms for particle methods such as a leap-frog method or a verlet method. Substituting the trail stress tensor \( \sigma^{tr} \) into the Drucker-Prager yield surface function \( F(\sigma, c) \) given in Eq. (4.16), a yield criterion can be derived as following:

\[
F(\sigma^{tr}, c) \begin{cases} 
\leq 0, \text{pure elastic deformation, no plastic correction is needed} \\
> 0, \text{plastic correction is needed}
\end{cases}
\]  

(4.23)
Figure 4.2: Cohesion as a piecewise linear function of accumulated plastic strain.

If the trial stress tensor $\sigma^t$ is out of the Ducker-Prager cone, a correction is needed to bring the stress tensor back to the cone. To achieve this, the following equation has to be solved for the plastic multiplier $\Delta \lambda_a$ using the Newton-Raphson method [19].

$$\tilde{F}_a(\Delta \lambda_a) = \sqrt{J_2(s_a)} - G\Delta \lambda_a + \eta I_1(\sigma_a) - K\bar{\eta}\Delta \lambda_a - \xi c_a(\bar{\varepsilon}_p + \xi \Delta \lambda_a) = 0 \quad (4.24)$$

After obtaining the plastic multiplier $\Delta \lambda_a$, the corrected stress on the Drucker-Prager cone can be computed as

$$\sigma_a^{\alpha \beta} = \text{tr} \sigma_a^{\alpha \beta} - \Delta \lambda_a \left( \frac{G}{\sqrt{J_2(\text{tr} s_a^{\alpha \beta})}} \text{tr} s_a^{\alpha \beta} + \frac{K\bar{\eta}}{3} \delta^{\alpha \beta} \right) \quad (4.25)$$

Equivalently, the deviatoric stress and hydrostatic pressure is given by

$$s_a^{\alpha \beta} = \left( 1 - \frac{G\Delta \lambda_a}{\sqrt{J_2(\text{tr} s_a^{\alpha \beta})}} \right) \text{tr} s_a^{\alpha \beta} \quad (4.26)$$

$$p_a = \text{tr} p_a - K\bar{\eta}\Delta \lambda_a \quad (4.27)$$
The updated deviatoric stress, \( s^{\alpha\beta} \) is obtained by scaling down the trial deviatoric stress, \( tr s^{\alpha\beta} \) by a factor depending on the plastic multiplier, \( \Delta \lambda_a \). The updated accumulated plastic strain is computed as Eq. (4.22)

\[
t_{\Delta t} \bar{\varepsilon}_{pa} = t \bar{\varepsilon}_{pa} + \xi \Delta \lambda
\]

(4.28)

### 4.2.3.2 Grady-Kipp damage model

A Grady-Kipp damage model is used to account for the tensile failure [56–58]. In this model, a scalar damage parameter \( D \) is used to characterize the volume-averaged micro-fracture of the volume of the material represented by each SPH particle. The damage parameter \( D \) describes the correction of the tensile stress between particles and varies between 0 and 1. \( D = 0 \) means the material is undamaged and the full tensile load can be transmitted. \( D = 1 \) means the material is fully damaged and particles cannot sustain any tensile stress, leading to a partial macro-crack. Multiple macro-cracks are connected together leading to the macroscopic fragmentation.

The damage parameter \( D \) can be computed as [58]

\[
dD^{\frac{3}{2}} \frac{dt}{dt} = \frac{m + 3}{3} \alpha \frac{1}{3} \varepsilon^{\frac{m}{3}}
\]

(4.29)

where

\[
\alpha = \frac{8 \pi c_g^3 k}{(m + 1)(m + 2)(m + 3)}
\]

(4.30)

where \( k \), \( m \) and \( c_g \) are material fracture parameters. A typical value for \( c_g \) is \( 0.4 \times c_s \) where \( c_s \) is the speed of sound in the material [59].

The damage parameter is initially 0 for all particles. It starts evolving according to Eq. (4.29) if some certain damage criterion is satisfied. Melosh [59] proposed an effective tensile strain \( \bar{\varepsilon} = \sigma_{\text{max}}/(K + \frac{4}{3}G) \) where \( \sigma_{\text{max}} \) is the maximum positive principle stress and \( K \) and \( G \) are the bulk and shear modulus. Damage occurs when the effective strain \( \bar{\varepsilon} \) exceeds a threshold value \( \varepsilon_{\text{min}} = (Vk)^{\frac{1}{m}} \) where \( V \) is the volume of the SPH particle.
Thus, if $\tilde{\epsilon} \leq \tilde{\epsilon}_{\text{min}}$, no damage occurs in the rock particles. Conversely if $\tilde{\epsilon} > \tilde{\epsilon}_{\text{min}}$, damage occurs and is evaluated from Eq. (4.29) through the time integration scheme.

After evaluating the damage parameter $D$, the trial principal stress tensor $\sigma_p^{tr}$ is corrected by the factor $(1 - D)$ if $\sigma_p^{tr}$ is positive $\sigma_p^{tr} \geq 0$. For the negative component of a principal stress tensor, no correction is required. The corrected principle stress tensor $\sigma_p^{cor}$ takes the following form

$$\sigma_p^{cor} \leftarrow (1 - D)\sigma_p^{tr} \quad (4.31)$$

The updated stress tensor in the global frame is then obtained by rotating the principal stress back to the global frame. Further reference to implementation of the Grady-Kipp damage model is given by Das and Cleary [17].

### 4.2.3.3 Coupling of Grady-Kipp and Drucker-Prager

It is necessary to couple both shear and tensile failure model to obtain a full failure model. In order to achieve this, the small tensile part of the Drucker-Prager cone ($I_1(\sigma_a) < 0$) needs to be cut to avoid conflicting with the Grady-Kipp model.

Hence, the modified criterion is

- if $I_1(\sigma_a) < 0$, either the stress state is elastic or a tensile correction is needed.
- If $F_a(\sigma_a, c_a) \leq 0$ and $I_1(\sigma_a) \geq 0$, the stress state is purely elastic.
- If $F_a(\sigma_a, c_a) > 0$ and $I_1(\sigma_a) \geq 0$, the stress state is outside the yield surface and a plastic correction is needed.

This coupling allows a separate computation of tensile failures by the Grady-Kipp damage model and shear failure by the Drucker-Prager plasticity model.

### 4.2.4 Stress boundary conditions

In solid mechanics experiments, it is common to apply the stress boundary condition and velocity boundary condition on the sample. Therefore, an accurate modeling
of both boundary conditions are important. The implementation of these boundary conditions both with and without dummy particles is derived in details in the reference [21]. In this section, only main conclusion is mentioned.

4.2.4.1 With dummy particles

Fig 4.3 is the schematic stress boundary conditions with dummy particles. \( \Omega_s \) and \( \Omega_d \) represents the region of the real solid particles and dummy solid particles respectively. The dummy particles are located outside the solid interface \( \Gamma_I \), on which stress, \( \sigma_s \), is applied. It is to be noted that the direct application of prescribed stress on the dummy particles is not accurate to model the boundary condition because it is difficult to maintain the stress continuity at the interface. Thus, the stress value on dummy particles has to be carefully chosen in order to enforce the imposed stress value at the boundary.

\[
\sigma_d = 2\sigma_o - \frac{\sum_{a \in \Omega_s \cap \Lambda_d} \sigma_a \frac{m_a}{\rho_a} W_{ad}}{\sum_{a \in \Omega_s \cap \Lambda_d} \frac{m_a}{\rho_a} W_{ad}} \tag{4.32}
\]

where \( \sigma_o \) is the imposed stress on the interface, \( \sigma_a \) is the stress of a real solid particle \( a \) and \( \Lambda_d \) is the support domain of dummy particle \( d \). This dummy stress \( \sigma_d \) allows
to balance the material response from the real particles and enables the system to converge towards the prescribed value $\sigma_o$ at the interface.

In order to maintain velocity continuity at the interface, the velocity of the dummy particles is also taken to be the interpolated value as

$$v_d = \frac{\sum_{a \in \Omega_o \cap \Omega_d} v_a \frac{m_a}{\rho_a} W_{ad}}{\sum_{a \in \Omega_o \cap \Omega_d} \frac{m_a}{\rho_a} W_{ad}}.$$  \hspace{1cm} (4.33)

### 4.2.4.2 Without dummy particles

In certain situations, it is difficult to fulfill the support domain with dummy particles. For example, in a line or hole crack problem, if the geometry is small, there is no enough space to fill the required number of dummy particles to fully support the kernel region. Another situation is that when applying stress boundary conditions on a moving interface or an opening fracture, it is difficult to dynamically generate new dummy particles along with the moving interface or newly created fracture interface. Therefore, a technique of applying stress boundary condition without dummy particles is necessary.

![Figure 4.4: Schematic stress boundary conditions without dummy particles](image)

As shown in Fig. 4.4, $\Omega_o$ is the region of real solid particles and $\Omega_f (\subset \Omega_o)$ is the boundary particle region on which the stress $\sigma_o$ is imposed. The final complete format of momentum equation for a solid particle $a$ (ignoring artificial viscosity term and tensile instability correction term) takes the following form [21]

$$\frac{Dv_a^\alpha}{Dt} = \sum_{b \in \Lambda_d} m_b \left( \frac{\sigma_a^{\alpha\beta}}{\rho_a^2} + \frac{\sigma_b^{\alpha\beta}}{\rho_b^2} - \frac{2\sigma_{fa}^{\alpha\beta}}{\rho_f \rho_b} \right) \frac{\partial W_{ab}}{\partial x_a^\beta} \frac{\partial^2}{\partial x_a^\alpha}.$$  \hspace{1cm} (4.34)
where $\sigma_{fa}$ reads

$$
\sigma_{fa} = \begin{cases} 
\sigma_0 & \text{if } a \in \Omega_f \\
\frac{\sum_{b \in \Omega_f \cap \Lambda_d} \sigma_{fb} \frac{m_b}{\rho_b} W_{ab}}{\sum_{b \in \Lambda_d \cap \Omega_f} \frac{m_b}{\rho_b} W_{ab}} & \text{if } a \in \Omega_f \cap \Lambda_d \\
0 & \text{elsewhere}
\end{cases}
$$

(4.35)

### 4.2.5 Velocity boundary conditions

#### 4.2.5.1 With dummy particles

A free-slip or no-slip boundary condition at a wall can be imposed by the choice of solid wall velocity. To impose a no-slip boundary condition, the velocity field is obtained by extrapolating the smoothed velocity field of regular particles to the dummy particle positions. Then, the velocity

$$
v_d = v_0 - \frac{\sum_{a \in \Omega_d \cap \Lambda_d} v_a \frac{m_a}{\rho_a} W_{ad}}{\sum_{a \in \Omega_d \cap \Lambda_d} \frac{m_a}{\rho_a} W_{ad}}.
$$

(4.36)

is assigned to the dummy particle $d$, where $v_0$ is the prescribed wall velocity. It is not necessary that the smoothing function taken here should have the same form as regular particles. Compared to Morris et al. [28], this method does not require explicit information about the geometry of the boundary and as the calculation of the extrapolated velocities can be restricted to interface dummy particles, the computational overhead is insignificant.

The above mentioned treatment of solid wall is not sufficient for modeling elasto-plastic flow; the presence of dummy particle is necessarily to be reflected in momentum Eq. (4.12) by specifying the stress components. Stress value can be imposed to dummy particle in a number of ways, the most general treatment was derived by Randles and Libersky [60]. Here, a simple technique is employed to obtain a smooth stress profile near solid boundary. The local stress tensor of dummy particles are estimated by extrapolating stress from the interior regular particles according to

$$
\sigma_d = \frac{\sum_{a \in \Omega_d \cap \Lambda_d} \sigma_a \frac{m_a}{\rho_a} W_{ad}}{\sum_{a \in \Omega_d \cap \Lambda_d} \frac{m_a}{\rho_a} W_{ad}}.
$$

(4.37)
This approach together with above velocity boundary condition is straightforward and reduce the computational cost compared with other methods. It has been seen that this combination of boundary conditions work well and produce a smooth variation of stress profile near the solid boundary.

4.2.5.2 Without dummy particles

The procedure follows the same approach as the one described in §4.2.4.2. Using the same notations, only an update of the velocity is needed.

\[
v_a = v_0 + \frac{\sum_{b \in \Omega_i \cap \Omega_a} (v_b - v_0)^m \rho_b W_{ab}}{1 - \left( \sum_{b \in \Omega_i \cap \Omega_a} \frac{m}{\rho_b} \right)^2}.
\] (4.38)

4.3 Numerical examples

4.3.1 Line Crack

The line crack problem is used to validate the proposed coupled fracture model where analytical solution is available. In this case, a line crack with length 0.4m is located in the middle of a 5m x 5m rock medium plate and the crack is pressurized by 15 MPa as shown in Fig. 4.5.

Figure 4.5: Schematic of the line crack problem
It is not necessary to create a real opening crack in the model and a 0.4m-long virtual crack line is plotted in the middle of the rock in Fig. 4.6(b) only for the explanatory purpose. Particles at two layers above and below this virtual crack line are labeled as “crack boundary particles” which are shown as grey particles in Fig. 4.6(b). A vertical compressive pressure of 15 MPa have been applied on these “crack boundary particles” with opposite normal direction above and below the crack by using the technique of applying stress boundary condition without dummy particles previously described in § 4.2.4.2.

Particles which make up opposing crack surfaces do not transmit tensile stresses. To achieve this, the kernel support domain for crack boundary particles is truncated at the crack interface, as shown in Fig. 4.6. When computing the interaction on crack boundary particle “i”, only particles within the truncated domain (dashed circle) are considered and crack boundary particles on the other side of the crack are ignored. By doing this, we can eliminate the tensile interaction between particles above and below the crack.

The analytical solution for the line crack problem under constant pressure is given by Sneddon [61]

\[
\begin{align*}
    u_y(x, 0) &= (1 - \nu^2) \frac{2P}{E} \sqrt{c^2 - x^2} \\
    \sigma_{yy}(x, 0) &= P \left( \frac{x}{\sqrt{x^2 - c^2}} - 1 \right)
\end{align*}
\]

where \(P\) is the applied pressure in the crack and \(2c\) the length of the crack.

The SPH simulations results of stress and displacement fields under an applied pressure 15MPa are shown in Fig. 4.7. The SPH stress field along the crack axis is compared with the analytical solution in Fig. 4.8. The numerical result is in good agreement with analytical solution especially at the crack tip where stress concentrates.

In order to study the propagation of the fracture, this model is again simulated with material non-linearity. Fig. 4.9 shows the variation of vertical stress beyond the
Figure 4.6: Truncation of the support domain for crack boundary particles and their neighbors. (a) Normal case, (b) Truncated case around the crack.
Figure 4.7: SPH results of the line crack problem submitted to a constant compressive vertical pressure of 15MPa (a) Vertical Stress (Pa) (b) Vertical Displacement (mm)

Figure 4.8: Comparison of vertical stress between SPH result (×) and analytical solution (-) of the elastic line crack problem with in-situ pressure $P = 15$ MPa.
crack tip in the medium as the fracture propagates when a higher fracture pressure of $P = 50$ MPa is applied.

4.4 Conclusions

An elasto-plastic damage model was employed in this work to model material non-linearity, fracture initiation, and propagation, within the SPH framework. A new method of applying stress boundary condition and velocity boundary condition both with and without dummy particles was developed and validated in the work. Results showed that SPH is able to correctly predict the evolution of fracture in brittle solids. The result predicted from SPH were compared to the analytical solution in a "line crack" problem and found to be in good agreement. It showed that the proposed SPH method is a promising numerical tool to simulate solid deformation problems and to predict crack initiation and propagation. The main advantage of this method is its meshless property, leading to adaptability, and allowing crack growth and bifurcation in any direction without special treatment. Moreover, SPH is already confirmed to produce good results in studying the behavior of fluids making it a good choice to study fluid-solid coupling problem, where strong interactions among fluids, solids and fracture occur.
Figure 4.9: SPH result showing vertical stress distribution at various time steps in fracture propagation in the line crack problem with fracture pressure $P = 50$ MPa.
Chapter 5

SPH modeling of fluid and deformable-solid coupling

5.1 Introduction

The fluid-structure interaction (FSI) is a complex problem which arises in many engineering applications. Among all FSI problems, there are a variety of natural phenomena and engineering applications where the fluid-induced loads on the structure are large enough to enable the initiation and propagation of cracks, which may ultimately result in catastrophic failure of the structure. This occurrence is termed as fluid-structure-fracture interaction (FSFI). This type of coupled interaction plays an important role in numerous industry processes which include, to name only a few, hydraulic fracturing for oil recovery, water jet excavation of materials, underwater implosion and underbody blast. Even in its most basic form, FSFI is a complicated process to model as it involves the coupling of at least three processes. (i) the mechanical deformation and fracture initiation induced by the fluid pressure; (ii) the flow of fluid within the fracture; and (iii) the fracture propagation. The development of a computational framework for such a system is challenging. It requires accounting for all possible interactions of fluid and solid which are governed by different physics
laws. It also calls for the modeling of large deformation, geometrical discontinuity, material failure and crack propagation, and the computation of flow induced loads on evolving fluid-structure interfaces. In addition, the velocity and stress continuity have to be well maintained at the fluid-solid interface which is not an easy task when it involves high density jumps and topological changes.

Many numerical methods have been developed in the study of FSI problems. Among these methods, Arbitrary Lagrangian-Eulerian (ALE) method and Immersed Boundary (IB) method are the most commonly used. An ALE method allows arbitrary motion of grid/mesh points with respect to their frame of reference by taking the convection of these points into account as described in [5-7] and many works thereafter. In an ALE formulation, the finite element mesh need not adhere to the material to be fixed in space but may be moved arbitrarily relative to the material. Alternative to ALE technique, IB methods [8-10] consider a finite difference grid for the fluid domain with an immersed set of non-conforming boundary points that are mutually interconnected by an elastic law. The local body forces at the position of the solid points are computed to model the interaction between fluid and solid boundary. The velocity and stress continuity at the interface are also maintained through the kinematic constraint imposed by this body force.

These methods can successfully predict the FSI process in one aspect or another [11]. However, these grid-based methods do not typically work well for simulation of large deformations, fractures, and fragmentation, especially if discontinuities occur in the solid failure process induced by fluid loads. The most widely used strategy to handle a moving discontinuity is to remesh the domain in each step such that the discontinuity is accurately represented in the simulation mesh and exists along the boundaries of mesh elements. However, this approach introduces numerical difficulties and can be computationally expensive. For example, due to the representation of cracks, artificial crack paths may emerge due to prescribed mesh linkages. In addition, the coupling of one numerical method for fluids and another for solids can introduce
Chapter 5. *SPH modeling of fluid and deformable-solid coupling* 73

numerical errors from inconsistencies and prove difficult for code maintenance.

SPH method is suitable for simulating this problem as the method has advantages in handling large structure deformation under a high pressure load thanks to its Lagrangian nature. This chapter devises a general SPH numerical modeling framework to model the interaction between fluids and brittle solids with highly nonlinear deformation and dynamic fracture initiation and propagation. SPH modeling of fluid and elastoplastic-solid coupling is presented. Fluids and solids are treated as a single system for the entire domain, and are computed using the same stress representation within a uniform SPH framework. Two stress-coupling approaches are proposed to enforce the stress continuity at the fluid-solid interface, namely, continuum approach and stress-boundary-condition approach. Both approaches allow interactions between particles from different materials when solving the momentum equation. A corrected SPH approximation form of the density continuity equation is implemented to handle the density discontinuity of the two phases at the interface. The method is applied to simulate rock deformation in a variety of rock mechanics test cases, including a hydrostatic problem and a fluid-pressurized borehole problem in the presence of in-situ stresses where analytic solutions are available. The results obtained from two approaches of stress-coupling are also compared. The simulation of fracture initiation and propagation in the hydrofracturing case with and without in-situ stresses are also presented as the application cases. Good results demonstrate that SPH has the potential to accurately simulate the hydraulic-fracturing phenomenon in brittle rocks.

5.2 Methodology

In this section we describe first the SPH formulation for the fluid and solid phases. The coupling of the two phases is then discussed.
5.2.1 Fluid model

In SPH formalism, the Navier Stokes equations take the following form [14]

\[
\frac{d\rho_i}{dt} = \rho_i \sum_j^N \frac{m_j}{\rho_j} (v_j^\alpha - v_i^\alpha) \frac{\partial W_{ij}}{\partial x_i^\alpha} \tag{5.1}
\]

\[
\frac{dv_i^\alpha}{dt} = \sum_j^N m_j \left( \frac{\sigma_i^{\alpha\beta}}{\rho_i^\beta} + \frac{\sigma_j^{\alpha\beta}}{\rho_j^\beta} \right) \frac{\partial W_{ij}}{\partial x_i^\beta} + f_i^\alpha \tag{5.2}
\]

where

\[
\sigma^{\alpha\beta} = -\rho \dot{\epsilon}^{\alpha\beta} + \mu \varepsilon^{\alpha\beta} \tag{5.3}
\]

and the SPH approximation of shear strain rate \( \dot{\varepsilon}^{\alpha\beta} \) for particle \( i \), with neighbor particles \( j \), is written as

\[
\dot{\varepsilon}^{\alpha\beta}_i = \sum_j^N \frac{m_j}{\rho_j} (v_j^\beta - v_i^\beta) \frac{\partial W_{ij}}{\partial x_i^\alpha} + \sum_j^N m_j (v_j^\alpha - v_i^\alpha) \frac{\partial W_{ij}}{\partial x_i^\beta} - \frac{2}{3} \sum_j^N \frac{m_j}{\rho_j} (v_j^\gamma - v_i^\gamma) \frac{\partial W_{ij}}{\partial x_i^\gamma} \tag{5.4}
\]

The pressure of a fluid particle is computed from the equation of state (EOS). Morris' and Monaghan's EOS are both commonly used, where the form of Morris' EOS, for example, is

\[
p = c_0^2 (\rho - \rho_0) \tag{5.5}
\]

where \( \rho_0 \) is the reference density and \( c_0 \) is the speed of sound of the fluid. The speed of sound is usually chosen as ten times the maximum fluid velocity, or \( c_0 = 10V_{\text{max}} \). These versions of EOS are usually good for simulating cases with relatively low fluid pressures. However, in order to generate sufficiently high pressures in the hydraulic fracturing process (usually at MPa level), the incompressible assumption needs to be more strictly enforced. In this sense, the common choice of EOS is too soft for simulating the hydraulic fracturing problem. Therefore the true EOS for water, the
Tait equation, is used here, and represents more closely the incompressible nature of water. The Tait equation takes the form as

\[ p = \frac{K}{n} \left( \left( \frac{\rho}{\rho_0} \right)^n - 1 \right) + p_0 \]  \hspace{1cm} (5.6)

where \( K = 2.15 \) GPa, \( n = 7.15 \) and \( \rho_0 \) and \( p_0 \) are the reference density and pressure respectively. The real speed of sound of water, \( c_0 = 1484 \) m/s, should be used here as well. The requirement of a realistic speed of sound is usually not enforced in a typical SPH fluid model, as it demands a very small timestep. To avoid this small timestep, the speed of sound is usually artificially reduced by two orders of magnitude. However, in this work we couple the fluid model to a solid model which requires an even larger speed of sound (and hence smaller timestep) in order to achieve an accurate simulation. It is therefore the speed of sound in the solid phase which determines the timestep, so there is no additional computational expense introduced by using the real speed of sound for the fluid as opposed to one that is artificially reduced.

### 5.2.2 Solid model

The SPH modeling for solid including elastic deformation, tensile and shear failure follows the exact the same theory described in previous sec. 4.2 in the chapter of SPH for solid mechanics.

### 5.2.3 Fluid-solid coupling

Numerical modeling of the interaction between fluids and deformable solids remains a big challenge due to their strong non-linearity and multidisciplinary nature. There are two numerical approaches to solve such problems. One approach is to consider the fluid and solid in a same mathematical framework, and to develop a single system for the entire domain in order to develop a uniform SPH framework. Another approach is to treat the fluid and solid as two computational domains which are solved separately.
Chapter 5. SPH modeling of fluid and deformable-solid coupling

with their respective governing equations. Interface boundary conditions must then be applied between the two phases in order to couple them. In this work, both approaches have been developed and compared. The details are presented below.

Consider a computational domain, denoted by \( \Omega \), with an external boundary \( \Gamma \). The domain includes the solid domain \( \Omega_s \), and the fluid domain, \( \Omega_f \); i.e., \( \Omega = \Omega_f \cap \Omega_s \). The fluid-structure interface is defined by \( \Gamma_{fs} \) as shown in Fig. 5.1.

5.2.3.1 Stress coupling approaches

For the interaction of fluid and solid at the interface \( \Gamma_{fs} \), it is required to follow the Dirichlet and Neumann boundary conditions, i.e.,

\[
\begin{align*}
\nu_f^\alpha &= \nu_s^\alpha \quad \text{on } \Gamma_{fs} \\
\sigma_f^{\alpha\beta} n_{fs}^\alpha &= \sigma_s^{\alpha\beta} n_{fs}^\alpha \quad \text{on } \Gamma_{fs}
\end{align*}
\]

where subscript \( f \) and \( s \) denote fluid and solid phases respectively. Two stress coupling approaches have been proposed to enforce the velocity and stress continuity at the fluid-solid interface:

Figure 5.1: Schematic of fluid and solid domain in fluid-solid coupling.
Chapter 5. *SPH modeling of fluid and deformable-solid coupling*  

- **Approach I: Continuum approach to treat stress continuity at fluid-solid interface** In the continuum approach, the velocity and stress continuity at the interface is maintained by using kernel summation, which allows interactions between particles from different materials when solving the momentum equation, i.e. for a fluid particle $i$,  
  \[
  \frac{dv_j^f}{dt} = \sum_{j \in \Omega_f} m_j \left( \frac{\sigma_{ij}^{\alpha \alpha}}{\rho_i^f} + \frac{\sigma_{ij}^{\alpha \beta}}{\rho_j^f} \right) \frac{\partial W_{ij}^f}{\partial x_i^f} + \sum_{j \in \Omega_s} m_j \left( \frac{\sigma_{ij}^{\alpha \alpha}}{\rho_i^s} + \frac{\sigma_{ij}^{\alpha \beta}}{\rho_j^s} \right) \frac{\partial W_{ij}^s}{\partial x_i^s} + f_i^o \tag{5.9}
  \]
  
  and vice-versa for solid particles.

  In this scheme, fluid and solid are treated as a single system for the entire domain. This is achieved by using the stress representation of the fluid in Eq. (5.2), so that fluid and solid can be solved in a uniform framework. The interaction of shear forces is also accounted for automatically. Finally, the resulting uniform framework based on the same stress representation for each phase significantly eases the computational implementation.

- **Approach II: Stress-boundary-condition approach to treat stress continuity at fluid-solid interface:** In this approach, the stress continuity at fluid-solid interface is maintained by using kernel interpolation technique. It assumes the fluid particles near the interface as dummy particles when they are interacting with rock particle adjoining to the interface [18]. In order to transmit the stress of the fluid particle to the neighboring rock particles, an interfacial stress tensor at the dummy fluid particles is introduced. Suppose a solid particle $a^s$ has a dummy (fluid) particle $b^f$ in its neighborhood. The interfacial stress tensor of the dummy particles is to be applied as boundary condition on the solid particles $a^s$ when evaluating momentum for $a^s$. To formulate the interfacial stress tensor at $b^f$, stress tensors of the neighboring solid particles are extrapolated to the position of fluid particle $b^f$ as  
  \[
  \hat{\sigma}_{b^f} = \frac{\sum_{c^s \in \Omega_{b^f}^s} \frac{m_{c^f}}{\rho_{c^f}} \sigma_{c^f} W_{c^f b^f}}{\sum_{c^s \in \Omega_{b^f}^s} \frac{m_{c^f}}{\rho_{c^f}} W_{c^f b^f}} \tag{5.10}
  \]
where $\Lambda_{bf}^t$ is the sub-support domain of particle $b^t$ containing all neighboring rock particles.

Then, the interfacial stress tensor, $\bar{\sigma}_{bf}^\alpha$ is estimated as

$$
\bar{\sigma}_{bf}^\alpha = \begin{cases} 
2\sigma_{bf}^\alpha - \sigma_{bf}^\beta & \text{if } \alpha = \beta \\
\sigma_{bf}^\alpha & \text{if } \alpha \neq \beta
\end{cases}
$$

(5.11)

where $\sigma_{bf}^\alpha$ is the calculated stress of fluid particle $b_f$ at time $t$ (Eq. (5.3)). Now, interfacial stress, $\bar{\sigma}_{bf}^\alpha$ is incorporated into momentum equation of $a^s$ as follows:

$$
\frac{Dv_{ia}^s}{Dt} = \sum_{b^t \in \Lambda_{as}^t} m_{b^t} \left( \frac{\sigma_{a^t}^{\alpha^t}}{\rho_{a^t}} + \frac{\sigma_{b^t}^{\beta^t}}{\rho_{b^t}} - \Pi_{a^t b^t} \delta_{\alpha^t \beta^t} \right) \frac{\partial W_{a^t b^t}}{\partial x_{ia}^s} + \sum_{b^t \in \Lambda_{a^t}^t} m_{b^t} \left( \frac{\sigma_{a^t}^{\alpha^t}}{\rho_{a^t}} + \frac{\sigma_{b^t}^{\beta^t}}{\rho_{b^t}} - \Pi_{a^t b^t} \delta_{\alpha^t \beta^t} \right) \frac{\partial W_{a^t b^t}}{\partial x_{ia}^s}
$$

(5.13)

The above procedure transmits fluid pressure into the surrounding solid medium by maintaining the traction continuity condition in the interface. It is to be mentioned that simultaneous integration of the governing equations for solid and fluid particle is required to be performed in a same time step.

5.2.3.2 Correction in density computation

For different densities of two phases, the calculation of density on the interface is very important. A deficiency of density in the particles near the interface may arise if particles of different phase, but within the same support domain, are not regarded as neighboring particles. On the other hand, neighboring particles with different mass contributing to calculation of the continuity equation can cause inappropriate values of density.

In the coupled scheme described here we apply the continuity equation shown in Eq. 5.1. This equation is the most original SPH discretization of the density continuity equation. It is however not commonly used in single phase models since $\rho_i$ and $\rho_j$ are
typically approximately equivalent, and cancel. This is not the case for the coupled fluid-solid framework and the original form must be used, as would be the case in a multiphase fluid framework.

5.3 Numerical validations

Hydrostatic problem and fluid-pressured borehole problem have been chosen to validate the proposed numerical scheme against the analytical solution. Despite its simpleness, both tests provide a rigorous check whether the density calculation of different phase is correct and whether velocity and stress continuity are well maintained at the fluid-solid interface with high density jumps and material shape deformations.

5.3.1 Fluid-solid hydrostatic validation

The examination of hydrostatic fluid pressure and solid stress in a solid tank partially filled with water is a good benchmark problem to validate the fluid-solid coupling. In this section, the hydrostatic pressure of fluid and development of vertical stress in the solid are obtained through application of the fluid-solid coupling scheme. Results are then compared to the analytic solution and discussed.

Illustrated in Fig. 5.2, the numerical model consists of three domains, fluid, solid, and boundary. The fluid is water with $\rho = 1000 \text{ kg/m}^3$. Properties for the solid phase represent gypsum, with density of 1540 kg/m$^3$, Young's modulus of 5.96 GPa and Poisson's ratio of 0.2. The periodic boundary condition is applied in the horizontal direction and free-slip boundary condition on the bottom boundary. In order to avoid the zero energy mode under the gravity force, numerical damping and the XSPH correction are applied to both fluid and solid phases. In Fig. 5.3 the numerical result of fluid hydro-static pressure and solid geo-static stress (vertical stress $\sigma_{yy}$ in this case) obtained from both continuum approach and stress-boundary-condition approach are compared to the analytic solution $\rho gh$. Good results are obtained from
both approaches. A stable fluid pressure and solid stress are observed and develop according to the density of each individual phase, agreeing well with the analytic solution. At the interface the stress continuity between two phases is also well maintained where solid particles are compressed by the hydrostatic pressure from fluid correctly. Large error on density calculation usually occurs at the phase interface because of the truncated support domain and density discontinuity between two different physical systems. However, by taking advantage of the alternative format of the density continuity equation as shown in Eq. (5.1), the density of both fluid and solid phases at the interface is well computed. It shows that results from both approaches are reasonably good, although the result obtained from continuum approach is more close to the analytic solution in this case.

5.3.2 Pressurized borehole validation

The pressurized hole in a plate problem has been considered as another benchmark case to validate the fluid-solid coupling. The schematic diagram is shown in Fig. 5.4(a). The radius of the hole is 0.1m while the dimensions of the plate are
Chapter 5. SPH modeling of fluid and deformable-solid coupling

Figure 5.3: Hydro-static pressure in fluid phase (right) and geo-static stress in solid phase (left) of the simulation, compared with the analytical solution (solid line). Results from both continuum approach (×) and stress-boundary-condition approach (○) are plotted.
5m × 5m. Fluid particles (red particles in Fig. 5.4(b)) are fully filled inside the hole with constant pressure 4MPa. In this model, the in-situ horizontal stress \( \sigma_{xx} = 1 \)MPa and vertical stress \( \sigma_{yy} = 2 \)MPa are applied on the plate boundary. This setup also represents a simplified model of the injection of fracturing fluid into a borehole which is one of the main process during hydraulic fracturing.

For a cylindrical hole in an infinite isotropic elastic medium under plain strain condition, the solution for stress distribution is given by \([62]\) and is presented in the following equations.

![Figure 5.4: (a) Schematic of the circular opening problem. (b) Particle packing for fluid particles (red) inside hole and solid particles (blue) as solid plate.](image)

\[
\sigma_r = \frac{1}{2}(\sigma_H + \sigma_h) \left( 1 - \frac{a^2}{r^2} \right) + \frac{1}{2}(\sigma_H - \sigma_h) \left( 1 + 3 \frac{a^4}{r^4} - 4 \frac{a^2}{r^2} \right) \cos(2\theta) + P \frac{a^2}{r^2} \quad (5.14)
\]

\[
\sigma_\theta = \frac{1}{2}(\sigma_H + \sigma_h) \left( 1 + \frac{a^2}{r^2} \right) - \frac{1}{2}(\sigma_H - \sigma_h) \left( 1 + 3 \frac{a^4}{r^4} \right) \cos(2\theta) - P \frac{a^2}{r^2} \quad (5.15)
\]
Chapter 5. **SPH modeling of fluid and deformable-solid coupling**

where \( a \) is the radius of the hole, \( \sigma_H = \text{max}(\sigma_{xx}, \sigma_{yy}) \), \( \sigma_h = \text{min}(\sigma_{xx}, \sigma_{yy}) \) and \( P \) is the pressure on the borehole boundary which is same as the fluid pressure in this case.

The concentration of stress field along vertical \( y \)-axis (Fig. 5.4) is considered for comparison. Along \( y \)-axis, the horizontal stress \( \sigma_{xx} \) and vertical stress \( \sigma_{yy} \) are equivalent to \( \sigma_\theta \) and \( \sigma_r \), respectively. Comparison of the SPH method against the analytical solution shows good agreement in Fig. 5.5. Both \( \sigma_{xx} \) and \( \sigma_{yy} \) reach to the in-situ boundary stress 1 MPa and 2 MPa respectively at the plate boundary edge (\( y = 0.5 \) m). The enforcement of stress continuity at fluid-solid interface requires \( \sigma_{yy} \) equivalent to applied fluid pressure (4 MPa) at the circular hole boundary. The SPH result of \( \sigma_{yy} \) indeed approaches to 4MPa at the borehole boundary and because of the discrete particle packing we cannot have the particle exactly located at the boundary surface, therefore a slightly higher value is expected as \( \sigma_{yy} \) is a monotone increasing function versus \( y \). In addition, results obtained from two stress-coupling approaches are also plotted and compared. The good agreement with analytical solution from both approaches indicate that both two methods are capable of enforcing the stress continuity and transmit the interface force correctly at the phase interface.

5.4 **Application on hydraulic fracturing**

Hydraulic fracturing can be broadly defined as the process by which a fracture initiates and propagates due to hydraulic loading (i.e., pressure) applied by a fluid. Examples and applications of hydraulic fracturing are abundant in geo-mechanics. On the application side, fracturing of oil and gas reservoirs using a mixture of viscous hydraulic fluids and proppant is the most commonly used enhanced oil recovery technique in the oil and gas industry [63]. Hydraulic fracturing is also used in the geothermal energy industry to increase the surface area for heat exchange, and the formation permeability [64].
Figure 5.5: Horizontal stress $\sigma_{xx}$ (a) and vertical stress $\sigma_{yy}$ (b) along vertical axis ($x = 0$) of the simulation, compared with the analytical solution (•). Results from both continuum approach (o) and stress-boundary-condition approach (x) are plotted.
Even in its most basic form, hydraulic fracturing is a complicated process to model as it involves the coupling of at least three processes. (i) the mechanical deformation induced by the fluid pressure on the fracture surfaces; (ii) the flow of fluid within the fracture; and (iii) the fracture propagation. Computational simulations are heavily used to investigate the process of rock degradation and failure involved in hydraulic fracturing. Using particle methods with no prescribed geometric linkages (such as in a mesh or a grid) allows high deformations to be dealt with easily in cases where grid-based methods would require expensive re-meshing.

5.4.1 Hydraulic fracturing simulation without in-situ stress

Presented here is a proof-of-concept simulation of a hydraulic fracture. The initial geometry of this model consists of a notched block of solid particles, where the notch is representative of a perforation as would be performed in a well completion. Water is injected at the notch by using a piston to drive the fluid with a speed of 1 m/s. This geometry is illustrated in Fig. 5.6. Material properties in this case are identical to those applied in the hydrostatic case, and are representative of water and gypsum. The XSPH correction is only applied on solid particles in this simulation.

As this is a proof-of-concept model, the simplest set of boundary conditions has
been applied. The free-slip boundary condition is applied at the walls of the injection region. Free-slip is also applied at the right-most boundary of the solid block, in order to prevent the fluid injection from simply displacing the entire solid block. The top and bottom surfaces of the solid block are considered to be a free surface. For the injection region, a no-slip condition could be applied, though it is expected that this would have little impact on the resulting fracture behavior.

Figure 5.7: Maximum positive principle stress at the fracture tip immediately before fracture propagation begins (t=0.01s) in the case of hydraulic fracturing simulation without in-situ stress.

As the piston begins to move, pressure and stress begin to build in the fluid and solid respectively. This causes an increase in the maximum positive principle stress $\sigma_{\text{max}}$ at the notch tip, which reaches a critical value at t=0.01s (shown in Fig. 5.7). This corresponds to the first observation of damage in the solid particles at the tip of the notch and initiates the propagation of a tensile fracture. This propagation is shown in Fig. 5.8, where we plot the damage parameter, $D$, used in the Grady-Kipp damage model.

The results show that the propagation of the fracture occurs in three distinct stages. The first stage is linear propagation of the fracture along the centerline of the solid block (Fig. 5.11(a)). The next stage is bifurcation of the fracture (Fig. 5.11(b)).
Finally Fluid begins to enter the fracture as it opens (Fig. 5.11(c) and Fig. 5.11(d)).

It should be noted that the presence of damage in this simulation indicates only that the material is weakened in the tensile regime, and does not necessarily correspond to an opening fracture. In the model shown, the bifurcation of the damage region results in two branches. The lack of symmetry in the fracture bifurcation is most likely due to numerical effects. This model is packed with a hexagonal packing of SPH particles, where this packing is not symmetric about the centerline of the initial notch.

The fact that damage is not equivalent to fracture opening should also be taken
in to account when considering the rate at which the fluid fills the fracture, which appears to be slower in this model. The propagation of damage in the model occurs at a speed of approximately $0.4c_s$, where $c_s$ is the speed of sound in the solid. This rate is much greater than the speed at which the fluid is driven, and this could be taken as reason for the lower rate of fluid filling. However, some factors are not considered in this model which should be taken in to account in future work. First, the solid sample is unconfined, where in a realistic hydraulic fracturing case the material would be confined, limiting both the magnitude and rate of any fracture opening. Second, for an impermeable material, opening of the fracture is likely to create vacuum pressures at the tip which would induce flow to the tip at a greater rate. Finally, for a permeable medium pore fluid will migrate into the opened fracture under the action of pore pressure. The model presented here is more representative of an impermeable material since pore fluid is not explicitly taken in to account.

5.4.2 Hydraulic fracturing simulation with in-situ stress

The size and orientation of a fracture, and the magnitude of the pressure needed to create it, are dictated by the formation’s in situ stress field. This stress field may be defined by three principal compressive stresses, which are oriented perpendicular to each other. The magnitudes and orientations of these three principal stresses are determined by the tectonic regime in the region and by depth, pore pressure and rock properties, which determine how stress is transmitted and distributed among formations. Therefore, it is important to study the effect of the in-situ stress on the hydraulic fracturing pattern.

The simulation setup is illustrated in Fig. 5.9. The dimension is exactly same as the previous case, but with additional in-situ vertical stress ($\sigma_{yy} = 3$ MPa) applied on top and bottom rock boundary and horizontal stress ($\sigma_{xx} = 1$ MPa) applied on the right rock boundary. Another difference is that a fixed no-slip solid boundary is implemented on the left side of rock, instead of right in previous case, to prevent the
Figure 5.9: Schematic diagram of the hydraulic fracture model with in-situ stress.

The fracture propagation is shown in Fig. 5.11, where we plot the damage parameter, $D$, used in the Grady-Kipp damage model. Different from the fracture pattern without in-situ stress (Fig. 5.8), Fig. 5.11 shows that in-situ stresses do control the...
orientation and propagation direction of hydraulic fractures which is a straight line propagating from left to right. Hydraulic fractures are tensile fractures, and they open in the direction of least resistance. In this case, since the in-situ vertical stress (1MPa) is smaller than the in-situ horizontal stress (3MPa) which makes the horizontal direction the direction of least resistance in this case. Therefore, the fractures start and propagate horizontally parallel to the maximum horizontal stress when the fracturing pressure exceeds the sum of the minimum principle stress $\sigma_{Hmin}$ and the tensile strength of the rock.

### 5.5 Conclusions

In this chapter we described an SPH framework for simulation of coupled fluid and solid models, including failure of the solid phase. In applying the SPH method to both the fluid and solid phases, the computational implementation of this coupled model is essentially equivalent to an implementation describing pure fluid or solid
Figure 5.11: Fracture propagation in a coupled fluid-solid model of a hydraulic fracture with in-situ stress.

Further, the SPH formulation for solid materials is well suited to simulation of fracture initiation and propagation.

The interaction of fluid and solid was validated against two cases. The first one is to measure the hydro/geo-static stress in a model where a fluid body rests upon a solid body and the second is stress concentration measurement in a model where a circular borehole in a plate is pressurized by fluid. This stress measurement from both cases compares very well to the analytical solution and shows that interfacial forces are correctly accounted for.

The coupled framework was then applied to a proof-of-concept model of a hydraulic fracture, where a fluid is pressurized to induce a tensile fracture in a solid
material. This model demonstrates the capabilities of the coupled SPH framework, fracture initiation, propagation, opening, and fluid filling, are all observed. The model also captures complex fracture bifurcation patterns. The effect of in-situ stress on the fracture orientation was also studied and the model proves that the fracture opens and propagates in the direction of least resistance as theoretical prediction.

The models presented herein demonstrate that the coupled SPH framework has great potential in solving complex problems which include both fluid and solid phases. Especially where fracture is expected.
Chapter 6

Conclusions

This dissertation studied fluid-solid coupling using the SPH numerical method. The SPH modeling of interaction between fluids and rigid-body solids was demonstrated and validated in Chapter 2. Following validation, this framework was applied to the simulation of an impact of a solitary wave on an offshore structure. In Chapter 3, a new numerical technique was proposed for generating multi-modal and multi-directional sea waves with SPH. The method was used to simulate two real extreme wave events, with good agreement compared to the recorded data. The potential of the method for practical situations was illustrated by a simulation of the impact of an extreme wave on a tension-leg platform. Chapter 4 studied the SPH modeling for solid mechanics including elastic deformation and failure. A new implementation of boundary condition was also developed in this chapter. Finally a general numerical framework for coupling fluid and deformable solids with fracture opening was presented in Chapter 5 along with applications in hydraulic fracturing modeling.

The main contributions in the field of SPH modeling presented in this dissertation are summarized here:

- **Extreme wave modeling**: A method to generate multi-modal and multi-directional sea waves within the framework of SPH was presented. A deforming wall was used to create extreme waves similar to the ones appearing due to
a diffractive focusing of multi-directional waves. The possibility to generate extreme wave events in SPH was validated by simulating a real extreme wave with good agreement compared to the recorded data.

- **Wave impact on tension-leg platform**: An accurate knowledge of response to rogue waves is a fundamental component of offshore structure design. An SPH simulation on a real world application to study the effect of extreme wave hitting a tension-leg platform was conducted. The impact force due to the wave and the reaction force exerted by the tension legs were analyzed separately in detail. Also studied, was the horizontal and vertical displacement of the platform under the effect of a sea wave and tension legs. This analysis shows that the platform may be pulled into the water by stretched tension legs, where the extension of the tension legs also governs the rotational behavior of the platform. This simulation can serve as a blueprint for more advanced simulations in the future where the platform model may include more detail, and the initial state of the fluid phase may be more representative of conditions observed in oceans.

- **Solid deformation modeling with new implementation of boundary condition**: An elasto-plastic damage model has been employed to model the solid. Drucker-Prager plasticity model was used to predict material shear failures due to plastic deformation. For tensile failure, the Grady-Kipp damage model was used. These models are coupled together so that both shear and tensile failures can be simulated within the same scheme. A new method of applying stress and velocity boundary condition with and without dummy particles is developed and validated in this work. Results showed that SPH is able to correctly predict the evolution of fracture in brittle solids. The model has been applied to the solution of crack propagation in a “line crack” problem. The influence of initial in-situ stresses was also accounted for. Comparison of SPH results for these cases to analytical solutions showed that SPH has great
potential to model the solid deformation.

- **Fluid and deformable-solid coupling**: A general SPH numerical modeling framework was devised to model the interaction between fluids and deformable solids with highly nonlinear deformation and dynamic fracture initiation and propagation. SPH modeling of fluid and elastoplastic-solid coupling was presented. Fluids and solids were treated as a single system for the entire domain, and were computed using the same stress representation within a uniform SPH framework. Two stress-coupling approaches, namely, continuum approach and stress-boundary-condition approach, have been proposed to enforce the stress continuity at the fluid-solid interface. Both approaches allowed interactions between particles from different materials when solving the momentum equation. A corrected SPH approximation form of the density continuity equation was implemented to handle the density discontinuity of the two phases at the interface. The method has been applied to simulate rock deformation in a variety of rock mechanics test cases, including a hydrostatic problem and fluid-pressurized borehole in the presence of in-situ stresses where analytic solutions were available. The simulation of fracture initiation and propagation in the hydrofracturing case with and without in-situ stresses were also presented as the application cases. Good results demonstrated that SPH has the potential to accurately simulate the coupling of fluid and deformable solids.
Bibliography


