COPYRIGHT STATEMENT

This copy of the thesis has been supplied on condition that anyone who consults it is understood to recognise that its copyright rests with its author and that no quotation from the thesis and no information derived from it may be published without the author’s prior consent.
SMOOTHED PARTICLE HYDRODYNAMICS SIMULATIONS OF MODEL-SCALE TSUNAMI WAVES

by

RUAA HISHAM WANA

A thesis submitted to the University of Plymouth in partial fulfilment for the degree of

DOCTOR OF PHILOSOPHY

School of Engineering, Computing and Mathematics

December 2020
Acknowledgements

First of all, I would like to express my sincere appreciation and gratitude to my supervisor Dr. Jason Hughes for his supporting, understanding and patience throughout my project. The completion of this study could not have been possible without his support and guid-ances. Special thanks to Dr. David Graham for pointing me toward useful guides for my research journey.

I would like to thank all my friends and colleagues in the school of computing, electronics, and mathematics. Owing to their friendship and continuous encouragement and support, I have been able to overcome many frustrations and sadness.

My deepest gratitude, to my father (Hisham), my mother (Fatimah), sister (Rana) and brother(Omar) who support me with their love, prayers, and continuous encouragement during my PhD research journey.

My unreserved love, thanks, and appreciation must go to my gorgeous family: my husband (Omar) and my boys (Abdullah and Yousif) who have been very patient, understanding, and inspiring to me throughout this hard journey. I hope the potential success of this research will compensate some of what they have missed.

I am grateful for the COAST Laboratory (University of Plymouth), who provided me with quality data that greatly assisted my PhD research.

Finally, I would like to thank the Higher Committee for Education Development in Iraq (HCEDiraq) for their support throughout my PhD studies.

Ruaa Hisham Wana
Author’s Declaration

At no time during the registration for the degree of Doctor of Philosophy has the author been registered for any other University award without prior agreement of the Doctoral College Quality Sub-Committee.

Work submitted for this research degree at the University of Plymouth has not formed part of any other degree either at the University of Plymouth or at another establishment.

A programme of advanced study was undertaken, which included taught modules taken, other as relevant

- Math3610 fluid dynamics (Term 1 2015)
- Math3610 fluid dynamics (Term 2 2016)
- Attend the Society for Underwater Technology Evening Meeting (University of Exeter) (17th January 2018).
- Attend the UK Fluids Network Smoothed Particle Hydrodynamics special interest groups (SIG) meeting SIG Meeting 3: The Centre for Modelling and Simulation. Hosted by the University of Bristol (18th December 2018)
- Attend the 14th International SPHERIC SPH Workshop, University of Exeter, United Kingdom, (25-27 June 2019)

Publications:

- Wana R., Perez del Postigo N., Hughes J., Graham D., Raby R., and Whittaker C. Smoothed Particle Hydrodynamics (SPH) modelling of tsunami waves generated by

Presentations:

• SPH Modelling of Tsunami Waves Generated by a Fault Rupture. The British Applied Mathematics Colloquium (BAMC) conference 2018. 26th-29th March 2018, University of St Andrews, Scotland, UK

• The Effect of Boundary Conditions in Weakly Compressible Smoothed Particle Hydrodynamics. The 5th IAHR Europe Congress. 13th-15th, June 2018, University of Trento, Italy

Word count for the main body of this thesis: 39741

Signed: Ruaa Wana

Date: 19/10/2020
Smoothed Particle Hydrodynamics Simulations of Model-Scale Tsunami Waves
Ruaa Hisham Wana

Abstract

Smoothed Particle Hydrodynamics (SPH) is a meshfree, Lagrangian, particle method. It was first invented to solve astrophysical problems, but has since been developed and used to model a wide variety of fluid flows. Also, it is particularly well suited to simulating flow problems that have large deformations or contain free surfaces.

This thesis describes in detail the SPH method and its application to single phase models of flows. Fortran code has been written to implement the method. To validate and establish SPH parameters to use, the SPH method has been used in different flow simulations, in particular, lid-driven cavity flow, a still water tank and dam break flow. In these flows, we considered the effect of boundary conditions and/or initial particle spacing on the solution obtained.

The main focus of this study is to use SPH to simulate of model-scale tsunami waves generated by fault rupture, with the experiments carried out in the University of Plymouth COAST laboratory. Simulations were carried out, to predict tsunami waves, that were generated by using either a flat uplift plate or an inclined uplift plate. There was a small sloped ramp in the experimental geometry just downstream of the uplift plate and in order to obtain an accurate and stable pressure solution, required careful consideration of the boundary conditions on both the slope and at the internal corner in the flow. The SPH predictions of the free surface elevation are, in general, in good agreement with the experimental data.
Thesis Structure

In Chapter 1, the implementation of the SPH method is discussed and the particle, meshless and Lagrangian characteristics are compared to other existing methods. The basics of the method are explained through the integral representation of a function and the particle approximation. Moreover, some smoothing functions are described. The SPH method being Lagrangian, requires the governing equations to be written in the Lagrangian form. Some practical issues linked to the SPH method such as the neighbouring particles search, the equations of state and the boundary conditions are discussed.

In Chapter 2, the SPH application to the lid driven cavity flow, is described. The practical implementation of the method and of the chosen options is explained in detail. It was found that modifying the density boundary condition enhances the results obtained.

In Chapter 3, the method is applied to a still water tank and a dam break flow. These are single phase flows with a free surface. The presented results give reasonably good agreement with experimental data and are also verified by analytical and other numerical results available in the literature. The SPH method results proved to be converged and accurate.

In Chapter 4, the SPH simulations of a model-scale earthquake fault rupture, for the cases of flat uplift plate, which have been carried out at University of Plymouth’s COAST laboratory, are presented and discussed. In particular, the boundary conditions needed to obtain an accurate and stable solution are described. The SPH predicted surface elevations are then compared with the experimental measurements, which are generally in good agreement.

In Chapter 5, the model-scale of the tsunami waves generated by an earthquake fault rapture, by an inclined uplift plate, are simulated using the SPH method. The free surface height profiles and wave speeds are in reasonably good agreement between the SPH simulations, experiments, and theory. Also, a comparison between the SPH models of the flat uplift plate and the inclined slope uplift is made, in order to illustrate the main differences
between these two models.

In Chapter 6, SPH simulations of experimental models of tsunami waves generated by a landslide are described. The landslide block has been created on top of the fixed inclined plate and released down to create a tsunami wave. Two different height of landslide (0.018 m and 0.036 m) have been simulated and compared with each other. The results show that the SPH model gives generally reasonable predictions of the free surface position. Chapter 7 discussed the main conclusion of this study and some suggestions for future work.
## Contents

**Acknowledgements** iii  
**Author’s Declaration** iv  
**Abstract** vi  
**Thesis Structure** vii  

### 1 Introduction 1  

1.1 Grid-Based Method ...................................... 5  
1.1.1 The Lagrangian description .......................... 5  
1.1.2 The Eulerian description ............................ 6  
1.2 Meshfree Particle Methods (MPM) ....................... 6  
1.3 SPH Theory and Implementation ....................... 8  
1.4 SPH approximations .................................... 8  
1.4.1 Functions representation in SPH .................. 8  
1.4.2 Representation of a function derivative in SPH .... 10  
1.5 Interpolating Kernel Function ......................... 12  
1.6 Smoothing Length ...................................... 14  
1.7 Governing Equations in SPH ......................... 14  
1.7.1 Conservation of mass ............................... 14  
1.7.2 Conservation of momentum ......................... 16  
1.8 Viscosity ............................................. 19
1.9 Pressure Formulation ........................................... 20
1.10 Density Evaluation ............................................ 21
1.11 Density Reinitialization ....................................... 22
1.12 Tensile Correction .............................................. 23
1.13 Initial Conditions ............................................... 24
1.14 Boundary Conditions .......................................... 25
1.15 Moving the Particles ........................................... 28
1.16 Time Step ....................................................... 29
1.17 Search for Neighbouring Particles (Linked List Method) .... 31
1.18 Conclusions .................................................... 32

2 Simulation of Lid-Driven Cavity Flow using SPH .......... 33
  2.1 Introduction .................................................. 33
  2.2 Simulation Conditions ........................................ 33
  2.3 Numerical Results ............................................ 35
    2.3.1 Simulation of the lid-driven cavity flow at Re = 100 ...... 37
    2.3.2 Simulation of the lid-driven cavity flow at Re = 400 ...... 38
    2.3.3 Simulation of the lid-driven cavity flow at Re = 1000 ...... 39
  2.4 Conclusions .................................................. 57

3 Smoothed Particle Hydrodynamics
  (SPH) Model for Free Surface Flows ....................... 58
  3.1 Introduction .................................................. 58
  3.2 Still Water Tank Simulations (Hydrostatic Tank) ........... 58
  3.3 Dam Break Flow Simulations ................................ 66
3.3.1 Simulation of dam break flow using fixed boundary particles with uniform spacing ............................................. 66

3.3.2 Simulation of dam break flow using ghost boundary particles with uniform spacing ........................................... 68

3.3.3 Simulation of dam break flow using fixed boundary particles with staggered spacing ............................................. 70

3.3.4 Simulation of dam break flow using fixed ghost boundary particles with uniform spacing ................................. 77

3.4 Conclusions ...................................................................................................................................................... 80

4 Tsunami Wave Simulated by a Fault Rupture using a Horizontal Uplift Plate 81

4.1 Introduction ..................................................................................................................................................... 81

4.2 Simulations of a Tsunami Wave Generated by a Flat Uplift Plate ................................................................. 82

4.2.1 Experimental model ..................................................................................................................................... 82

4.2.2 SPH model .................................................................................................................................................. 82

4.3 Including the Small Sloped Ramp in the Simulations of a Tsunami Wave Generated by a Flat Uplift Plate ................................................................................................................................. 96

4.4 The Effect of Plate Uplift Time and Water Depth on the Model-Scale Tsunami Waves ........................................ 103

4.5 Conclusions ..................................................................................................................................................... 108

5 Tsunami Wave Generated by Fault Rupture Using an Inclined Uplift Plate (IU) 109

5.1 Introduction ..................................................................................................................................................... 109

5.2 Experimental Model with an Inclined Uplift Plate (IU) ....................................................................................... 109

5.3 SPH Simulation of Tsunami Waves Generated by an Inclined Uplift Plate (IU) .................................................. 110
List of Figures

1.1 Lagrangian description. ................................................. 6
1.2 Eulerian description. .................................................. 7
1.3 Neighbouring particles and the kernel function on the domain. .... 10
1.4 Cubic spline kernel function and its derivative. ....................... 13
1.5 Quintic (Wendland) kernel function and its derivative. ............. 13
1.6 Pressure solution ...................................................... 22
1.7 Fluid particles position effecting without using tensile correction term. 24
1.8 Initial particle position .................................................. 25
1.9 Boundary conditions. ................................................... 27
1.10 Fixed Ghost Particles. .................................................. 27
1.11 XSPH variant. .......................................................... 29
1.12 The neighbours of a host particle and the linked list cells. .......... 32
2.1 Initial particles distribution ............................................ 34
2.2 Location of mid horizontal line $x = 0.5$ and mid vertical line $y = 0.5$ in the lid driven cavity flow geometry. ......................... 35
2.3 The effect of adding and not adding a background pressure for a lid driven cavity flow of $Re = 1000$. ........................................ 36
2.4 $u$ and $v$ velocity profile respectively, with $\rho = \rho_0$ as the dummy particle boundary condition for $Re = 1000$. ......................... 37
2.5 $u$ and $v$ velocity profile respectively, with $\frac{\partial \rho}{\partial n} = 0$ as the dummy particle boundary condition for $Re = 1000$. ......................... 37
2.6 $u$-velocity profile against $y$, on the mid horizontal line ($x = 0.5$) with Re = 100 and various particle resolutions, summation density method, uniform spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme. .................................................. 44

2.7 $v$-velocity profile against $x$, on the mid vertical line ($y = 0.5$) with Re = 100 and various particle resolutions, summation density method, uniform spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme. .................................................. 44

2.8 $u$-velocity profile against $y$, on the mid horizontal line ($x = 0.5$) with Re = 100 and various particle resolutions, evolved density method, uniform spacing and cubic spline function with tensile correction term and mixed Euler-Verlet time step scheme. .................................................. 45

2.9 $v$-velocity profile against $x$, on the mid vertical line ($y = 0.5$) with Re = 100 and various particle resolution, evolved density method, uniform spacing and cubic spline function with tensile correction term and mixed Euler-Verlet time step scheme. .................................................. 45

2.10 $u$-velocity profile against $y$, on the mid horizontal line ($x = 0.5$) with Re = 100 and various particle resolutions, summation density method, staggered spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme. .................................................. 46

2.11 $v$-velocity profile against $x$, on the mid vertical line ($y = 0.5$) with Re = 100 and various particle resolutions, summation density method, staggered spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme. .................................................. 46

2.12 $u$-velocity profile against $y$, on the mid horizontal line ($x = 0.5$) with Re = 400 and various particle resolutions, evolved density method, uniform spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme. .................................................. 47
2.13 \(v\)-velocity profile against \(x\), on the mid vertical line \((y = 0.5)\) with \(Re = 400\) and various particle resolutions, evolved density method, uniform spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme. ......................... 47

2.14 \(u\)-velocity profile against \(y\), on the mid horizontal line \((x = 0.5)\) with \(Re = 400\) and various particle resolutions, summation density method, staggered spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme. ......................... 48

2.15 \(v\)-velocity profile against \(x\), on the mid vertical line \((y = 0.5)\) with \(Re = 400\) and various particle resolutions, summation density method, staggered spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme. ......................... 48

2.16 \(u\)-velocity profile against \(y\), on the mid horizontal line \((x = 0.5)\) with \(Re = 1000\) and various particle resolutions, evolved density method, uniform spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme. ......................... 49

2.17 \(v\)-velocity profile against \(x\), on the mid vertical line \((y = 0.5)\) with \(Re = 1000\) and various particle resolutions, evolved density method, uniform spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme. ......................... 49

2.18 \(u\)-velocity profile against \(y\), on the mid horizontal line \((x = 0.5)\) with \(Re = 1000\) and various particle resolutions, summation density method, staggered spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme. ......................... 50

2.19 \(v\)-velocity profile against \(x\), on the mid vertical line \((y = 0.5)\) with \(Re = 1000\) and various particle resolutions, summation density method, staggered spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme. ......................... 50
2.20 Streamline using evolved density method and uniform spacing (130×130) for Re = 100. 51
2.21 Streamline using summation density method and staggered spacing (130×130) for Re = 100. 51
2.22 Streamline of Re = 100, (130 × 130) obtained by Ghia et al. (1982). 51
2.23 Streamline using evolved density method and uniform spacing (130×130) for Re = 400. 52
2.24 Streamline using summation density method and staggered spacing (130×130) for Re = 400. 52
2.25 Streamline of Re = 400, (130 × 130) obtained by Ghia et al. (1982). 52
2.26 Streamline using evolved density method and uniform spacing (130×130) for Re = 1000. 53
2.27 Streamline using summation density method and staggered spacing (130×130) for Re = 1000. 53
2.28 Streamline of Re = 1000, (130 × 130) obtained by Ghia et al. (1982). 53
2.29 Vorticity using evolved density method and uniform spacing for Re = 100. 54
2.30 Vorticity using summation density method and staggered spacing for Re = 100. 54
2.31 Vorticity of Re = 100 obtained by Ghia et al. (1982). 54
2.32 Vorticity counter using evolve density method and uniform spacing for Re = 400. 55
2.33 Vorticity using summation density method and staggered spacing for Re = 400. 55
2.34 Vorticity of Re = 400 obtained by Ghia et al. (1982). 55
2.35 Vorticity counter using evolve density method and uniform spacing for Re = 1000. 56
LIST OF FIGURES

2.36 Vorticity using summation density method and staggered spacing for $Re = 1000$. ........................................ 56

2.37 Vorticity of $Re = 1000$ obtained by Ghia et al. (1982). .............. 56

3.1 Initial geometry of the test case of a still water in a tank. ............... 59

3.2 Pressure solution with the use of $(P = P_w)$ and $(\rho = \rho_0)$, using Wendland kernel function and staggered spacing. ....................... 61

3.3 Pressure solution with the use of $(P = P_w + \rho gd)$ and $(\rho = \rho_w)$, using Wendland kernel function and staggered spacing. ....................... 62

3.4 (a) Pressure solution with the use of $(P = P_w + \rho gd)$ and $(\rho = \rho_w)$, using cubic spline kernel function and staggered spacing. .............. 62

3.5 Pressure solution with the use of $(P = P_w + \rho gd)$ and $(\rho = \rho_w)$, using Wendland kernel function and uniform spacing with fixed boundary particles. ........................................ 62

3.6 Pressure solution with the use of $(P = P_w + \rho gd)$ and $(\rho = \rho_w)$, using Wendland kernel function and uniform spacing with fixed ghost boundary particles. ........................................ 63

3.7 Snapshots at different times shows the pressure solution, using staggered spacing and Wendland kernel function. ......................... 63

3.8 Snapshots at different times shows the pressure solution, using uniform spacing with fixed boundary particles and Wendland kernel function. ........ 64

3.9 Snapshots at different times shows the pressure solution, using uniform spacing with fixed ghost boundary particles and Wendland kernel function. ......................... 65

3.10 Initial geometry of the dam break. ........................................ 66

3.11 Comparison of the leading edge location using uniform spacing particle with two different resolutions and two different time step schemes. ........ 68

3.12 Comparison of the water column height using uniform spacing particle with two different resolutions and two different time step schemes. ........ 68
3.13 Dam break solution at various normalized times.

3.14 \( W = H/2, D = H \) dam break solution for mixed Euler-Verlet time step using uniform spacing at \( t\sqrt{2g/W} = 9.124 \), using (a) 9800 fluid particles (c) 20000 fluid particles and at \( t\sqrt{2g/W} = 11.25 \), using (b) 9800 fluid particles (d) 20000 fluid particles.

3.15 \( W = H/2, D = H \) dam break solution for Verlet time step using uniform spacing at \( t\sqrt{2g/W} = 9.124 \), using (a) 9800 fluid particles (c) 20000 fluid particles and at \( t\sqrt{2g/W} = 11.25 \), using (b) 9800 fluid particles (d) 20000 fluid particles.

3.16 Comparison of the leading edge location using uniform particle spacing with ghost boundary particles and mixed Euler-Verlet time step.

3.17 Comparison of the water column height using uniform particle spacing with ghost boundary particles and mixed Euler-Verlet time step.

3.18 \( W = H/2, D = H \) dam break solution for mixed Euler-Verlet time step using uniform spacing and ghost boundary particles at \( t\sqrt{2g/W} = 9.124 \), using (a) 9800 fluid particles (c) 20000 fluid particles and at \( t\sqrt{2g/W} = 11.25 \), using (b) 9800 fluid particles (d) 20000 fluid particles.

3.19 Comparison of the leading edge location using staggered spacing with two different resolutions and two different time step schemes.

3.20 Comparison of the water column height using staggered spacing particle with two different resolutions and two different time step schemes.

3.21 \( W = H/2, D = H \) dam break solution for mixed Euler-Verlet time step using staggered spacing at \( t\sqrt{2g/W} = 9.124 \), using (a) 6400 fluid particles (c) 40000 fluid particles and at \( t\sqrt{2g/W} = 11.25 \), using (b) 6400 fluid particles (d) 40000 fluid particles.
3.22 W = H/2, D = H dam break solution for Verlet time step using staggered spacing at $t \sqrt{2g/H} = 9.124$, using (a) 6400 fluid particles (c) 40000 fluid particles and at $t \sqrt{2g/H} = 11.25$, using (b) 6400 fluid particles (d) 40000 fluid particles. .................................................. 76

3.23 Comparison of the leading edge location using uniform spacing with fixed ghost boundary particles and 20000 fluid particles. ......................... 78

3.24 Comparison of the water column height using uniform spacing particle with fixed ghost boundary particles and 20000 fluid particles. .......... 78

3.25 W = H/2, D = H dam break solution for mixed Euler-Verlet time step, using uniform spacing and fixed ghost boundary particles at $t \sqrt{2g/H} = 9.124$ and $t \sqrt{2g/H} = 11.25$, using 20000 fluid particles. ...................... 79

4.1 Experimental wave tank ................................................. 82

4.2 Plate and wave gauges locations. ..................................... 83

4.3 Ghost boundary .......................................................... 85

4.4 Schematic representation of wave tank. ............................ 85

4.5 Experiment plate displacement. ........................................ 86

4.6 Effect of tank length on WG1 surface elevation using SPH method. . 86

4.7 Effect of tank length on WG8 surface elevation using SPH method. . 87

4.8 Comparison of WG1 surface elevation using SPH method with two different particle resolution. .................................................. 87

4.9 SPH plate displacement and elevation at WG1. ...................... 87

4.10 Experiment plate displacement and elevation at WG1. ............... 88

4.11 SPH predictions of surface height at the eight different wave gauges. 89

4.12 Experimental measurements of surface height at the eight different wave gauges. ......................................................... 89
4.13  Comparison between SPH predictions and experimental measurements of surface height at WG1. ................................................. 90
4.14  Comparison between SPH predictions and experimental measurements of surface height at WG2. ................................................. 90
4.15  Comparison between SPH predictions and experimental measurements of surface height at WG3. ................................................. 90
4.16  Comparison between SPH predictions and experimental measurements of surface height at WG4. ................................................. 91
4.17  Comparison between SPH predictions and experimental measurements of surface height at WG5. ................................................. 91
4.18  Comparison between SPH predictions and experimental measurements of surface height at WG6. ................................................. 91
4.19  Comparison between SPH predictions and experimental measurements of surface height at WG7. ................................................. 92
4.20  Comparison between SPH predictions and experimental measurements of surface height at WG8. ................................................. 92
4.21  SPH pressure solution near the plate rupture the times at which it is mov-
ing upwards. ................................................................. 93
4.22  SPH vertical velocity solution when the plate is half way up, at \( t = 1.19 \) s. 95
4.23  SPH pressure solution at various times after the plate has moved upwards. 95
4.24  Fixed ghost boundary particles on the slopes in the wave tank. ............ 98
4.25  Schematic representation of wave tank, including a sloped ramp. .......... 99
4.26  Fixed ghost particles creation at the corner of the moving flat uplift plate and the fixed slope. ....................................................... 99
4.27  Comparison between SPH predictions of surface height at WG1 with and without the small sloped ramp. ................................. 100
4.28 Comparison between SPH predictions of surface height at WG3 with and without the small sloped ramp. ........................................ 100
4.29 Comparison between SPH predictions of surface height at WG5 with and without the small sloped ramp. ........................................ 100
4.30 Comparison between SPH predictions of surface height at WG7 with and without the small sloped ramp. ........................................ 101
4.31 SPH pressure solution near the plate rupture the times at which it is moving upwards. ........................................ 101
4.32 SPH vertical velocity solution when the plate is half way up, at \( t = 1.19 \) s. ........................................ 102
4.33 SPH pressure solution at various times after the plate has moved upwards. ........................................ 102
4.34 Plot of the uplift displacement profile, for various uplift times. ........................................ 104
4.35 Surface elevation at WG1 for various plate uplift times. ........................................ 105
4.36 Surface elevation at WG2 for various plate uplift times. ........................................ 105
4.37 Surface elevation at WG4 for various plate uplift times. ........................................ 105
4.38 Surface elevation at WG8 for various plate uplift times. ........................................ 106
4.39 Difference in surface elevation at WG1 for various water depths. ........................................ 106
4.40 Difference in surface elevation at WG2 for various water depths. ........................................ 106
4.41 Difference in surface elevation at WG4 for various water depths. ........................................ 107
4.42 Difference in surface elevation at WG8 for various water depths. ........................................ 107

5.1 Inclined uplift plate used in the experiment. ........................................ 110
5.2 Schematic representation of wave tank, including a sloped ramp. ........................................ 111
5.3 Experiment plate displacement for inclined uplift. ........................................ 111
5.4 SPH plate displacement for inclined slope uplift and elevation at WG1. ........................................ 112
5.5 Experiment plate displacement for inclined slope uplift and elevation at WG1. ........................................ 112
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.6</td>
<td>SPH predictions of surface height at the sixteen different wave gauges.</td>
<td>114</td>
</tr>
<tr>
<td>5.7</td>
<td>Experimental measurements of surface height at the sixteen different wave gauges.</td>
<td>114</td>
</tr>
<tr>
<td>5.8</td>
<td>Comparison between SPH predictions and experimental measurements of surface height at WG1.</td>
<td>115</td>
</tr>
<tr>
<td>5.9</td>
<td>Comparison between SPH predictions and experimental measurements of surface height at WG2.</td>
<td>115</td>
</tr>
<tr>
<td>5.10</td>
<td>Comparison between SPH predictions and experimental measurements of surface height at WG4.</td>
<td>115</td>
</tr>
<tr>
<td>5.11</td>
<td>Comparison between SPH predictions and experimental measurements of surface height at WG6.</td>
<td>116</td>
</tr>
<tr>
<td>5.12</td>
<td>Comparison between SPH predictions and experimental measurements of surface height at WG8.</td>
<td>116</td>
</tr>
<tr>
<td>5.13</td>
<td>Comparison between SPH predictions and experimental measurements of surface height at WG10.</td>
<td>116</td>
</tr>
<tr>
<td>5.14</td>
<td>Comparison between SPH predictions and experimental measurements of surface height at WG12.</td>
<td>117</td>
</tr>
<tr>
<td>5.15</td>
<td>Comparison between SPH predictions and experimental measurements of surface height at WG14.</td>
<td>117</td>
</tr>
<tr>
<td>5.16</td>
<td>Comparison between SPH predictions and experimental measurements of surface height at WG16.</td>
<td>117</td>
</tr>
<tr>
<td>5.17</td>
<td>SPH pressure solution near the inclined slope uplift over the times at which it is moving upwards.</td>
<td>119</td>
</tr>
<tr>
<td>5.18</td>
<td>SPH vertical velocity solution when the inclined uplift is half way up, at $t = 1.17$ s.</td>
<td>119</td>
</tr>
<tr>
<td>5.19</td>
<td>SPH pressure solution at various times after the inclined uplift has moved upwards.</td>
<td>120</td>
</tr>
</tbody>
</table>
5.20 Schematic representation of the SPH wave tank for the inclined uplift model. ................................. 121

5.21 Comparison between SPH predictions of surface height for the flat plate and the inclined uplift at WG1. ................................................................. 122

5.22 Comparison between SPH predictions of surface height for the flat plate and the inclined uplift at WG2. ................................................................. 122

5.23 Comparison between SPH predictions of surface height for the flat plate and the inclined uplift at WG4. ................................................................. 122

5.24 Comparison between SPH predictions of surface height for the flat plate and the inclined uplift at WG8. ................................................................. 123

6.1 Schematic diagram of the experimental test rig showing the fault rupture (uplift plate) and landslide model. ................................. 125

6.2 Schematic diagram of the SPH model showing the fault rupture (inclined plate) and landslide model. ................................................................. 126

6.3 Moving landslide (fixed ghost particles creation) at the fixed inclined plate. ................................................................. 127

6.4 SPH predictions of surface height at the three different wave gauges for the 0.018m landslide height. ................................................................. 128

6.5 SPH predictions of surface height at the three different wave gauges for the 0.036m landslide height. ................................................................. 128

6.6 Experimental measurements of surface height at the three different wave gauges for the 0.018m landslide height. ................................................................. 128

6.7 Comparison between SPH predictions of surface height at WG1 for the 0.018m and 0.036m landslides. ................................................................. 129

6.8 Comparison between SPH predictions of surface height at WG2 for the 0.018m and 0.036m landslides. ................................................................. 130
6.9 Comparison between SPH predictions of surface height at WG3 for the 0.018m and 0.036m landslides. 130
6.10 SPH pressure solution at various times after the 0.018 m landslide has moved downwards. 131
6.11 SPH pressure solution at various times after the 0.018 m landslide has moved downwards. 132
6.12 SPH pressure solution at various times after the 0.036 m landslide has moved downwards. 133
6.13 SPH pressure solution at various times after the 0.036 m landslide has moved downwards. 134
6.14 SPH pressure solution after the 0.036 m landslide has moved downwards. 134
List of Tables

2.1 Reynolds number and viscosity values ........................................... 34
2.2 Initial particles distribution for lid driven cavity flow simulations .... 34
2.3 Comparison of parameters used in the lid-driven cavity flow simulations. .... 41
2.4 Comparison of Re = 100 simulations between the evolved density method with
uniform spacing of present work, with Ghia et al. (1982), Botella & Peyret (1998)
and Sahin & Owens (2003) results for the minimum velocity of the moving fluid
particles, using mixed Euler-Verlet time step and cubic spline kernel with tensile
correction term, on the $x = 0.5$ and $y = 0.5$ lines. ............................ 42
2.5 Comparison of Re = 100 simulations between the summation density method of
present work, using uniform spacing, with Ghia et al. (1982), Botella & Peyret
(1998) and Sahin & Owens (2003) results for the minimum velocity of the mov-
ing fluid particles, using mixed Euler-Verlet time step and Wendland spline ker-
nel function without tensile correction term, on the $x = 0.5$ and $y = 0.5$ lines.       ............... 42
2.6 Comparison of Re = 100 simulations between the summation density method,
using staggered spacing of present work, with Ghia et al. (1982), Botella &
Peyret (1998) and Sahin & Owens (2003) results for the minimum velocity of the
moving fluid particles, using mixed Euler-Verlet time step and Wendland kernel
function without tensile correction term, on the $x = 0.5$ and $y = 0.5$ lines. 42
2.7 Comparison of Re = 400 simulations between the evolved density method, using
uniform spacing of present work, with Ghia et al. (1982) and Sahin & Owens
(2003) results for the minimum velocity of the moving fluid particles, using
mixed Euler-Verlet time step and Wendland kernel function without tensile cor-
rection term, on the $x = 0.5$ and $y = 0.5$ lines. ................................. 43
<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.8</td>
<td>Comparison of Re = 400 simulations between the summation density method, using staggered spacing of present work, with Ghia et al. (1982) and Sahin &amp; Owens (2003) results for the minimum velocity of the moving fluid particles, using mixed Euler-Verlet time step and Wendland kernel function without tensile correction term, on the $x = 0.5$ and $y = 0.5$ lines.</td>
<td>43</td>
</tr>
<tr>
<td>2.9</td>
<td>Comparison of Re = 1000 simulations between the evolved density method, using uniform spacing of present work, with Ghia et al. (1982), Botella &amp; Peyret (1998) and Sahin &amp; Owens (2003) results for the minimum velocity of the moving fluid particles, using mixed Euler-Verlet time step and Wendland kernel function without tensile correction term, on the $x = 0.5$ and $y = 0.5$ lines.</td>
<td>43</td>
</tr>
<tr>
<td>2.10</td>
<td>Comparison of Re = 1000 simulations between the summation density method, using staggered spacing of present work, with Ghia et al. (1982), Botella &amp; Peyret (1998) and Sahin &amp; Owens (2003) results for the minimum velocity of the moving fluid particles, using mixed Euler-Verlet time step and Wendland kernel function without tensile correction term, on the $x = 0.5$ and $y = 0.5$ lines.</td>
<td>44</td>
</tr>
<tr>
<td>3.1</td>
<td>Initial particle distribution for dam break simulations</td>
<td>60</td>
</tr>
<tr>
<td>3.2</td>
<td>Initial particle distribution</td>
<td>66</td>
</tr>
<tr>
<td>4.1</td>
<td>Tsunami speed between WG4 and WG8</td>
<td>107</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

Computational fluid dynamics (CFD) has been used widely over the past few decades, to predict the fluid flow in problems by means of mathematical modelling, numerical methods (discretization and solution techniques) and software tools. The strategy is to replace the continuous problem domain by a discrete domain.

Eulerian and Lagrangian approaches are the main classifications of CFD methods which are used to model the flow of fluid. The Lagrangian specification of the flow field is a way of considering the fluid motion, where the observer follows an individual fluid particle as it moves through space and time. The Eulerian specification of the flow field is a way of considering the fluid motion, that focuses on specific locations in the space through which the fluid flows as time passes. In CFD, Lagrangian particle tracking is a numerical technique for simulated tracking of particle paths (Lagrangian) within an Eulerian phase. It is commonly referred to as Discrete Particle Simulation (DPS). Since the grid is required only within the problem domain, no additional grid beyond the problem domain is required i.e. the Lagrangian approach does not necessarily require a mesh in CFD (Liu & Liu 2003).

The Eulerian approach is based on grids and includes the finite difference method (FDM) (Morton & Mayers 2005) and the finite volume method (FVM) (Toro & Solvers 1999). These methods are known as fixed grid-based methods. In contrast, the Lagrangian approach is based on meshes attached to the modelled region and is presented in the finite element method (FEM) (Sharma et al. 1985). These methods have been applied to model the fluid flow problems by representing it as a set of partial differential equations that model the physical phenomena and then solve it. The main idea of the grid-based method
is that the continuous domain is divided into smaller sub-domains, that is, grids in FDM, cells in FVM, and elements in FEM. All of these grid-based numerical methods are used in numerical simulations because of their flexibility and accuracy when applied in the field of computational fluid dynamics and have had great success (Becker 1992).

Although the grid-based methods succeeded, these methods suffer from several barriers. The use of the grid in the calculations has limitations in the application of many complex problems, due to the problems of creating a grid for a complex geometry. This means an expensive task in time and mathematical calculations (Vitanza 2014). However, for free surface problems, Eulerian methods have difficulties in time history (at a specific point located on the material), moving boundaries and interfaces and also irregular geometries, and so accurate results may not be obtained. Moreover, because the grid remained unchanged, an irregular geometry is difficult to deal with (Crespo 2008). So, the attention has been focused on Lagrangian meshfree numerical methods. As opposed to the Eulerian method, the time history and moving boundaries and interfaces are easy to compute, and an irregular geometry is easy to handle (Crespo 2008). The main idea of the Lagrangian meshfree numerical procedure is that the physical problems are solved by using integral or partial differential equations with a certain boundary condition, by representing the fluid by a set of particles or nodes which are just interpolation points for which the properties of the fluid can be calculated (Belytschko et al. (1996) and Liu et al. (2005)).

Smoothed Particle Hydrodynamics is one of the main Lagrangian meshfree particles-based numerical methods. It is a powerful method for solving CFD problems governed by the Navier-Stokes equations. Gingold & Monaghan (1977), Lucy (1977) and Monaghan (2005) introduced this method for modelling complex and highly non-symmetrical fluid-dynamical phenomena in astrophysics. This method was then applied on problems of continuum solid and fluid mechanics. The basic idea of the SPH method is to use a set of discrete particles to represent a fluid, such that every particle has its own properties and arranges in such a way to simulate the continuous domain.

There are some limitations in using the SPH method, such as the choice of boundary condition may affect particle penetration at the boundaries which should be avoided. Also, because of the time step, which needs to be very small ($10^{-4}$) and is dependent on the
sound speed in the fluid, the method is computationally slow. Moreover, the simple SPH interpolation method that is used for fluid particles leads to particle disorder. Bonet & Lok (1999) suggested the pressure gradient correction in order to solve this problem and showed that including this correction produced more accurate solution.

The most common SPH methods used to model fluid flows are Incompressible Smoothed Particle Hydrodynamics (ISPH), in which a Poisson equation is used to calculate the pressure and Weakly Compressible Smoothed Particle Hydrodynamics (WCSPH) which uses a stiff equation of state to calculate the pressure directly from the density. Using the stiff equation of state to calculate the pressure does not produce a solution which satisfies the incompressibility condition of the fluid, as there will be a variation in the density of the particles. Hence, the fluid is modelled as 'quasi-incompressible'. the fluctuations in the density are typically around one per cent, due to the numerical speed of sound used, which is normally taken as ten times the maximum fluid velocity. This is one of the advantages of the WCSPH method, that it computes the pressure from the particle density, rather than solving a linear system of equations, which is computationally expensive.

SPH has many applications especially in recent years when it was developed to model the continuum physics. It was first proved to be applicable in Astrophysics (Benz 1989) and used to model a collapse of galaxies (Monaghan & Lattanzio 1991) and the universe evolution (Monaghan 1990). It has also been applicable in Fluid Dynamics and especially in free-surface flows (Monaghan 1994), Monaghan & Kos (1999) and Monaghan (2000)), gravity currents (Monaghan 1996 and Monaghan et al. (1999)) and multi-phase flows (Monaghan & Kocharyan 1995), in addition to Solid Mechanics like impact problems (Libersky & Petschek (1991), Libersky et al. (1993) and Johnson et al. (1996)).

From the SPH literature, there are many numerical studies that have discussed the lid-driven cavity flow as a simple application for the SPH method, for both ISPH and WC-SPH, such as Hughes & Graham (2007), Szewc et al. (2012), Lee et al. (2008) and Sun et al. (2015). Many authors have used different methods to model the lid-driven cavity flow, for instance Sahin & Owens (2003) who used the finite volume method (FVM) to model the lid-driven cavity problem for both steady and unsteady flows at Reynolds numbers up to 10000. The results are smooth and in excellent agreement with benchmark
results in the literature. However, Botella & Peyret (1998) used the Chebyshev collocation method to give extensive numerical results in the case of a Reynolds number of 1000, which are in agreement with Ghia et al. (1982), who used the finite difference method (FDM).

However, the dam break problem is widely used as a test case for free surface flows, starting from Monaghan & Kos (1999) who used the traditional WCSPH method, with a stiff equation of state, and Cummins & Rudman (1999) who have brought the pressure correction from grid methods. Also, Lee et al. (2008) have applied the SPH method, for both ISPH and WCSPH, to both of the lid-driven cavity flow and the dam break flow. Moreover, Hughes & Graham (2010) compared both the ISPH and WCSPH methods for dam break flow. Colagrossi & Landrini (2003) have proposed a numerical density re-initialization technique on the pressure field to model flow fields with free-surfaces with sharp interfaces. Also, Greaves (2006) presented the lid driven cavity at a Reynolds number of 1000 and the dam break flow, with the adapting hierarchicol grids method. Results are in excellent agreement with experimental and other numerical data of Ghia et al. (1982). Later, Marrone et al. (2011) model the dam break flow with numerical diffusive terms to analyse violent water flows with fixed ghost boundary particles.

Furthermore, there are many kinds of research in the literature which discuss in detail both the experimental and the numerical modelling of tsunami waves. Tsunami waves, which have extremely long wavelengths, are disastrous sea waves that can damage the geographic area. A number of countries have been severely damaged by tsunamis, for example Indonesia, Japan, Thailand, and Chile. There are many different ways in which a tsunami wave can be generated, such as an earthquake, volcano, eruption, or landslide. This work focuses on tsunami waves generated by an earthquake fault rupture or a landslide. Most of the previous studies, for both physical and numerical modelling, have investigated seven governing parameters: still water tank depth and length, slide velocity, slide thickness, total slide volume, total slide density and slide angle, such as Hammack Jr (1972), Hammack (1973) and Heller & Hager (2010). Heller & Hager (2011) and Heller & Spinneken (2015) have presented many experimental studies that discuss the tsunami waves generated by a subaerial landslide. Also, the physical moving landslide (solid
1.1. GRID-BASED METHOD

block) has a wide representation in the tsunami literature with various geometries, as presented in Heller & Spinneken (2013) and Sue (2007).

The SPH method has been widely used to model complex fluid flows and flows with fluid-structure interaction (Ye et al. (2019)). Trimulyono & Hashimoto (2019) discussed an experimental validation of smoothed particle hydrodynamics (SPH) on the generation and propagation of water waves. Cunningham et al. (2014) used WCSPH to improve the resilience of shore-based structures in tsunami events, by investigating the effects of more complex building geometry, orientation, and proximity to adjacent structures.

The presented work in this thesis, correctly simulated the model-scale of tsunami waves generated by an earthquake fault rupture or a landslide. The experiments were carried out in the COAST laboratory at the University of Plymouth, and are described by Perez del Postigo Prieto et al. (2018), who developed a unique set-up to produce a dual-source tsunami generation mechanism of a two-dimensional underwater fault rupture followed by a submarine landslide. Perez del Postigo Prieto et al. (2019) produced quality data for developing a parametrisation of the initial conditions for tsunami generation processes which are generated by a dual-source.

1.1 Grid-Based Method

The grid-based methods require the use of a mesh. Methods like the finite element method (FEM) or the finite volume method (FVM) are grid-based methods. There are two fundamental descriptions to describe the physical governing equations: (Liu & Liu 2003).

1.1.1 The Lagrangian description

The Lagrangian description is based on material description, where the grid or mesh is fixed to the material in the simulation, which is a technique that can be used in the finite element method (FEM). This means that when the object studied deforms itself, then the grid or mesh is also deformed. The track of the material is known at grid points. At each point, the time-history data (such as position, mass, momentum, energy, etc.) are computed.
1.2. MESHFREE PARTICLE METHODS (MPM)

Advantages and disadvantages of Lagrangian methods

- Ability to track certain particles
- Easily defines a boundary condition, as it is fairly straightforward to keep track of any geometrical changes, and modify the grid or mesh as required.
- No additional grid beyond the problem domain is required
- Difficult to apply for cases with extremely distorted mesh

1.1.2 The Eulerian description

The Eulerian description is based on a fixed grid in space. This means that the grid remains in the same position during the whole simulation, independently from the material position and it is a technique that can be used in the finite volume method (FVM), where the grid or mesh is not fixed to the material during the simulation. So, at a point attached to the material, the time-history data (such as position, mass, momentum, energy, etc.) are difficult to obtain (Crespo 2008).

1.2 Meshfree Particle Methods (MPM)

The traditional grid-based methods, such as the finite element method (FEM) and the finite difference method (FDM) use grids, volumes or cells according to the method used. Generally, these grids, volumes or cells are named meshes. The meshes are necessary
for FEM and FDM to solve the differential equations that describe the physical problem. Meshfree methods facilitate the simulation of problems that require the ability to treat large deformations, advanced materials, complex geometry, non-linear material behaviour, discontinuities and singularities. So, meshfree particle methods (MPM) treat the system as a set of particles, which represents a physical object or a parcel of the domain. For Computational Fluid Dynamics (CFD) problems, variables such as mass, momentum, energy, position, etc. are computed at each particle. There are other meshfree methods, which use meshless nodes or particles without the mass or volume, such as radial basis function (RBF) methods (Zhang et al. 2000).

A MPM is a Lagrangian method. In fact, the particles carry properties of the flow (such as position, velocity, density, and pressure) at every time step. Thus, we are interested in the value of a variable for a given particle independently from the position of the particle and not the value of a variable at a fixed position in space (Liu & Liu 2003).

In general, for Integral equations or PDEs, the meshfree methods have the ability of defining the boundary conditions using a set of particles. Using meshfree methods it is easier to model:

- Large deformations
- Materials interfaces
- Free surfaces
- Shockwaves
• Explosions

• High velocity impacts

1.3 SPH Theory and Implementation

Smoothed Particles Hydrodynamics was originally developed to simulate astrophysical problems by Lucy (1977) and Gingold & Monaghan (1977). Then Monaghan (1992) modified this meshfree particle method and applied it to continuum solid and fluid mechanics. The main idea of this method is using discrete particles, such that every particle has its own physical properties (position, velocity, density, and pressure) and then evolves according to the relevant conservation laws.

1.4 SPH approximations

The Navier-Stokes equations together with the continuity equation are:

\[
\frac{Du}{Dt} = -\frac{1}{\rho} \nabla P + \nu \nabla^2 u + f, \\
-\frac{1}{\rho} \frac{D\rho}{Dt} + \nabla \cdot u = 0,
\]

(1.1)

where \( u \) represents the velocity, \( t \) is the time, \( \rho \) is the density, \( P \) is the pressure, \( f \) is the body force on the fluid, \( \nu \) is the kinematic viscosity, \( \nabla \) is the gradient operator, \( \nabla^2 \) is the Laplacian operator, \( \nabla \cdot \) is the divergence operator and \( \frac{D}{Dt} \) is the convective derivative. The bold face letters are the vector quantities.

1.4.1 Functions representation in SPH

In SPH, the discretized continuum domain can be described as a set of portions. These portions are usually called “Particles”. Each particle has its own physical properties that are interpolated using the properties of its neighbouring particles. Now, the integral interpolant of any function \( A(r) \) is defined by:
A(\textbf{r}) = \int_{\Omega} A(\textbf{r}') \delta(\textbf{r} - \textbf{r}') d\textbf{r}', \quad (1.2)

where \( A(\textbf{r}) \) is a continuous function in the integration domain \( \Omega \) and \( \delta(\textbf{r} - \textbf{r}') \) is the Dirac delta function and given by:

\[
\delta(\textbf{r} - \textbf{r}') = \begin{cases} 
\infty & \text{if } \textbf{r} = \textbf{r}', \\
0 & \text{if } \textbf{r} \neq \textbf{r}'.
\end{cases} \quad (1.3)
\]

Since the Dirac function lacks some required properties for a “well behaved function”, such as continuity and differentiability, and in order to mimic this, it is replaced with an interpolating kernel \( W(\textbf{r}, h) \) to interpolate any value of any physical property \( A \) at a position \( \textbf{r} \) in the domain. Monaghan (1992) defines the integral interpolation as

\[
A(\textbf{r}) = \int_{\Omega} A(\textbf{r}') W(\textbf{r} - \textbf{r}', h) d\textbf{r}', \quad (1.4)
\]

where \( h \) is the smoothing length that determines the size of the neighbouring domain and \( W(\textbf{r} - \textbf{r}', h) \) is a weighted function or kernel that dictates the size of contribution from each neighbouring particle, and is a symmetric function. Monaghan (1992) approximated this integral interpolant by a summation of interpolants, that is

\[
A(\textbf{r}_a) = \sum_b m_b \frac{A_b}{\rho_b} W(\textbf{r}_a - \textbf{r}_b, h),
\]

\[
= \sum_b A_b V_b W_{ab}, \quad (1.5)
\]

where \( A(\textbf{r}_a) \) is a physical property at particle \( a \), \( A_b = A(\textbf{r}_b) \), \( V_b \) is the volume of the particle \( b \), \( \textbf{r}_a - \textbf{r}_b \) is the distance between the particles \( a \) and \( b \) and \( W_{ab} = W(\textbf{r}_a - \textbf{r}_b, h) \) is the weighted or kernel function over the domain of neighbouring particles, as represented in figure 1.3, where \( kh \) is the domain of the support kernel: (Monaghan 1992) From equation 1.5, it is worth noting that the summation approximation gives two features,
1.4. SPH APPROXIMATIONS

Figure 1.3: Neighbouring particles and the kernel function on the domain.

(Vitanza 2014):

- For only the particles inside the support domain, the summation is applied.
- The particle physical quantities (mass and density) appear in the equations.

Because $A$, which appeared in equation 1.2, 1.4 and 1.5 is a generic function, it can be replaced by a particle density $\rho$ to give the density summation approach equation:

$$\rho(r_a) = \sum_b m_b W_{ab}. \tag{1.6}$$

1.4.2 Representation of a function derivative in SPH

The derivative of an interpolated function such as that in equation 1.4, can be obtained by applying the divergence operator to the approximated function $A(r)$ as Liu & Liu (2003), Goffin et al. (2013) and Vitanza (2014):

$$\nabla \cdot A(r) = \int_\Omega \left[ \nabla \cdot A(r') \right] W(r - r', h) \, dr', \tag{1.7}$$

where the derivative is taken with respect to $r'$. Since the following relation holds:

$$\left[ \nabla \cdot A(r') \right] W(r - r', h) = \nabla \cdot \left[ A(r') W(r - r', h) \right] - A(r') \cdot \nabla W(r - r', h),$$
then, equation 1.7 can be written as

$$\nabla \cdot A (r) = \int_\Omega \nabla \cdot \left[ A \left( r' \right) W \left( r - r', h \right) \right] dr' - \int_\Omega A \left( r' \right) \cdot \nabla W \left( r - r', h \right) dr'.$$

Now, using Gauss’s divergence theorem in the first integral equation 1.7 to the right hand side of the last equation, we have:

$$\nabla \cdot A (r) = \int_S A \left( r' \right) W \left( r - r', h \right) \cdot n dS - \int_\Omega A \left( r' \right) \cdot \nabla W \left( r - r', h \right) dr', \quad (1.8)$$

where $S$ is the surface if $\Omega$ at the edge of the interpolation domain and $n$ is the unit normal vector of surface $S$. The first integral of equation 1.8 vanishes (by means $W \left( r - r', h \right) = 0$ when $|r_i - r_j| > kh$). i.e. on the surface $S$ at the edge of the interpolation domain. However, if the smoothing function $W$ in equation 1.8 is truncated at the boundary, such as the free surface, the surface integral on the right hand side of equation 1.8 is no longer zero.

Since the first integral in equation 1.8 is zero. Then the derivative of the function becomes:

$$\nabla \cdot A (r) = -\int_\Omega A \left( r' \right) \cdot \nabla W \left( r - r', h \right) dr', \quad (1.9)$$

Equation 1.9 can be written in discrete form as:

$$\nabla \cdot A (r) = -\sum_b \frac{m_b}{\rho_b} A \left( r_b \right) \cdot \nabla_a W_{ab}, \quad (1.10)$$

Then, for a particle $a$ and its neighbouring particle $b$, equation 1.10 can be written as:

$$\nabla \cdot A (r_a) = -\sum_b \frac{m_b}{\rho_b} A \left( r_b \right) \cdot \nabla_a W_{ab}, \quad (1.11)$$

where $\nabla_a W_{ab} = \nabla W \left( r - r', h \right)$.

Similarly, for the gradient, it can be shown that:

$$\nabla A (r_a) = -\sum_b \frac{m_b}{\rho_b} A \left( r_b \right) \nabla_a W_{ab}, \quad (1.12)$$
1.5. INTERPOLATING KERNEL FUNCTION

where, for example \( \nabla_a W_{ab} = \frac{x_a - x_b}{r_{ab}} \frac{\partial W_{ab}}{\partial r_{ab}} \) is the gradient in the \( x \) direction. Here \( \mathbf{r}_a \) is the vector containing the position of the particle and \( r_{ab} \) is the absolute value of the distance between particles \( a \) and \( b \). Furthermore \( \nabla_a W_{ab} = -\nabla_a W_{ba} \), where \( W_{ab} \) is the weighted or kernel function.

Then equation 1.12 becomes:

\[
\nabla A(\mathbf{r}_a) = \sum_b \frac{m_b}{\rho_b} A(\mathbf{r}_b) \nabla_a W(\mathbf{r}_a - \mathbf{r}_b, h).
\]

(1.13)

1.5 Interpolating Kernel Function

The Kernel approximation or the so-called integral representation is the main feature of the SPH method Liu & Liu (2003).

The choice of the kernel function is subject to the following properties

- Normalization condition \( \int_\Omega W(\mathbf{r} - \mathbf{r}', h) \, d\mathbf{r}' = 1 \)
- Delta function Property \( \lim_{h \to 0} W(\mathbf{r} - \mathbf{r}', h) = \delta(\mathbf{r} - \mathbf{r}') \)
- Compact support condition \( W(\mathbf{r} - \mathbf{r}', h) = 0 \), when \(|\mathbf{r} - \mathbf{r}'| > kh\)
- Positivity \( W(\mathbf{r} - \mathbf{r}', h) \geq 0 \) inside the domain \( \Omega \)
- Monotonically decreasing behaviour of \( W(\mathbf{r} - \mathbf{r}', h) \)

The smoothing length \( h \) and the non-dimensional distance \( q \), between the particles are the main parameters on which the kernel depends. We let \( q = r/h \), where \( r \) is the distance between the particles \( a \) and \( b \). The kernel typically takes the form:

\[
W(\mathbf{r}_{ab}, h) = \frac{1}{h^\sigma} f\left(\frac{|\mathbf{r}_{ab}|}{h}\right),
\]

where \( \sigma \) is the number of dimensions. The most common kernel interpolating function which is normally used in 2D simulation is the cubic spline function (figure 1.4), which
1.5. INTERPOLATING KERNEL FUNCTION

takes the form shown in equation 1.14: (Monaghan 1992).

\[
W_{ab} = \frac{7}{10 \pi h^2} \begin{cases} 
1 - \frac{3}{2}q^2 + \frac{3}{4}q^3 & \text{for } 0 \leq q < 1 \\
\frac{(2-q)^3}{4} & \text{for } 1 \leq q < 2 \\
0 & \text{for } q \geq 2
\end{cases}
\] (1.14)

However, for the best accuracy in 2D simulation, we use the quintic kernel function (Wendland Function) (figure 1.5) which takes the form shown in equation 1.15: (Wendland 1995)

\[
W_{ab} = \frac{7}{4 \pi h^2} \left(1 - \frac{q}{2}\right)^4 (2q + 1) \quad \text{for } 0 \leq q \leq 2
\] (1.15)

In this work, both the cubic spline and the Wendland kernel functions were used.

Figure 1.4: Cubic spline kernel function and its derivative.

Figure 1.5: Quintic (Wendland) kernel function and its derivative.
1.6 Smoothing Length

In the smoothed particle hydrodynamics method, the choice of the smoothing length affects the stability. It has been shown that the error depends on both the smoothing length $h$ and the ratio of particle spacing to smoothing length, $\frac{dx}{h}$ (Quinlan et al. 2006). In the SPH continuous domain, the kernel approximation is of second order accuracy ($O(h^2)$). If the kernel approximation is discretized, the kernel approximation will have a first order accuracy ($O(h)$) (Liu & Liu 2003). The discretization error is affected by the kernel order, and it can increase as the smoothing length decreases (Quinlan et al. 2006). If the smoothing length is small, there are not enough particles inside the range of the influence of the kernel, to interact with the host particle and the simulation becomes unstable. Also, when the smoothing length is chosen to be much bigger, this leads to many particles to be interacted with, and the simulation becomes computationally expensive. In this work, the smoothing length is chosen to be around 1.3 times the initial particle spacing $dx$ and the domain of the support kernel has a radius 2 times the smoothing length, $h$.

1.7 Governing Equations in SPH

In this section we derive the SPH representations for the governing equations of flow. The equations are derived according to Monaghan (1994).

1.7.1 Conservation of mass

There are different methods to calculate the density, by summation method, or by using the continuity equation (Monaghan 1992). The summation density is an SPH formulation to calculate the mass of the particle and can be directly derived from the summation interpolation, by substituting $\rho_a$ in equation 1.5 in place of $A(r_a)$

$$\rho_a = \sum_{b} m_b \frac{\rho_b}{\rho_a} W(r_a - r_b, h),$$

(1.16)

where $\rho_b \neq 0$ which leads to

$$\rho_a = \sum_{b} m_b W_{ab},$$

(1.17)
where $\rho_a = \rho(r_a)$ and $W_{ab} = W(r_a - r_b, h)$.

In SPH simulations, the use of the summation density near edges of the simulation region, such as free surfaces, will lead to a drop in density at the edge. Here we use the second SPH formulation (continuity density method) for applying the conservation of mass and named “Evolved Density” method (Monaghan 1992), (Monaghan 1994). According to the Lagrangian formalism, the total mass of an arbitrary control volume $V$ remains constant (Goffin et al. 2013). That is:

$$\delta m = \rho \delta V,$$  \hspace{1cm} (1.18)

hence, from equation 1.18

$$\frac{D(\delta m)}{Dt} = \frac{D(\rho \delta V)}{Dt} = 0,$$  \hspace{1cm} (1.19)

then from equation 1.19, we have:

$$\frac{D \rho}{Dt} \delta V + \rho \frac{D(\delta V)}{Dt} = 0,$$  \hspace{1cm} (1.20)

for every volume $\delta V$ we can use the properties of the divergence theorem:

$$\frac{D(\delta V)}{Dt} = \int_{\delta V} \nabla \cdot u \, d(\delta V),$$  \hspace{0.5cm} (1.21)

From 1.20 and 1.21 we have:

$$\Longrightarrow \frac{D \rho}{Dt} = -\rho \nabla \cdot u$$  \hspace{1cm} (1.22)

Equation 1.22 is the conservation of mass equation, written in Lagrangian formalism.

Moreover, by using the product rule for the divergence operator, we have:

$$\frac{D \rho}{Dt} = -\rho \nabla \cdot u = -\nabla \cdot (\rho u) + u \cdot \nabla \rho,$$  \hspace{1cm} (1.23)

then for a particle $a$

$$\frac{D \rho_a}{Dt} = -\nabla_a \cdot (\rho u) + u_a \cdot \nabla \rho,$$  \hspace{1cm} (1.24)
then the SPH discrete form is:

\[ \frac{D\rho_a}{Dt} = - \sum_b m_b \rho_b (\rho_b u_b) \cdot \nabla_a W_{ab} + u_a \sum_b m_b \rho_b \cdot \nabla_a W_{ab}, \]  

(1.25)

and simply:

\[ \frac{D\rho_a}{Dt} = \sum_b m_b (u_a - u_b) \cdot \nabla_a W_{ab}, \]  

(1.26)

where \( \nabla_a W_{ab} \) is the gradient of the kernel with respect to the coordinates of the given particle \( a \). Since

\[ q_{ab} = \frac{r_{ab}}{h}, \]

then

\[ \frac{\partial W_{ab}}{\partial x_{ab}} = \frac{\partial q_{ab}}{\partial x_a} \frac{DW_{ab}}{Dq_{ab}}, \]

and we can write

\[ \nabla_a W_{ab} = \frac{r_{ab}}{hr_{ab}} \frac{\partial W_{ab}}{\partial q_{ab}}, \]

where \( r_{ab} = \sqrt{(x_a - x_b)^2 + (y_a - y_b)^2} \) and \( W_{ab} = W(q_{ab}) \).

In this work we have used either the summation density method or the evolved density method, as appropriate for a particular flow simulation. Hence, equation 1.26 can be used to evolve the density by using an appropriate time stepping method to calculate the density of a particle at a given time.

### 1.7.2 Conservation of momentum

In SPH, to formulate the conservation of momentum, we need to calculate the pressure and the viscous forces. The pressure was calculated by applying \( P(r_a) \) instead of \( A(r_a) \) in equation 1.11 to get the following pressure gradient formula:

\[ \nabla P_a = \sum_b m_b \frac{P_b}{\rho_b} \nabla W (r_a - r_b, h). \]  

(1.27)

According to the product rule of differentiation we get

\[ \frac{\nabla P}{\rho} = \nabla \left( \frac{P}{\rho} \right) + \frac{P \nabla \rho}{\rho^2}. \]  

(1.28)
Using this equation, we can reformulate the pressure gradient term given by equation 1.27, by substituting \( \nabla P/\rho \) instead of \( P \) (Vitanza 2014):

\[
\frac{\nabla P_a}{\rho_a} = \sum_b m_b \frac{P_b}{\rho_b^2} \nabla W (r_a - r_b, h) + \frac{P_a}{\rho_a^2} \sum_b m_b \nabla W (r_a - r_b, h)
\]

\[
= \sum_b m_b \left( \frac{P_a}{\rho_a^2} + \frac{P_b}{\rho_b^2} \right) \nabla_a W_{ab}, \tag{1.29}
\]

hence, from equation 1.1, we get the particle approximation of the Navier-Stokes equation, including the viscosity term:

\[
\frac{Du_a}{Dt} = f_a - \sum_b m_b \left( \frac{P_a}{\rho_a^2} + \frac{P_b}{\rho_b^2} \right) (x_a - x_b) \frac{\partial W_{ab}}{\partial x_{ab}} + \pi_{ab}, \tag{1.30}
\]

where \( \pi_{ab} \) is a viscosity term and it can be defined as the laminar viscosity (Morris et al. 1997):

\[
\pi_{ab} = \sum_b m_b \frac{\mu_a + \mu_b}{\rho_a \rho_b} r_{ab} \cdot \nabla_a W_{ab} u_{ab} \tag{1.31}
\]

After defining the viscous term, we can now substitute the viscous term given by equation 1.31 in equation 1.30 to get the SPH momentum equation in the \( x \)-direction. Then we have (Morris et al. 1997):

\[
\frac{Du_a}{Dt} = -\sum_b m_b \left( \frac{P_a}{\rho_a^2} + \frac{P_b}{\rho_b^2} \right) (x_a - x_b) \frac{\partial W_{ab}}{\partial q_{ab}} \\
+ \sum_b m_b \frac{\mu_a + \mu_b}{\rho_a \rho_b} (u_a - u_b) \frac{\partial W_{ab}}{\partial r_{ab}} + f_x, \tag{1.32}
\]

Similarly, for the SPH momentum equation in the \( y \) direction, we have:

\[
\frac{Dv_a}{Dt} = -\sum_b m_b \left( \frac{P_a}{\rho_a^2} + \frac{P_b}{\rho_b^2} \right) (y_a - y_b) \frac{\partial W_{ab}}{\partial q_{ab}} \\
+ \sum_b m_b \frac{\mu_a + \mu_b}{\rho_a \rho_b} (v_a - v_b) \frac{\partial W_{ab}}{\partial r_{ab}} + f_y, \tag{1.33}
\]

where \( r_{ab} \) is the distance between the particles \( a \) and \( b \), \( q_{ab} = r_{ab}/h \), \( u \) is the velocity in the \( x \)-direction, \( v \) is the velocity in the \( y \)-direction, \( f_x \) and \( f_y \) are the forces acting in the \( x \) and \( y \) directions, respectively (Liu & Liu 2003).

Dalrymple (2007) used a different pressure gradient formulae for the relevant equation.
of motion. The first one was as described above and the second one is obtained from applying the product rule of differentiation:

\[ \nabla (P1) = 1 \nabla P + P \nabla 1 \]  

(1.34)

Applying the pressure gradient formula, of equation 1.27, in equation 1.29, we have:

\[ \nabla P_a = \sum_b m_b \frac{P_b}{\rho_b} \nabla_a W_{ab} + P_a \sum_b \frac{m_b}{\rho_b} 1 \nabla_a W_{ab}, \]  

(1.35)

then, we have:

\[ \nabla P_a = \sum_b \frac{m_b}{\rho_b} (P_a + P_b) \nabla_a W_{ab}, \]  

(1.36)

then the Navier-Stokes equation 1.1 will become:

\[ \frac{Du_a}{Dt} = f_a - \sum_b m_b \left( \frac{P_a + P_b}{\rho_a \rho_b} \right) \nabla_a W_{ab} + \nu \nabla^2 u, \]  

(1.37)

then the momentum equation with an artificial viscosity term, which is explained in the next section, is:

\[ \frac{Du_a}{Dt} = f_a - \sum_b m_b \left( \frac{P_a + P_b}{\rho_a \rho_b} + \pi_{ab} \right) \nabla_a W_{ab}, \]  

(1.38)

or with the laminar viscosity, it will be:

\[ \frac{Du_a}{Dt} = f_a - \sum_b m_b \left( \frac{P_a + P_b}{\rho_a \rho_b} \right) \nabla_a W_{ab} + \sum_b m_b \left( \frac{(\mu_a + \mu_b) r_{ab} \nabla_a W_{ab}}{\rho_a \rho_b (r_{ab}^2 + 0.01 h^2)} \right) u_{ab}, \]  

(1.39)

where \( \pi_{ab} \) is as defined in equation 1.31 and \( f_a \) is the body force evaluated at particle \( a \). The 0.01\( h^2 \) is added to the \( r_{ab}^2 \) term, to keep the viscosity bounded for particles as they approach each other.
1.8 Viscosity

In SPH, the use of the viscosity is essential in order to avoid the large unphysical oscillations that can occur without using it (Morris 1995). There are two common types of viscosity used in SPH:

- **Artificial viscosity** The artificial viscosity introduced by Monaghan (1992) and used by Morris et al. (1997), which is the most widely used SPH viscosity formulation:

\[
\pi_{ab} = \begin{cases} 
-\alpha \mu_{ab} c_{ab} + \beta \mu_{ab}^2, & \text{if } v_{ab} \cdot r_{ab} < 0; \\
0, & \text{otherwise},
\end{cases}
\]  

(1.40)

where

\[
\mu_{ab} = \frac{h v_{ab} \cdot r_{ab}}{r_{ab}^2 + 0.01h^2}
\]  

(1.41)

where \(c_{ab} = \frac{c_a + c_b}{2}\) and \(\rho_{ab} = \frac{\rho_a + \rho_b}{2}\), and \(c_a\) is the speed of sound at a particle \(a\). \(\alpha\) is a free parameter which can be changed according to the problem and often \(\alpha \in [0.01, 0.5]\).

- **Laminar viscosity** The Laminar viscosity used by Morris et al. (1997) is defined in 1.31 where in this work, the dynamic viscosity is constant, that is \((\mu_a = \mu_b)\). Also there are other viscosity formulae (Crespo 2008) which are all essentially equivalent to the formula given in equation 1.31, such as:

\[
\frac{D\mathbf{u}_a}{Dt} = f_a - \sum_b m_b \left( \frac{P_a + P_b}{\rho_a \rho_b} \right) \nabla_a W_{ab} + \sum_b m_b \left( \frac{4 \nu_a r_{ab} \nabla_a W_{ab}}{\rho_a + \rho_b \left( r_{ab}^2 + 0.01h^2 \right)} \right) \mathbf{u}_{ab},
\]  

(1.42)

where \(\nu_a = \nu_b\), and

\[
\frac{D\mathbf{u}_a}{Dt} = f_a - \sum_b m_b \left( \frac{P_a + P_b}{\rho_a \rho_b} \right) \nabla_a W_{ab} + \sum_b m_b \left( \frac{4(\mu_a + \mu_b) r_{ab} \nabla_a W_{ab}}{(\rho_a + \rho_b)^2 \left( r_{ab}^2 + 0.01h^2 \right)} \right) \mathbf{u}_{ab},
\]  

(1.43)

where \(\mu = \rho \nu\) is the dynamic viscosity.
1.9 Pressure Formulation

In an SPH simulation for water, we use the stiff equation of state (Batchelor 1967). Monaghan (1994) used this equation of state for water, to model free surface flows:

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

(1.44)

where \( c_s \) is the speed of sound, \( \rho_0 \) is the initial density and \( \gamma \) is a constant and usually chosen to equal 7 in water simulations, so that large pressure variations can be obtained with small variations in density.

Also, adding a background pressure \( \zeta \) to the equation of state can control pressure oscillations in certain flow simulations. That is

\[ P = \frac{c_s^2 \rho_0}{\gamma} \left( \left( \frac{\rho}{\rho_0} \right)^\gamma - 1 + \zeta \right), \]  

(1.45)

where \( \zeta \) is the background pressure, and its value is dependent on the flow being simulated. Morris et al. (1997) introduced the fluid pressure related to the particle density as

\[ p = c_s^2 \rho, \]  

(1.46)

where the speed of sound is chosen to be about ten times the maximum velocity in the flow.

In the simulation of the lid-driven cavity flow, we applied the Monaghan formulation for pressure and it implies that the Mach number remains sufficiently small. The main reason of using Monaghan pressure formulation (the stiff equation of state) in the simulations the work described in this thesis, is that we get a large density variation for a given pressure variation. According to Monaghan (1994) the following condition has to be satisfied, with the characteristic velocity \( u \) of the problem:

\[ Ma = \frac{u}{c_s} < 0.1 \]  

(1.47)

Also, in some flow simulations, it is sensible to initialise the particle density so that the pressure is hydrostatic. The initial density is used to produce the hydrostatic pressure
from the stiff equation of state. Then the initial density is evaluated with

\[ \rho (y_i) = \rho_0 \left( 1 + \frac{\rho_0 g (H - y_i)}{\beta} \right)^{\frac{1}{\gamma}}, \]  

(1.48)

where \( H \) is the initial water depth, \( y_i \) is the vertical particle position and the other parameters are as defined for equation 1.44.

In the literature, there is an alternative form for the equation of state that has the following form:

\[ P = \frac{\gamma}{\gamma - 1} \rho (\rho - \rho_0) \]  

(1.49)

This type of equation was used by researchers such as Müller et al. (2003).

1.10 Density Evaluation

The summation density in SPH is directly derived from the summation interpolation, by substituting \( \rho_a \) in equation 1.5 instead of \( A(\mathbf{r}_a) \) to obtain:

\[ \rho_a = \sum_b \frac{m_b}{\rho_b} \rho_b W (\mathbf{r}_a - \mathbf{r}_b, h), \]  

(1.50)

where \( \rho_b \neq 0 \), which leads to

\[ \rho_a = \sum_b m_b W_{ab}, \]  

(1.51)

where \( \rho_a = \rho (\mathbf{r}_a) \) and \( W_{ab} = W (\mathbf{r}_a - \mathbf{r}_b, h) \).

In SPH simulations, the use of the summation density near the boundaries of the flow region will lead to a reduction in density, due to a deficit of particles surrounding the host particle. At the solid boundaries, additional particles outside of the flow region, can be used to prevent this reduced density occurring, as described in section 1.14. However, this is not possible at free surfaces, so we use the SPH formulation for conservation of mass which we call the “Evolved Density” (Monaghan 1992), as derived in equation 1.26, in section 1.7.1.
1.11 Density Reinitialization

In SPH simulations, we often obtain unphysical pressure oscillations, which are a result of fluctuations in the density. There are two methods employed to control the density fluctuations.

The first method is the Shepard filter which is a quick and simple correction to the density field, and the following procedure is applied usually every 20 time steps in the simulations (Crespo 2008):

\[ \rho_{a}^{\text{new}} = \sum_{b} \rho_{b} \tilde{W}_{ab} \frac{m_{b}}{\rho_{b}} = \sum_{b} m_{b} \tilde{W}_{ab}, \]  

(1.52)

where the kernel has been corrected using the zeroth-order correction:

\[ \tilde{W}_{ab} = \frac{W_{ab}}{\sum_{b} W_{ab} \frac{m_{b}}{\rho_{b}}} \]  

(1.53)

Figure 1.6 shows the effect of using the Shepard filter on the pressure solution in a SPH simulation of a dam break flow. The second method is the first order Moving Least

Square (MLS) approach which was developed by Dilts (1999) and used successfully by Colagrossi & Landrini (2003), but this has not been used in the work described in this thesis, due to it being more computationally expensive.

Figure 1.6: Pressure solution

(a) Without density reinitialisation.  (b) With density reinitialisation.
1.12 Tensile Correction

In SPH simulations, we sometimes obtain particles clumping, which is due to a tensile instability. The clumping of SPH particles is unphysical. To remove this instability we employed the artificial pressure (Monaghan 2000) and (Crespo 2008), the repulsive force is added to a momentum equation in \(x\)-direction as:

\[
\frac{D u_a}{Dt} = - \sum \limits_b m_b \left( \frac{P_a}{\rho_a^2} + \frac{P_b}{\rho_b^2} + RF_{ab}^4 \right) \frac{\partial w_{ab}}{\partial x_a} + \sum \limits_b m_b \frac{\mu_a + \mu_b}{\rho_a \rho_b} (u_{ab}) \left( \frac{\partial w_{ab}}{r_{ab} \partial q_{ab}} \right),
\]

(1.54)

and similarly for the \(y\)-direction as:

\[
\frac{D v_a}{Dt} = - \sum \limits_b m_b \left( \frac{P_a}{\rho_a^2} + \frac{P_b}{\rho_b^2} + RF_{ab}^4 \right) \frac{\partial w_{ab}}{\partial y_a} + \sum \limits_b m_b \frac{\mu_a + \mu_b}{\rho_a \rho_b} (v_{ab}) \left( \frac{\partial w_{ab}}{r_{ab} \partial q_{ab}} \right),
\]

(1.55)

where \(RF_{ab}\) is the tensile correction (Crespo 2008) term and

\[
F_{ab} = \left( \frac{W(q)}{W(\Delta p)} \right),
\]

where \(\Delta p\) is the average particle spacing divided by the smoothing length \(h\) and \(R = R_a + R_b\) where

\[
R_a = \begin{cases} 
0.006 \frac{P_a}{\rho_a^2} & \text{if } P_a > 0 \\
0.6 \frac{P_a}{\rho_a^2} & \text{if } P_a < 0
\end{cases}
\]

and a similar criteria is used for calculating \(R_b\).

When the distance between two particles decreases, this generates a repulsive force between the particles, which prevent them clumping. The next figure shows the effect on the fluid particle distribution of using the tensile correction when the cubic spline kernel was used. Note that when using the Wendland kernel function, the clumping seen in the figure does not occur, so there is no need to use the tensile correction with the Wendland kernel function.
1.13 Initial Conditions

In the SPH code, the initial fluid particles are set to be in a fixed position. Two different types of initial configuration can be chosen, each has one line of wall particles and three lines of dummy particles:

1. **Uniform spacing** (Monaghan & Kos 1999)

   In 2 dimensions, the particles are located at nodes of a square grid. This type of configuration can be seen in figure 1.8 (a). The dimensions of the box are $0.5 \times 0.5$ and in the simulations, the step size is 0.0125. Thus gives the number of particles as 2181 and 501 of them are boundary particles.

2. **Staggered spacing** (Monaghan & Lattanzio 1991)

   In 2 dimensions, the particles are located at nodes of a square grid with a node also in the center of the square. This type of configuration can be seen in figure 1.8 (b). This is a natural configuration that the fluid particles will settle down to, so the boundary particle configuration is then consistent with the fluid particle configuration. The dimensions of the box are $0.5 \times 0.5$ and the step size is 0.0125. The number of particles is 4141, 664 of them are boundary particles.
1.14 Boundary Conditions

In SPH, quantities calculated for the host particles near the boundary of the flow region, will be affected if there is a deficit of particles within the interpolation domain. To avoid problems at the boundaries, dummy particles outside of the flow region, are introduced. In the SPH code, we use three lines of dummy particles, with the cubic spline kernel and the Wendland kernel function. Each with one line of wall particles.

There are various other methods to impose boundary conditions in SPH. Monaghan (1994) introduced boundary particles that are located on the solid boundary and exert repulsive forces to the fluid particles near the boundary, which can prevent fluid particles from penetrating the solid boundary. Lee et al. (2008) and Szewc et al. (2012) applied the fixed dummy particles, where the pressure of the dummy particles is set equal to that of the corresponding wall particles and the density of the dummy particles is set to the initial particle density, i.e. \( P = P_w \) and \( \rho = \rho_0 \), for both weakly compressible and incompressible fluid flows. Also, Libersky & Petschek (1991), Randles & Libersky (1996) and Colagrossi & Landrini (2003) used ghost particles to impose a boundary condition. Marrone et al. (2011) use the fixed ghost particles technique on solid surfaces. The boundary particle methods considered on this thesis are as follows:
1.14. BOUNDARY CONDITIONS

- **Fixed dummy particles**
  
The velocity of the wall particles and the dummy particles is set to the velocity of the wall boundary. In this work, we modify the approach used by Lee et al. (2008) and Szewc et al. (2012), who set the density of the dummy particles to be the initial particle density. Instead, we set the density of the dummy particles to be the same as the density of the wall particles, effectively setting a zero normal density gradient at the boundary. We also effectively set a zero normal pressure gradient at the boundary, as in figure 1.9 (a), by setting the pressure of the dummy particles to be the same as the pressure of the wall particles.
  
In this work, we used this modified boundary condition for the density to obtain improved accuracy in the solution.

- **Ghost particles**
  
The ghost particles are defined outside the fluid domain and are created when the fluid particles are close to the wall boundary, with a distance shorter than the kernel domain. The purpose of the ghost particles is to exert a force on the fluid particles which prevent them moving outside of the flow boundaries. The ghost particle location is a reflection of the fluid particle, about the boundary wall. For the ghost particle, the velocity component in the direction normal to the boundary will be in the opposite direction to that of the fluid particle. The velocity component of the ghost particle, which is tangential boundary, will be in the same direction as the fluid particle for a slip condition, and in the opposite direction as the fluid particle for a no-slip condition. The velocity component shown in figure 1.9 (b) illustrate a slip condition.

- **Fixed Ghost particles**
  
The fixed ghost particles are also defined outside the fluid domain and are created with corresponding interpolation points within the fluid particles. The main reason for the use of fixed ghost particles is to ensure a more regular configuration of particles defining the boundaries. The fluid interpolation points give their properties to the corresponding fixed ghost boundary particles. A slip boundary condition was used. Figure (1.10) shows the fixed ghost boundary particles and the corresponding
1.14. BOUNDARY CONDITIONS

interpolation points which are located within the fluid particles. In the simulation of the Tsunami waves, we use these fixed ghost boundary particles, to ensure a regular configuration of particles on slope boundaries and around an internal corner in the flow. The density and pressure of the fixed ghost boundary particles is set to be the same as the density and pressure of the corresponding interpolation points, i.e. using the condition \( \frac{\partial \rho}{\partial n} = 0 \) and \( \frac{\partial P}{\partial n} = 0 \) at the wall boundaries, to obtain improved accuracy in the solution. SPH results show that using the fixed ghost boundary particle technique produces improved results, especially when treating a complex geometry that included slopes, not aligned with the coordinate system, and an internal 75° corner within the flow. The main advantage of using the fixed ghost boundary particles technique is that it produced a stable and accurate pressure solution in the geometry described.

![Figure 1.9: Boundary conditions.](image)

(a) Dummy particles.  
(b) Ghost particles.

Figure 1.9: Boundary conditions.

![Figure 1.10: Fixed Ghost Particles.](image)

Figure 1.10: Fixed Ghost Particles.
1.15 Moving the Particles

In this work, the particles are moved using the simple numerical scheme, that is:

\[
\frac{dx_a}{dt} = u_a
\]

\[
\frac{du_a}{dt} = F_a
\]

where \( u_a \) and \( F_a \) are the velocity and acceleration, respectively. These values are used at the end of the time step to update the velocity and position as follows:

\[
u^{n+1}_a = u^n_a + F^n_a dt
\]

and for position:

\[
x^{n+1}_a = x^n_a + u^n_a dt + \frac{1}{2} F^n_a dt^2
\]

Also, particles are moved using the XSPH variant Monaghan (1994)

\[
\frac{dx_a}{dt} = V_a + \epsilon \sum_b \frac{m_b}{\bar{\rho}_{ab}} V_{ab} W_{ab} \tag{1.56}
\]

where \( \bar{\rho}_{ab} = \frac{1}{2} (\rho_a + \rho_b) \) and \( \epsilon \) is a constant, whose value is chosen to be \( \epsilon = 0.5 \) in the simulations.

This method is a correction for the velocity of a particle \( a \), which is used to keep the particles more orderly and, in a high speed flow, prevents the penetration of one fluid by another. Note that each particle has effectively two velocities. The velocity \( u^n_a \) is used in the momentum equation, on the subsequent time step, while the XSPH corrected velocity is used only for moving the particles on the current time step (Crespo 2008). In SPH simulations described in this thesis, we used the XSPH variant and conclude that there is a little difference in the velocity profile. Hence, the velocity is enhanced by using XSPH method as shown in figure 1.11, where it is seen that the use of XSPH method has produced a smoother velocity solution.
1.16 Time Step

In this work, we employed two different time stepping schemes. For both schemes, the time step is chosen according to the following limitations:

- Courant-Friedrichs-Lewy (CFL) condition;
- Viscous diffusion condition
- Speed of sound condition

Lee et al. (2008) used these three conditions in an incompressible SPH method. In this work we apply these conditions with the following implementation:

1- Courant-Friedrichs-Lewy (CFL) condition:

The CFL condition is given as:

$$\Delta t_{CFL} \leq \frac{0.25 \, (dx)}{u_{max}}$$

where $dx$ is the initial particle spacing.

2- Viscous diffusion condition:
The viscous diffusion condition is:

$$\Delta t_v \leq \frac{0.125 (dx)^2}{\nu}$$

where $\nu$ is the kinematic viscosity.

3- Speed of sound condition

$$\Delta t_{cs} \leq \frac{0.25 (dx)}{c_s}$$

where $c_s$ is the speed of sound.

The time step is minimum $\delta t$ given by these three conditions, i.e.:

$$\Delta t = \min(\Delta t_{CFL}, \Delta t_v, \Delta t_{cs})$$ \hspace{1cm} (1.57)

using a longer time step leads to instabilities in the SPH simulation.

Then for every time step, we apply a time stepping scheme that we describe as a “mixed Euler-Verlet time step scheme”, and the procedure is:

$$u^{n+1}_a = u^n_a + (\Delta t) F^n_a$$

$$\rho^{n+1}_a = \rho^n_a + (\Delta t) D^n_a$$

$$x^{n+1}_a = x^n_a + (\Delta t) u^n_a + 0.5 (\Delta t)^2 F^n_a$$

Where $F_a = \frac{du_a}{dt}$ and $D_a = \frac{d\rho_a}{dt}$.

The Verlet time step scheme is obtained by writing two third-order time expansions, one forward and one backward in time. The algorithm is to split the equation $du_a/dt = F_a$ into two parts. The variables are calculated using the following procedure (Verlet 1967):

$$u^{n+1}_a = u^{n-1}_a + 2(\Delta t) F^n_a$$

$$\rho^{n+1}_a = \rho^{n-1}_a + 2(\Delta t) D^n_a$$

$$x^{n+1}_a = x^n_a + (\Delta t) u^n_a + 0.5(\Delta t)^2 F^n_a$$
Then on every 20\textsuperscript{th} time step, the second-order Verlet method is replaced by a first-order Verlet method for a single time step to ensure stability (Dalrymple & Rogers 2006). The variables are calculated according to:

\begin{align*}
    u_a^{n+1} &= u_a^n + (\Delta t) tF_a^n \\
    \rho_a^{n+1} &= \rho_a^n + (\Delta t) D_a^n \\
    x_a^{n+1} &= x_a^n + (\Delta t) u_a^n + 0.5 (\Delta t)^2 F_a^n
\end{align*}

The main reason of using the mixed Euler-Verlet time step scheme is to increase the stability, compared to the traditional Euler time step scheme, but it is a scheme that is simpler to implement than the Verlet time step scheme.

### 1.17 Search for Neighbouring Particles (Linked List Method)

In the code, the domain is divided into square cells. These cells are used in a linked list method, which is applied to search for the neighbouring particles, Monaghan & Lattanzio (1985). For each time step, each particle changes its position. So, for the new position, we need to know its neighbouring particles by using the linked list method. This method is an efficient method in searching for the neighbouring particles (Liu & Liu 2003) since it decreases the order of the operations to $N\log N$, while without using it the operations will be of order $N^2$.

In the SPH calculations, we divide the flow region into a Cartesian grid of square cells, with dimensions depending on the kernel domain, as shown in the figure below. Then in the search for the neighbours of the host particle in linked list cell $(i, j)$, the code only needs to consider particles that are in the nine nearest linked list cells $(i-1, j-1)$ to $(i+1, j+1)$. It is the neighbouring particles within the kernel domain, coloured blue, within these linked list cells that interact with the host particle.
1.18 Conclusions

In this chapter, the SPH formulation that will be used have been introduced. The Wendland kernel function will be used (Wendland 1995) as the tensile instability correction is then not needed. The evolve density method (Monaghan 1992), the stiff equation of state (Batchelor 1967) (Monaghan 1994), the laminar viscosity (Morris et al. 1997), Shepard filter (zeroth order density re-normalization) will be applied (Crespo 2008). Both mixed Euler-Verlet and Verlet time step schemes will be tested in the validation cases of lid driven cavity flow and dam break flow. All of the dummy, ghost or fixed ghost boundary conditions will be considered, with the most appropriate selected.
Chapter 2

Simulation of Lid-Driven Cavity Flow using SPH

2.1 Introduction

In this chapter, the SPH application to the lid driven cavity flow is described. The practical implementation of the method and of the chosen options is explained in detail. In this study, numerical studies have been carried out to obtain the steady state solution of the lid driven cavity flow for various Reynolds number (100, 400, and 1000). The results obtained using Fortran code are validated by comparing with the benchmark case by Ghia et al. (1982), Botella & Peyret (1998) and Sahin & Owens (2003). All the numerical results used in the SPH simulations, give good agreement with Ghia et al. (1982). It was found that modifying the density boundary condition enhances the results obtained.

2.2 Simulation Conditions

The lid driven cavity flow is a fluid flow in a closed square where the top (lid) side of the cavity is moving with a constant velocity parallel to itself, while the other three sides are stationary. The cavity size is $L$, the lid velocity is $u_{lid}$, the density is $\rho$, and the viscosity is $\mu$, so the Reynolds number is:

$$Re = \frac{\rho u_{lid} L}{\mu}$$

In the SPH simulations, the lid wall and the solid walls consist of a line of wall of particles and two lines of dummy particles, when using the cubic spline function and the Wendland kernel function. Figure 2.1 shows the initial distribution of fluid particles.
2.2. SIMULATION CONDITIONS

In the simulation at Re = 100, we consider initial distances $dx = dy$ between neighbouring particles of $L/40$, $L/80$ and $L/130$. For the physical properties in the simulation we use $L = 1$, the density $\rho = 1000 \text{ kg/m}^3$ and the kinematic viscosity $\nu = 1/Re \text{ m}^2/\text{s}$.

<table>
<thead>
<tr>
<th>Reynolds number</th>
<th>100</th>
<th>400</th>
<th>1000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Viscosity</td>
<td>0.01</td>
<td>0.0025</td>
<td>0.001</td>
</tr>
</tbody>
</table>

Table 2.1 contains the number of particles information for four different resolutions. For all resolutions there was one line of wall particles and two lines of dummy particles.

<table>
<thead>
<tr>
<th>Particle Resolution</th>
<th>40 × 40</th>
<th>80 × 80</th>
<th>130 × 130</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid particles</td>
<td>1600</td>
<td>6400</td>
<td>16641</td>
</tr>
<tr>
<td>Wall particles</td>
<td>164</td>
<td>324</td>
<td>520</td>
</tr>
<tr>
<td>Dummy particles</td>
<td>352</td>
<td>672</td>
<td>1064</td>
</tr>
<tr>
<td>Total particles</td>
<td>2116</td>
<td>7396</td>
<td>18225</td>
</tr>
<tr>
<td>Particle spacing</td>
<td>1/40</td>
<td>1/80</td>
<td>1/130</td>
</tr>
</tbody>
</table>
2.3. NUMERICAL RESULTS

Figure 2.2: Location of mid horizontal line $x = 0.5$ and mid vertical line $y = 0.5$ in the lid driven cavity flow geometry.

2.3 Numerical Results

We carried out the numerical simulations for $Re = 100$, 400 and 1000 using either the summation or evolved density methods in weakly compressible SPH, with either uniform or staggered initial particle spacing. Also, mixed Euler-Verlet and Verlet time stepping schemes was used with uniform spacing and the modified SPH velocity (XSPH variant) was used with staggered spacing.

For a high Reynolds number simulation using uniform spacing and the evolved density scheme, a hole on the particles occurred near the centre of the cavity, without including background pressure. So, different values of background pressure were considered, to solve this problem, so that the hole disappeared (Ramachandran & Puri 2016) (Sänger et al. 2015). Different values of background pressure (0.01, 0.02, 0.05, 0.07, 0.09 and 0.1) were tested and all gave good results. In the current SPH simulations, calculations the background pressure used was 0.05. Figure 2.3 shows the difference between adding and not adding a background pressure term for a lid driven cavity flow of $Re = 1000$. It is seen that the hole disappears when the background pressure is included. Furthermore, in all simulations, if the Wendland kernel function is used, then we have found that the tensile correction term is not needed. The tensile correction term needs to be used in SPH simulations when the cubic spline function is applied to avoid the instability for which there will be an unphysical clumping of SPH particles when using the cubic spline function.
2.3. NUMERICAL RESULTS

(a) Without adding the background pressure. (b) With adding the background pressure.

*Figure 2.3:* The effect of adding and not adding a background pressure for a lid driven cavity flow of Re = 1000.

In the original simulations carried out, the density of the boundary particles was set to be equal to the initial particle density, $\rho_0$. The boundary condition for the pressure was to effectively set $\frac{\partial P}{\partial n} = 0$ at the boundary, by setting the pressure of the dummy particles to be the same as that of the corresponding wall particles. For high Reynolds number, we set the density of the boundary particles to be the same as the density of the corresponding wall particles, i.e. using the condition $\frac{\partial \rho}{\partial n} = 0$ as the dummy particle boundary condition, to obtain improved accuracy in the solution. This boundary condition was used for both the summation and evolved density methods with uniform and staggered spacing. Figures 2.4 and 2.5 show the $u$ and $v$ velocity profiles on the mid horizontal and the vertical lines, $x = 0.5$ and $y = 0.5$, respectively for Re = 1000. From figure 2.5, it can be concluded that the use of the boundary condition $\frac{\partial \rho}{\partial n} = 0$, improves the solution of the velocity in the SPH simulations of lid-driven cavity flow. In particular, the peaks and troughs in the velocity profile are now captured at the higher resolution, whereas the old boundary condition not convergent to the peaks. Table 2.3, gives a summary of the parameters used in all of the lid driven cavity flow simulations.

From figures 2.6 to 2.19, we can see the smoother and more accurate velocity profile obtained by the higher number of particles, that is using the high resolution produces better results than the low resolution. The $u$-velocity and $v$-velocity profiles, at $x = 0.5$
and \( y = 0.5 \) respectively, for three different particle resolutions are in a good agreement with the Ghia et al. (1982) results who used the vorticity-stream function formulation to study the effectiveness of the coupled implicit multi grid method in the determination of high-Re fine mesh flow simulations.

![Graphs](image1.png)

*Figure 2.4: u and v velocity profile respectively, with \( \rho = \rho_0 \) as the dummy particle boundary condition for Re = 1000.*

![Graphs](image2.png)

*Figure 2.5: u and v velocity profile respectively, with \( \frac{\partial \rho}{\partial n} = 0 \) as the dummy particle boundary condition for Re = 1000.*

### 2.3.1 Simulation of the lid-driven cavity flow at Re = 100

Using the evolved density method in WCSPH gives good results for both the horizontal and vertical velocity profiles. Many cases have been considered for the Re = 100 simulations, for both the summation and evolved density methods, using either the mixed Euler-Verlet or Verlet time step with the cubic spline kernel function or the Wendland kernel function. The present work is then compared with the numerical results obtained by Ghia et al. (1982), Botella & Peyret (1998) and Sahin & Owens (2003). Tables 2.4 to 2.10 contain the minimum \( u \) velocity on the \( y = 0.5 \) line and minimum \( v \) velocity on
the \( x = 0.5 \) line, using the mixed Euler-Verlet time step with cubic spline kernel function or Wendland kernel function. From these tables, it can be seen that there is reasonable agreement between the SPH results and the results obtained by Ghia et al. (1982), Botella & Peyret (1998) and Sahin & Owens (2003). The maximum average density change when using the summation density method, with the higher particle resolution of \( 130 \times 130 \) particles, is 1.84%. However, the variation in the density of particles is 0.5% for the evolved density method.

The same parameters for the summation density at \( \text{Re} = 100 \), are employed with the Wendland kernel function using a mixed Euler-Verlet time stepping scheme and without using the tensile correction term. Tables 2.4, 2.5 and 2.6 show the results of minimum velocity for uniform and staggered spacing respectively, in comparison with Ghia et al. (1982), Botella & Peyret (1998) and Sahin & Owens (2003) results. In the simulations with uniform spacing, using the Wendland kernel function and a particle resolution of \( 130 \times 130 \), there was a gap in the particles near the lid of the cavity and also negative pressures and an oscillating pressure. So, we added 0.05 background pressure in order to solve this problem. The results show that these minimum velocity values are more accurate as the particle resolution increases, as shown in figures 2.6 to 2.19. From these figures, we conclude that the use of summation density methods with mixed Euler-Verlet time step in simulations, gave the best results when we used the Wendland kernel function with staggered spacing. This is due to the natural fluid particle configuration that particles settle, then being consistent with that on the boundary if the staggered spacing is used.

The streamline contours for the cavity flow with \( \text{Re} = 100 \) are shown in figures 2.20 and 2.21 for both uniform and staggered spacing respectively. The vorticity contours are plotted in figure 2.29 for uniform spacing and figure 2.30 for staggered spacing. The shapes of the streamline plots and the vorticity plots are consistent with the results of Ghia et al. (1982). The SPH code was run to 100 seconds to obtain a converged steady state solution.

### 2.3.2 Simulation of the lid-driven cavity flow at \( \text{Re} = 400 \)

In SPH simulations of \( \text{Re} = 400 \), we used the condition \( \frac{\partial \rho}{\partial n} = 0 \) as the dummy particle boundary condition along with \( \frac{\partial P}{\partial n} = 0 \) without applying the XSPH variant. The highest number of particles used is \( 130 \times 130 \), and the maximum variation of density in this
2.3. NUMERICAL RESULTS

Simulation is 1.04%, which was for some particles located at the top right corner of the cavity. For the lower resolutions, the average density variation of the particles is 0.1%, for the evolved density method. In the uniform spacing simulations, with the evolved density method, a small hole appeared in the particles at the centre of the cavity, but after adding a background pressure, the hole disappeared. The XSPH variant was used with the staggered spacing without adding a background pressure and gave good agreement with the results of Ghia et al. (1982). Tables 2.7 and 2.8 show that the SPH calculated minimum velocity is in more better agreement with the results of Ghia et al. (1982) and Sahin & Owens (2003) as the particle resolution increases, for uniform and staggered spacing respectively. Figures 2.12, 2.13 show the velocity profile, using mixed Euler-Verlet time step and Wendland kernel function, with evolved density method and uniform spacing on the $x = 0.5$ and $y = 0.5$ lines respectively. Also, figures 2.14 and 2.15 show the velocity profile, using mixed Euler-Verlet time step and Wendland kernel function, with summation density method and staggered spacing on the $x = 0.5$ and $y = 0.5$ lines respectively. From these results, it can be concluded that when using the summation density method, with mixed Euler-Verlet time step, and Wendland kernel function, then the staggered spacing simulations give a better velocity solution. The streamline contours and the vorticity contours are plotted in figures 2.23, 2.24 and 2.32, 2.33 respectively, and the shapes are consistent with the results of Ghia et al. (1982). The SPH code was run to 140 seconds to obtain a converged steady state solution.

2.3.3 Simulation of the lid-driven cavity flow at Re = 1000

The lid driven cavity flow is now simulated at Re = 1000 using the same parameters and resolutions that were used in section (2.2.2). Tables 2.9 and 2.10 show that, for uniform and staggered spacing respectively, the minimum velocity is in better agreement with Ghia et al. (1982), for the large particle resolutions. Also, figures 2.16, 2.17 and 2.18, 2.19 show the velocity profile, using mixed Euler-Verlet time step and Wendland kernel function with uniform and staggered spacing on the $x = 0.5$ and $y = 0.5$ lines respectively. The best results we have, are when we used the evolved density method with uniform spacing, mixed Euler-Verlet time step and Wendland kernel function. The streamline contours and the vorticity contours are plotted in figures 2.26, 2.27 and 2.35.
2.3. NUMERICAL RESULTS

2.36 respectively, and the shapes are consistent with the results of Ghia et al. (1982). The SPH code was run to 200 seconds to obtain a converged steady state solution.
Table 2.3: Comparison of parameters used in the lid-driven cavity flow simulations.

<table>
<thead>
<tr>
<th>Number of figure</th>
<th>Reynolds number</th>
<th>Summation</th>
<th>Evolved</th>
<th>Uniform</th>
<th>Staggered</th>
<th>Kernel</th>
<th>Time stepping</th>
<th>Tensile correction</th>
<th>Background pressure</th>
<th>XSPH variant</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.6 and 2.7</td>
<td>100</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td></td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>2.8 and 2.9</td>
<td>100</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td></td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>2.10 and 2.11</td>
<td>100</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td></td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>2.12 and 2.13</td>
<td>400</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td></td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>2.14 and 2.15</td>
<td>400</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td></td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>2.16 and 2.17</td>
<td>1000</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td></td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>2.18 and 2.19</td>
<td>1000</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td></td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
</tbody>
</table>
2.3. NUMERICAL RESULTS

Table 2.4: Comparison of Re = 100 simulations between the evolved density method with uniform spacing of present work, with Ghia et al. (1982), Botella & Peyret (1998) and Sahin & Owens (2003) results for the minimum velocity of the moving fluid particles, using mixed Euler-Verlet time step and cubic spline kernel with tensile correction term, on the $x = 0.5$ and $y = 0.5$ lines.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid size</td>
<td>$x$</td>
<td>$y$</td>
<td>$x$</td>
<td>$y$</td>
</tr>
<tr>
<td>40 × 40</td>
<td>-1.78E-01</td>
<td>-1.94E-01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>80 × 80</td>
<td>-1.87E-01</td>
<td>-1.95E-01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>130 × 130</td>
<td>-1.93E-01</td>
<td>-2.00E-01</td>
<td>-2.10E-01</td>
<td>-2.45E-01</td>
</tr>
</tbody>
</table>

Table 2.5: Comparison of Re = 100 simulations between the summation density method of present work, using uniform spacing, with Ghia et al. (1982), Botella & Peyret (1998) and Sahin & Owens (2003) results for the minimum velocity of the moving fluid particles, using mixed Euler-Verlet time step and Wendland spline kernel function without tensile correction term, on the $x = 0.5$ and $y = 0.5$ lines.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid size</td>
<td>$x$</td>
<td>$y$</td>
<td>$x$</td>
<td>$y$</td>
</tr>
<tr>
<td>40 × 40</td>
<td>-1.85E-01</td>
<td>-1.95E-01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>80 × 80</td>
<td>-1.87E-01</td>
<td>-2.15E-01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>130 × 130</td>
<td>-1.95E-01</td>
<td>-2.25E-01</td>
<td>-2.10E-01</td>
<td>-2.45E-01</td>
</tr>
</tbody>
</table>

Table 2.6: Comparison of Re = 100 simulations between the summation density method, using staggered spacing of present work, with Ghia et al. (1982), Botella & Peyret (1998) and Sahin & Owens (2003) results for the minimum velocity of the moving fluid particles, using mixed Euler-Verlet time step and Wendland kernel function without tensile correction term, on the $x = 0.5$ and $y = 0.5$ lines.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid size</td>
<td>$x$</td>
<td>$y$</td>
<td>$x$</td>
<td>$y$</td>
</tr>
<tr>
<td>40 × 40</td>
<td>-1.81E-01</td>
<td>-1.99E-01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>80 × 80</td>
<td>-1.96E-01</td>
<td>-2.24E-01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>130 × 130</td>
<td>-1.97E-01</td>
<td>-2.29E-01</td>
<td>-2.10E-01</td>
<td>-2.45E-01</td>
</tr>
</tbody>
</table>
### 2.3. NUMERICAL RESULTS

*Table 2.7:* Comparison of Re = 400 simulations between the evolved density method, using uniform spacing of present work, with Ghia et al. (1982) and Sahin & Owens (2003) results for the minimum velocity of the moving fluid particles, using mixed Euler-Verlet time step and Wendland kernel function without tensile correction term, on the $x = 0.5$ and $y = 0.5$ lines.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid size</td>
<td>$x$</td>
<td>$y$</td>
<td>$x$</td>
</tr>
<tr>
<td>40 × 40</td>
<td>-1.93E-01</td>
<td>-2.76E-01</td>
<td></td>
</tr>
<tr>
<td>80 × 80</td>
<td>-2.29E-01</td>
<td>-3.21E-01</td>
<td></td>
</tr>
<tr>
<td>130 × 130</td>
<td>-2.92E-01</td>
<td>-4.14E-01</td>
<td>-3.27E-01</td>
</tr>
</tbody>
</table>

*Table 2.8:* Comparison of Re = 400 simulations between the summation density method, using staggered spacing of present work, with Ghia et al. (1982) and Sahin & Owens (2003) results for the minimum velocity of the moving fluid particles, using mixed Euler-Verlet time step and Wendland kernel function without tensile correction term, on the $x = 0.5$ and $y = 0.5$ lines.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid size</td>
<td>$x$</td>
<td>$y$</td>
<td>$x$</td>
</tr>
<tr>
<td>40 × 40</td>
<td>-2.39E-01</td>
<td>-3.40E-01</td>
<td></td>
</tr>
<tr>
<td>80 × 80</td>
<td>-2.79E-01</td>
<td>-3.93E-01</td>
<td></td>
</tr>
<tr>
<td>130 × 130</td>
<td>-2.92E-01</td>
<td>-4.14E-01</td>
<td>-3.27E-01</td>
</tr>
</tbody>
</table>

*Table 2.9:* Comparison of Re = 1000 simulations between the evolved density method, using uniform spacing of present work, with Ghia et al. (1982), Botella & Peyret (1998) and Sahin & Owens (2003) results for the minimum velocity of the moving fluid particles, using mixed Euler-Verlet time step and Wendland kernel function without tensile correction term, on the $x = 0.5$ and $y = 0.5$ lines.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid size</td>
<td>$x$</td>
<td>$y$</td>
<td>$x$</td>
<td>$y$</td>
</tr>
<tr>
<td>40 × 40</td>
<td>-2.49E-01</td>
<td>-3.88E-01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>80 × 80</td>
<td>-2.77E-01</td>
<td>-4.27E-01</td>
<td></td>
<td></td>
</tr>
<tr>
<td>130 × 130</td>
<td>-3.51E-01</td>
<td>-4.93E-01</td>
<td>-3.82E-01</td>
<td>-5.15E-01</td>
</tr>
</tbody>
</table>
Table 2.10: Comparison of Re = 1000 simulations between the summation density method, using staggered spacing of present work, with Ghia et al. (1982), Botella & Peyret (1998) and Sahin & Owens (2003) results for the minimum velocity of the moving fluid particles, using mixed Euler-Verlet time step and Wendland kernel function without tensile correction term, on the \( x = 0.5 \) and \( y = 0.5 \) lines.

<table>
<thead>
<tr>
<th>Grid size</th>
<th>( x )</th>
<th>( y )</th>
<th>( x )</th>
<th>( y )</th>
<th>( x )</th>
<th>( y )</th>
<th>( x )</th>
<th>( y )</th>
</tr>
</thead>
<tbody>
<tr>
<td>40 ( \times ) 40</td>
<td>-2.37E-01</td>
<td>-3.44E-01</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>80 ( \times ) 80</td>
<td>-3.16E-01</td>
<td>-4.38E-01</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>130 ( \times ) 130</td>
<td>-3.41E-01</td>
<td>-4.69E-01</td>
<td>-3.82E-01</td>
<td>-5.15E-01</td>
<td>-3.88E-01</td>
<td>-5.27E-01</td>
<td>-3.88E-01</td>
<td>-5.28E-01</td>
</tr>
</tbody>
</table>

Figure 2.6: \( u \)-velocity profile against \( y \), on the mid horizontal line (\( x = 0.5 \)) with Re = 100 and various particle resolutions, summation density method, uniform spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme.

Figure 2.7: \( v \)-velocity profile against \( x \), on the mid vertical line (\( y = 0.5 \)) with Re = 100 and various particle resolutions, summation density method, uniform spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme.
2.3. NUMERICAL RESULTS

Figure 2.8: $u$-velocity profile against $y$, on the mid horizontal line ($x = 0.5$) with Re = 100 and various particle resolutions, evolved density method, uniform spacing and cubic spline function with tensile correction term and mixed Euler-Verlet time step scheme.

Figure 2.9: $v$-velocity profile against $x$, on the mid vertical line ($y = 0.5$) with Re = 100 and various particle resolution, evolved density method, uniform spacing and cubic spline function with tensile correction term and mixed Euler-Verlet time step scheme.
2.3. NUMERICAL RESULTS

Figure 2.10: $u$-velocity profile against $y$, on the mid horizontal line ($x = 0.5$) with $Re = 100$ and various particle resolutions, summation density method, staggered spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme.

Figure 2.11: $v$-velocity profile against $x$, on the mid vertical line ($y = 0.5$) with $Re = 100$ and various particle resolutions, summation density method, staggered spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme.
2.3. NUMERICAL RESULTS

Figure 2.12: $u$-velocity profile against $y$, on the mid horizontal line ($x = 0.5$) with Re $= 400$ and various particle resolutions, evolved density method, uniform spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme.

Figure 2.13: $v$-velocity profile against $x$, on the mid vertical line ($y = 0.5$) with Re $= 400$ and various particle resolutions, evolved density method, uniform spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme.
2.3. NUMERICAL RESULTS

**Figure 2.14:** \( u \)-velocity profile against \( y \), on the mid horizontal line \((x = 0.5)\) with \( \text{Re} = 400 \) and various particle resolutions, summation density method, staggered spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme.

**Figure 2.15:** \( v \)-velocity profile against \( x \), on the mid vertical line \((y = 0.5)\) with \( \text{Re} = 400 \) and various particle resolutions, summation density method, staggered spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme.
2.3. NUMERICAL RESULTS

Figure 2.16: $u$-velocity profile against $y$, on the mid horizontal line ($x = 0.5$) with $Re = 1000$ and various particle resolutions, evolved density method, uniform spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme.

Figure 2.17: $v$-velocity profile against $x$, on the mid vertical line ($y = 0.5$) with $Re = 1000$ and various particle resolutions, evolved density method, uniform spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme.
2.3. NUMERICAL RESULTS

Figure 2.18: $u$-velocity profile against $y$, on the mid horizontal line ($x = 0.5$) with $Re = 1000$ and various particle resolutions, summation density method, staggered spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme.

Figure 2.19: $v$-velocity profile against $x$, on the mid vertical line ($y = 0.5$) with $Re = 1000$ and various particle resolutions, summation density method, staggered spacing and Wendland kernel function without tensile correction term and mixed Euler-Verlet time step scheme.
2.3. NUMERICAL RESULTS

**Figure 2.20:** Streamline using evolved density method and uniform spacing (130 × 130) for Re = 100.

**Figure 2.21:** Streamline using summation density method and staggered spacing (130 × 130) for Re = 100.

**Figure 2.22:** Streamline of Re = 100, (130 × 130) obtained by Ghia et al. (1982).
2.3. NUMERICAL RESULTS

*Figure 2.23:* Streamline using evolved density method and uniform spacing (130 × 130) for Re = 400.

*Figure 2.24:* Streamline using summation density method and staggered spacing (130 × 130) for Re = 400.

*Figure 2.25:* Streamline of Re = 400, (130 × 130) obtained by Ghia et al. (1982).
2.3. NUMERICAL RESULTS

*Figure 2.26:* Streamline using evolved density method and uniform spacing \((130 \times 130)\) for \(Re = 1000\).

*Figure 2.27:* Streamline using summation density method and staggered spacing \((130 \times 130)\) for \(Re = 1000\).

*Figure 2.28:* Streamline of \(Re = 1000\), \((130 \times 130)\) obtained by Ghia et al. (1982).
2.3. NUMERICAL RESULTS

Figure 2.29: Vorticity using evolved density method and uniform spacing for Re = 100.

Figure 2.30: Vorticity using summation density method and staggered spacing for Re = 100.

Figure 2.31: Vorticity of Re = 100 obtained by Ghia et al. (1982).
2.3. NUMERICAL RESULTS

*Figure 2.32:* Vorticity counter using evolve density method and uniform spacing for $Re = 400$.

*Figure 2.33:* Vorticity using summation density method and staggered spacing for $Re = 400$.

*Figure 2.34:* Vorticity of $Re = 400$ obtained by Ghia et al. (1982).
2.3. NUMERICAL RESULTS

Figure 2.35: Vorticity counter using evolve density method and uniform spacing for Re = 1000.

Figure 2.36: Vorticity using summation density method and staggered spacing for Re = 1000.

Figure 2.37: Vorticity of Re = 1000 obtained by Ghia et al. (1982).
2.4 Conclusions

In this work, we have performed lid driven cavity flow WCSPH simulations at different Reynolds numbers, with uniform and staggered initial particle spacing. The SPH numerical results are in good agreement with the Ghia et al. (1982) data for both types of initial particle configuration. Adding a background pressure, solved the problem of a hole in the particles, which appeared at the centre of the cavity when using the evolved density scheme.

For the boundary condition, by setting the density of the dummy particles to be the same as that of the solid wall particles, so effectively setting a zero normal density gradient, solved the problem of velocity oscillations in the simulation. More accurate results were for the initial uniform particle spacing than with initial staggered particle spacing. Hence, in subsequent simulations the initial uniform spacing will be used, with the Wendland kernel function, the mixed Euler-Verlet time step scheme and the XSPH variant with $\epsilon = 0.5$.

Finally, velocity profile plots show that by increasing the number of particles, the results become more accurate. Furthermore, the use of the Shepard filter technique for density correction will control the density fluctuations and give a more accurate solution.
Chapter 3

Smoothed Particle Hydrodynamics (SPH) Model for Free Surface Flows

3.1 Introduction

In order to verify the effectiveness and accuracy of the developed SPH codes in reproducing hydrodynamic phenomena, for free surface flows, a series of numerical tests are performed and discussed in this chapter. This includes a single phase flow with a free surface in which different boundary implementations are considered. The effect of the different methods of imposing the wall boundary condition, on the smoothness of the pressure distribution are presented. All the results shown in this work, have been obtained using the Wendland or cubic spline kernels, with mixed Euler-Verlet or Verlet time step, the evolved density method, and using $\frac{\partial p}{\partial n} = 0$ and $\frac{\partial P}{\partial n} = 0$ boundary conditions described in chapter 2 section 2.3.

Initially, in SPH simulations, we used traditional fixed dummy particles or ghost dummy particles, but when simulating the experimental model of Tsunami waves, we established that we need to use fixed ghost dummy particles. So, we have some additional simulations for fixed ghost boundary particles just to show that we obtain the same solution.

3.2 Still Water Tank Simulations (Hydrostatic Tank)

Although the simulation of a hydrostatic tank seems to be easy, it is still challenging to achieve long time stability of a hydrostatic tank simulation in SPH. The accuracy of the pressure simulation decreases over time, and also, the particle spacing increases near the free surface. In this thesis, a hydrostatic tank is simulated for comparison and consistency.
3.2. STILL WATER TANK SIMULATIONS (HYDROSTATIC TANK)

Therefore, SPH simulations are carried out in order to establish the SPH parameters and boundary conditions required to obtain an accurate pressure solution in this free surface flow. Getting a simulation started is not straightforward as well, because the particles initially become unstable with time until they settled to the expected hydrostatic pressure (Bergmeister (2015)).

This test case consists of a tank of water with a free surface. The main idea of this test is to check the equilibrium of the system and to show that the pressure solution of the flow is that of hydrostatic pressure, and is a popular basic test for SPH models (Goffin et al. (2013)). The SPH simulation used either staggered or uniform spacing with either Wendland or cubic spline kernel functions. For the physical properties in the simulation, we use the density $\rho = 1000 \text{ kg/m}^3$, the kinematic viscosity $0.001 \text{ m}^2/\text{s}$ and the XSPH variant with $\epsilon = 0.5$. At the beginning of the SPH simulation, the particles are set to have a hydrostatic pressure and it is expected the particles will not move and will keep the initial pressure. Also, fixed wall and dummy particles were used in the simulations. Figure 3.1 shows the initial geometry of still water in a tank, in which $H$ is the initial depth of water.

![Figure 3.1: Initial geometry of the test case of a still water in a tank.](image)

Table 3.1 shows the parameters used in the simulation of a still water in a tank for both staggered and uniform spacing:
Table 3.1: Initial particle distribution for dam break simulations

<table>
<thead>
<tr>
<th>Characteristics</th>
<th>Staggered spacing</th>
<th>Uniform spacing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid particles</td>
<td>900</td>
<td>1600</td>
</tr>
<tr>
<td>Dummy particles</td>
<td>408</td>
<td>528</td>
</tr>
<tr>
<td>Total particles</td>
<td>1308</td>
<td>2128</td>
</tr>
<tr>
<td>Particle spacing</td>
<td>0.016666667</td>
<td>0.0125</td>
</tr>
<tr>
<td>Speed of sound</td>
<td>44.3</td>
<td>44.3</td>
</tr>
<tr>
<td>Simulation time</td>
<td>10s</td>
<td>10s</td>
</tr>
</tbody>
</table>

A simulation has been carried out for the case when the pressure of the dummy particles is set equal to that of the corresponding wall particles and the density of the dummy particles is set to the initial particle density, i.e. \((P = P_w)\) and \((\rho = \rho_0)\). Figure 3.2 (a) shows that at 10s the SPH predicted pressure along the vertical centre line, at \(x = 0.5\), is much lower than the analytical solution. The pressure solution is oscillating, as seen in figure 3.2 (b), where the pressure at the centre of the tank is plotted against time. In order to solve this problem, we apply a hydrostatic pressure to the dummy particles at the base of the tank, that is \(P = P_w + \rho gd\) and also use \(\rho = \rho_w\), where \(d\) is the distance of the dummy particle from the lower wall.

After 10s, this gives an accurate prediction of pressure, as shown in figure 3.3 (a), which shows the SPH predicted pressure along the vertical centre line, at \(x = 0.5\), of the tank. The results show that the pressure of the SPH numerical solution is close to that of the analytic solution. Also, figure 3.3 (b) shows the pressure at the centre of the tank, which over 10s, fluctuates slightly, but gives good agreement with the analytical solution of \(\rho g H\).

The results shown in figures 3.2 and 3.3 were obtained using the Wendland kernel function. Figure 3.7 shows snapshots of the particle positions and pressure at different times, when using the Wendland kernel function with staggered spacing. The results shown in figures 3.4 (a) and (b) used the cubic spline function with staggered spacing and applying a hydrostatic pressure to the dummy particles at the base of the tank, along with \((\rho = \rho_w)\) for boundary conditions. Figure 3.5 (a) and (b) used the same boundary conditions but with the Wendland kernel function and uniform spacing. Then figure 3.8 shows snapshots.
3.2. STILL WATER TANK SIMULATIONS (HYDROSTATIC TANK)

of the pressure solution and particle positions at different times. In addition, Figure 3.6 (a) and (b) used the fixed ghost boundary conditions with the Wendland kernel function and uniform spacing. Then, figure 3.9 shows snapshots of the pressure solution and particle positions at different times.

The use of staggered spacing is to ensure that the fixed wall and dummy particle spacing is consistent with that of the fluid particles, which settle to a staggered type spacing, irrespective of the original particle spacing. Figure 3.7 shows the fluid particle spacing is always consistent with the wall and dummy particle spacing, when the staggered spacing is used. In figures 3.8 and 3.9, it is seen that the spacing of the fluid particles changes from uniform type spacing at $t = 0 \text{s}$ to a staggered type spacing at $t = 10 \text{s}$, which is then inconsistent with the uniform spacing of the wall and dummy particles. However, this does not give any significant improvement on the predicted pressure. It is using $\rho = \rho_w$ and adding hydrostatic pressure to the dummy particles at the base of the tank, that gives an accurate prediction of pressure.

From figures 3.2 to 3.6, it can be concluded that the best solution of pressure with the least oscillations is obtained when used the Wendland kernel function with the uniform spacing, as shown in figure 3.5.

![Figure 3.2: Pressure solution with the use of ($P = P_w$) and ($\rho = \rho_0$), using Wendland kernel function and staggered spacing.](image)

(a) Pressure distribution along vertical centre line $x = 0.5$, at $t = 10\text{s}$.

(b) Pressure at centre of tank against time.
3.2. STILL WATER TANK SIMULATIONS (HYDROSTATIC TANK)

(a) Pressure distribution along vertical centre line \( x = 0.5 \), at \( t = 10 \) s.

(b) Pressure at centre of tank against time.

Figure 3.3: Pressure solution with the use of \( P = P_w + \rho gd \) and \( \rho = \rho_w \), using Wendland kernel function and staggered spacing.

(a) Pressure distribution along vertical centre line \( x = 0.5 \), at \( t = 10 \) s.

(b) Pressure at centre of tank against time.

Figure 3.4: (a) Pressure solution with the use of \( P = P_w + \rho gd \) and \( \rho = \rho_w \), using cubic spline kernel function and staggered spacing.

(a) Pressure distribution along vertical centre line \( x = 0.5 \), at \( t = 10 \) s.

(b) Pressure at centre of tank against time.

Figure 3.5: Pressure solution with the use of \( P = P_w + \rho gd \) and \( \rho = \rho_w \), using Wendland kernel function and uniform spacing with fixed boundary particles.
3.2. STILL WATER TANK SIMULATIONS (HYDROSTATIC TANK)

(a) Pressure distribution along vertical centre line $x = 0.5$, at $t = 10\text{s}$.

(b) Pressure at centre of tank against time.

Figure 3.6: Pressure solution with the use of $P = P_w + \rho gd$ and $(\rho = \rho_w)$, using Wendland kernel function and uniform spacing with fixed ghost boundary particles.

Figure 3.7: Snapshots at different times shows the pressure solution, using staggered spacing and Wendland kernel function.
3.2. STILL WATER TANK SIMULATIONS (HYDROSTATIC TANK)

Figure 3.8: Snapshots at different times show the pressure solution, using uniform spacing with fixed boundary particles and Wendland kernel function.
Figure 3.9: Snapshots at different times shows the pressure solution, using uniform spacing with fixed ghost boundary particles and Wendland kernel function.
3.3 Dam Break Flow Simulations

This test case has been performed using SPH method, by many other workers, such as Colagrossi & Landrini (2003), Hughes & Graham (2010), Cherfils et al. (2012) and Adami et al. (2012).

![Initial geometry of the dam break.](image)

Table 3.2 shows the parameters used in the simulation of dam break flow for both uniform and staggered spacing:

<table>
<thead>
<tr>
<th>Characteristics</th>
<th>Uniform spacing</th>
<th>Staggered spacing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Resolution</td>
<td>70 × 140</td>
<td>100 × 200</td>
</tr>
<tr>
<td></td>
<td>40 × 80</td>
<td>100 × 200</td>
</tr>
<tr>
<td>Fluid particles</td>
<td>9800</td>
<td>20000</td>
</tr>
<tr>
<td></td>
<td>3200</td>
<td>4000</td>
</tr>
<tr>
<td>Wall particles</td>
<td>1961</td>
<td>2801</td>
</tr>
<tr>
<td></td>
<td>2404</td>
<td>6002</td>
</tr>
<tr>
<td>Dummy particles</td>
<td>5908</td>
<td>8428</td>
</tr>
<tr>
<td></td>
<td>6776</td>
<td>16859</td>
</tr>
<tr>
<td>Particle spacing</td>
<td>0.00357</td>
<td>0.0025</td>
</tr>
<tr>
<td></td>
<td>0.00625</td>
<td>0.0025</td>
</tr>
<tr>
<td>Speed of sound</td>
<td>44.3</td>
<td>44.3</td>
</tr>
<tr>
<td></td>
<td>44.3</td>
<td>44.3</td>
</tr>
<tr>
<td>Simulation time</td>
<td>2s</td>
<td>2s</td>
</tr>
<tr>
<td></td>
<td>2s</td>
<td>2s</td>
</tr>
</tbody>
</table>

3.3.1 Simulation of dam break flow using fixed boundary particles with uniform spacing

In this case, we chose $H = 0.5$, so $W = 0.25$ and $D = 0.5$. Water is held behind the dam, in a rectangular region, as shown in figure 3.10 and the dam is then instantaneously released at time $t = 0$, and the water starts moving due to the effects of gravity. Simulations were carried out using the uniform spacing and both mixed Euler-Verlet and Verlet time step schemes. Two different resolutions were used, with 9800 and 20000 particles,
3.3. DAM BREAK FLOW SIMULATIONS

with a particle spacing of $H/140$ and $H/200$ respectively. In figure 3.11, the position of the leading edge is normalised with respect to the width, $W$ of the initial column of the water, and is plotted against normalised time, $t(2g/w)^{1/2}$. The current SPH results shown in figure 3.11 for the leading edge location, are in acceptable agreement with the WCSPH results of Hughes & Graham (2010) and also the results of Greaves (2006), which were obtained using adaptive quadtree grids. It is seen in figure 3.11 that the current SPH results predict that the leading edge is moving more rapidly than in the ISPH predictions of Hughes & Graham (2010) and the experimental data of Martin & Moyce (1952).

Figure 3.12 shows the height of the water column on the $x = 0$ line, and the results give a very close agreement with the numerical results of Hughes & Graham (2010) and Greaves (2006) and also the experimental data of Martin & Moyce (1952). Moreover, the results are more accurate than the ISPH results of Hughes & Graham (2010).

Figure 3.13 shows the dam break solution at various normalized times. It can be seen that the fluid splashes up the wall at the right and then fluid moves downwards and starts flowing in the left direction. Figure 3.14 shows the SPH prediction for the dam break flow, at two different times, using uniform spacing with an mixed Euler-Verlet time step scheme, with 9800 fluid particles in (a) and (d) as well as 20000 particles in (c) and (d). There is not much difference between these resolutions for both the leading edge location and water column height, but the higher particle resolution improves the surface definition in areas of high deformation. The same parameters were used for the simulations shown in figure 3.15 but with the Verlet time step scheme.

In conclusion, there is little difference in the leading edge location and water column height predictions between using mixed Euler-Verlet and Verlet time stepping and using 9800 and 20000 particles, indicating a converged solution. However, the difference in the prediction between using mixed Euler-Verlet and Verlet time step, with 20000 particles, is that there is no hole in the particles, approximately halfway across the fluid at $t\sqrt{2g/W} = 8.029$ s, when using the Verlet time step scheme. This hole is present in the solutions obtained by Hughes & Graham (2010) and Greaves (2006).
3.3. DAM BREAK FLOW SIMULATIONS

3.3.2 Simulation of dam break flow using ghost boundary particles with uniform spacing

The dam break flow is now simulated using ghost boundary particles. Uniform spacing and an mixed Euler-Verlet time step scheme are used, with 9800 and 20000 particles, which correspond to a particle spacing of $H/140$ and $H/200$ respectively. The current SPH results shown in figure 3.16, for the leading edge location, are in acceptable agreement with the WCSPH results of Hughes & Graham (2010) and Greaves (2006) numerical results, but it is seen that the leading edge is predicted to move more rapidly than in the ISPH results of Hughes & Graham (2010) and the experimental data of Martin & Moyce.
3.3. DAM BREAK FLOW SIMULATIONS

Figure 3.13: Dam break solution at various normalized times.

Figure 3.17 shows the height of the water column, and the current SPH results give a very close agreement with the numerical results of Hughes & Graham (2010) and Greaves (2006) and also the experimental data of Martin & Moyce (1952). Also, the results have shown that there is little difference in the prediction of leading edge location and water column height, between using fixed boundary particles and ghost boundary particles.

Figure 3.18 shows the SPH prediction for the dam break flow, at two different times, using uniform spacing with ghost dummy particles and an mixed Euler-Verlet time step scheme, with 9800 fluid particles in (a) and (d) as well as 20000 particles in (c) and (d). There is not much difference between these resolutions for both the leading edge location and the water column height, but the higher particle resolution improves the surface definition in areas of high deformation.
3.3. DAM BREAK FLOW SIMULATIONS

3.3.3 Simulation of dam break flow using fixed boundary particles with staggered spacing

The dam break flow is now simulated using fixed boundary particles, but initially the particles are in the staggered spacing configuration. The simulations have been carried out using both mixed Euler-Verlet and Verlet time stepping schemes, with 6400 or 40000 particles, which correspond to a particle spacing of \(H/80\) and \(H/200\) respectively. The current SPH results shown in figure 3.19, for the leading edge location, are in acceptable

\[
\sqrt{2g/W} = 9.124, \quad 11.25
\]

Figure 3.14: \(W = H/2, D = H\) dam break solution for mixed Euler-Verlet time step using uniform spacing at \(\sqrt{2g/W} = 9.124\), using (a) 9800 fluid particles (c) 20000 fluid particles and at \(\sqrt{2g/W} = 11.25\), using (b) 9800 fluid particles (d) 20000 fluid particles.
3.3. DAM BREAK FLOW SIMULATIONS

Figure 3.15: $W = H/2$, $D = H$ dam break solution for Verlet time step using uniform spacing at $\sqrt{2g/W} = 9.124$, using (a) 9800 fluid particles (c) 20000 fluid particles and at $\sqrt{2g/W} = 11.25$, using (b) 9800 fluid particles (d) 20000 fluid particles.

Figure 3.15: $W = H/2$, $D = H$ dam break solution for Verlet time step using uniform spacing at $\sqrt{2g/W} = 9.124$, using (a) 9800 fluid particles (c) 20000 fluid particles and at $\sqrt{2g/W} = 11.25$, using (b) 9800 fluid particles (d) 20000 fluid particles.

agreement with the results of Hughes & Graham (2010) and Greaves (2006), but it is seen that the leading edge is predicted to move more rapidly than in the ISPH results of Hughes & Graham (2010) and the experimental data of Martin & Moyce (1952).

Figure 3.20 shows the height of the water column, and the current SPH results give very good agreement with the results of Hughes & Graham (2010) and Greaves (2006) and also the experimental data of Martin & Moyce (1952).

Figure 3.21 shows the SPH prediction for the dam break flow, at two different times, using staggered spacing and a mixed Euler-Verlet time step scheme, with 6400 fluid particles in
3.3. DAM BREAK FLOW SIMULATIONS

Figure 3.16: Comparison of the leading edge location using uniform particle spacing with ghost boundary particles and mixed Euler-Verlet time step.

Figure 3.17: Comparison of the water column height using uniform particle spacing with ghost boundary particles and mixed Euler-Verlet time step.

(a) and (d) as well as 40000 particles in (c) and (d). There is not much difference between these resolutions for both the leading edge location and water column height, but the higher particle resolution improves the surface definition in areas of high deformation. The same parameters were used for the simulations shown in figure 3.22 but with the Verlet time step scheme.

It is seen that compared to when using fixed dummy particles with the uniform spacing, as in figures 3.14 and 3.15, there is less splashing of particles when using either ghost dummy particles or the fixed dummy particles with the staggered spacing, as in figures 3.18, 3.21 and 3.22.
3.3. DAM BREAK FLOW SIMULATIONS

Figure 3.18: \( W = H/2, D = H \) dam break solution for mixed Euler-Verlet time step using uniform spacing and ghost boundary particles at \( t\sqrt{2g/W} = 9.124 \), using (a) 9800 fluid particles (c) 20000 fluid particles and at \( t\sqrt{2g/W} = 11.25 \), using (b) 9800 fluid particles (d) 20000 fluid particles.
3.3. DAM BREAK FLOW SIMULATIONS

Figure 3.19: Comparison of the leading edge location using staggered spacing with two different resolutions and two different time step schemes.

Figure 3.20: Comparison of the water column height using staggered spacing particle with two different resolutions and two different time step schemes.
3.3. DAM BREAK FLOW SIMULATIONS

Figure 3.21: $W = H/2, D = H$ dam break solution for mixed Euler-Verlet time step using staggered spacing at $t\sqrt{2g/W} = 9.124$, using (a) 6400 fluid particles (c) 40000 fluid particles and at $t\sqrt{2g/W} = 11.25$, using (b) 6400 fluid particles (d) 40000 fluid particles.
3.3. DAM BREAK FLOW SIMULATIONS

Figure 3.22: W = H/2, D = H dam break solution for Verlet time step using staggered spacing at $t \sqrt{2g/W} = 9.124$ s, using (a) 6400 fluid particles (c) 40000 fluid particles and at $t \sqrt{2g/W} = 11.25$ s, using (b) 6400 fluid particles (d) 40000 fluid particles.
3.3.4 Simulation of dam break flow using fixed ghost boundary particles with uniform spacing

In this case, simulations were carried out using the uniform spacing, with a mixed Euler-Verlet time step scheme and fixed ghost boundary particles. In these simulations, 20000 particles were used, which corresponds to a particle spacing of $H/200$. In figure 3.23 the position of the leading edge is normalised with respect to the width, $W$ of the initial column of the water, and is plotted against normalised time, $t(2g/w)^{1/2}$. The current SPH results shown in figure 3.23, for the leading edge location, are in acceptable agreement with the results of Hughes & Graham (2010) and Greaves (2006), but it is seen that the leading edge is predicted to move more rapidly than in the ISPH results of Hughes & Graham (2010) and the experimental data of Martin & Moyce (1952).

Figure 3.24 shows the height of the water column on the $x = 0$ line, and the current SPH results give a very close agreement with the results of Hughes & Graham (2010) and Greaves (2006) and the experimental data of Martin & Moyce (1952).

Figure 3.25 shows the SPH prediction for the dam break flow, using uniform spacing with fixed ghost dummy particles and a mixed Euler-Verlet time step scheme with 20000 particles.
3.3. DAM BREAK FLOW SIMULATIONS

Figure 3.23: Comparison of the leading edge location using uniform spacing with fixed ghost boundary particles and 20000 fluid particles.

Figure 3.24: Comparison of the water column height using uniform spacing particle with fixed ghost boundary particles and 20000 fluid particles.
3.3. DAM BREAK FLOW SIMULATIONS

Figure 3.25: $W = H/2, D = H$ dam break solution for mixed Euler-Verlet time step, using uniform spacing and fixed ghost boundary particles at $t\sqrt{2g/W} = 9.124$ and $t\sqrt{2g/W} = 11.25$, using 20000 fluid particles.
3.4 Conclusions

In this chapter, we have performed WCSPH simulations for a still water tank, to determine the density and pressure boundary conditions that should be used in order to obtain the correct pressure solution of the flow, which is hydrostatic pressure. The SPH simulations used either staggered or uniform spacing, with either the Wendland or the cubic spline kernel functions. Simulations have been carried out using either fixed dummy particles or fixed ghost dummy particles. By setting the density of the fixed dummy particles, in the boundary condition, to be the same as that of the solid wall particles, so effectively setting a zero normal density gradient, solved the problem of velocity oscillations in the simulation. This gives more accurate results for uniform spacing than the staggered spacing in comparison with the analytic solution. Also, the dam break flow simulations, discussed using both uniform and staggered spacing with either fixed dummy particles, ghost particles or fixed ghost dummy particles. Based on this chapter’s results for a dam break flow, simulations for tsunami waves will be carried out using the initial uniform spacing, Wendland kernel function, mixed Euler-Verlet time step scheme, and the XSPH variant with $\epsilon = 0.5$.

In conclusion, the current SPH results, for the leading edge location, are in acceptable agreement with the results of Hughes & Graham (2010) and Greaves (2006), but it is seen that the leading edge is predicted to move more rapidly than in the ISPH results of Hughes & Graham (2010) and the experimental data of Martin & Moyce (1952). For the height of the water column, the current SPH results give a very close agreement with the results of Hughes & Graham (2010) and Greaves (2006) and the experimental data of Martin & Moyce (1952).
Chapter 4

Tsunami Wave Simulated by a Fault Rupture using a Horizontal Uplift Plate

4.1 Introduction

In this chapter, we describe the SPH simulations of model-scale tsunami waves generated by an earthquake fault rupture, and in particular, the boundary conditions needed to obtain an accurate and stable solution. The model-scale experiments are described, and then we consider the effect of the length of the SPH numerical wave tank on the surface elevation predictions obtained. It would be computationally expensive to simulate the full length of the experimental wave tank.

We use an SPH model wave tank that is sufficiently long so that the predictions of surface elevation, at the wave gauge locations considered, are not influenced by the end wall of the tank. The SPH predicted surface elevations are then compared with the experimental measurements, and are generally in good agreement. There is also a discussion of possible reasons for any differences between the SPH simulations, and the experiments. The surface profiles have also been used to calculate wave speeds, and there is reasonably good agreement between the SPH simulations, experiments and theory. In this chapter, we also consider the effect of the uplift speed and the water depth on the free surface elevation.
4.2 Simulations of a Tsunami Wave Generated by a Flat Uplift Plate

4.2.1 Experimental model

The experimental models of tsunami waves generated by an earthquake fault rupture have been carried out at University of Plymouth’s COAST laboratory. This model consists of a horizontal water tank with a water depth of 0.3 m and 20 m long. The tsunami wave is generated by the rapid uplift of a plate on the bed of the tank, where it moves up very rapidly by 0.06 m in 0.2 s. Figure 4.1 (a) shows the upstream region of the tank, where the horizontal uplift plate is located, and figure 4.1 (b) shows the downstream region of the tank. The side view of the experimental tank in figure 4.2 shows the wave gauge locations. Eight wave gauges were placed along the centreline of the tank. The distance between each wave gauge is 0.1m.

![Figure 4.1: Experimental wave tank](image)

(a) Upstream region, with the red box indicating the location of the flat uplift plate.
(b) Downstream region.

4.2.2 SPH model

In this work we use the SPH model described in subsection 4.2.1, to simulate the flow that occurs in experimental models of tsunamis generated by an earthquake fault rupture. We use a standard SPH approximation as defined in Monaghan (1994) and for the best
4.2. SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

Figure 4.2: Plate and wave gauges locations.

accuracy in the SPH simulations, we use the quintic kernel function (Wendland function). However, as the simulations involve a free surface, it is necessary to calculate the particle density by evolving the density, using the rate of change of density at a particle. If the density was calculated using an SPH summation, then there would be an inaccurate lower density at the free surface, due to a deficit of SPH particles within the kernel region of influence of particles at or near the free surface. In the SPH simulations of water, we use a stiff equation of state (Batchelor, 1967). Monaghan (1994) applied this equation of state for water to model free surface flows. The speed of sound is usually chosen to be ten times the maximum fluid velocity, and $\gamma$ is usually chosen to be equal 7 in water simulations, so that large pressure variations can be obtained with small variations in the density. It was found that modifying the pressure boundary condition, by adding a hydrostatic pressure gradient ($\rho g$) to the dummy particles below the moving plate and the floor of the wave tank, enhances the results obtained. In SPH simulations of a still water tank, adding a hydrostatic pressure gradient ($\rho g$) to the dummy particles at the base of the water tank was required in order to maintain a hydrostatic pressure solution over time. Also, a Shepard filter was used, which is a quick and simple correction to the density field, to avoid any unphysical pressure oscillations which are a result of fluctuations in the density, and is usually applied every ten time steps, in the simulations (Crespo, 2008). A mixed Euler-Verlet time stepping scheme was used to update the particle position, velocity and density on each time step.
In this work, we have carried out simulations using both fixed dummy particles and ghost dummy particles and found that we obtained a more stable and accurate pressure solution if we use ghost dummy particles.

The fixed dummy particles are defined at regularly spaced positions behind a line of fixed wall particles, and their position does not change throughout the simulation. With the Wendland kernel function, we need two lines of fixed dummy particles.

The ghost particles are defined outside the fluid domain and are created when the fluid particles are close to the wall boundary, within a distance shorter than the kernel domain from any boundary wall. The purpose of the ghost particles is to exert a force on the fluid particles which prevents them moving outside of the flow boundaries. The pressure and density of the ghost particles is to be the same as that of the corresponding fluid particles.

In this work, a slip boundary condition is used on the solid walls of the flow geometry. Figure 4.3 (a) shows a typical configuration of fluid and ghost particles at a wall boundary and also illustrates particle velocity components when a slip boundary condition is used.

Figure 4.3 (b) shows how ghost dummy particles are created to define the corner of the moving plate. In this region there will be ghost dummy particles created by reflecting fluid particles from 3 different fluid regions, 3, 4 and 5, with the corresponding ghost dummy particles denoted as type $3'$, $4'$ and $5'$ respectively. Fluid particles in a given region are then only influenced by the corresponding ghost dummy particles for that region. For example, fluid particles in region 3 would be influenced by ghost dummy particles of type $3'$, but would not be influenced by ghost dummy particles of types $4'$ and $5'$.

Figure 4.3 (b) also shows the ghost dummy particles in the flow region at the edge of the moving plate, with a different colour used to represent each type of ghost dummy particle.

Figure 4.4 shows a schematic representation of the wave tank. The SPH model consists of a truncated horizontal water tank, with a water depth of 0.3 m. We have carried out simulations for various length SPH model wave tanks in order to establish the effect of the end wall boundary conditions on the SPH simulations. The tsunami wave is generated by the rapid uplift of a plate on the bed of the tank, where it moves up very rapidly and the vertical displacement time history is shown in figure 4.5. The plate displacement profile
4.2. SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

(a) Ghost particles.

(b) Ghost particles creation at the corner of the moving plate.

Figure 4.3: Ghost boundary

is applied to the plate boundary in the SPH model. The uplifted plate is 0.6 m long x 0.6 m wide (the full width of the flume). The SPH simulations used approximately 250,000 particles, with a particle spacing of 0.003 m. In this work, initial uniform spacing with mixed Euler-Verlet time step scheme have been used with Wendland kernel function and XSPH variant ($\epsilon = 0.5$). The tank contains eight wave gauges (WG), and the distance between each wave gauge is 0.1 m, with the first located in line with the end of the moving plate. A slip boundary condition is used at the end wall in the SPH simulations.

Figure 4.4: Schematic representation of wave tank.

SPH simulations have been carried out for model wave tanks of three different lengths (3 m, 5.5 m and 7.5 m), in order to establish if there is any effect of the SPH tank end wall boundary condition, on the predicted surface height over the time range considered, which was until approximately 2.5 seconds after the plate started moving. In this work, the surface elevation is defined as the depth of water related to the depth of the surface above the bed of the tank. Figure 4.6 and 4.7 show the SPH predicted surface height at
wave gauges 1 and 8 respectively, for the three different lengths of SPH model wave tank. It can be seen that very similar results are obtained at wave gauge 1 for each of the three different lengths and that for wave gauge 8, there is little difference in the surface height predictions obtained from the 5.5 m and 7 m length SPH wave tank simulations. Hence, we conclude that for the time range considered, the SPH surface height predictions at the wave gauge locations are not influenced by the SPH end wall boundary condition, if the length of the SPH wave tank is at least 5.5 m. It is noted that, with the wave speeds obtained in the experiments and the SPH simulations, the wave does not reach the end of the 5.5 m and 7.5 m length wave tanks, within the time range shown in these figures.

An SPH interpolation of the density is calculated along vertical lines of fictitious points (each at a distance of $dx/10$ apart) at each wave gauge location and the free surface is defined where the SPH interpolated density is below a certain value (850 kg/m$^3$). Using
4.2. SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

Figure 4.7: Effect of tank length on WG8 surface elevation using SPH method.

A higher resolution of SPH particles decreases the ‘noise’ in the prediction of the free surface location, as shown in figure 4.8, where the SPH predicted surface height at wave gauge 1 is presented for initial particle spacings of $dx = 0.006$ and $dx = 0.003$.

Figure 4.8: Comparison of WG1 surface elevation using SPH method with two different particle resolution.

Figure 4.9: SPH plate displacement and elevation at WG1.
4.2. SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

Figure 4.10: Experiment plate displacement and elevation at WG1.

Figure 4.9 shows the SPH predicted surface height at the first wave gauge together with
the plate displacement applied in the SPH simulations. It shows that the free surface at
wave gauge 1 starts moving almost instantly after the plate starts moving. Figure 4.10
shows the same comparison for the experiments. From figures 4.9 and 4.10 we conclude
that both the SPH simulation and experimental trends in free surface height at wave gauge
1 are similar. However, in the SPH simulations, the surface height rises more quickly after
the plate starts moving, than in the experiments. We believe that the difference is due to
some water being able to move around below the plate, as it lifts upwards in the experi-
ments, whereas in the SPH simulations all of the water is always above the plate. In the
experiments, any movement of water from above to below the plate is through small gaps
at the plate edges. If this were to be included in the SPH model, then it would not be
possible to have a sufficient resolution of particles in the gaps without making the simu-
lations too computationally expensive.

Figures 4.11 and 4.12 respectively show the SPH predictions and the experimental mea-
urements of surface height at the eight wave gauges. In the SPH simulations and the
experiments the plate starts moving at $t = 1.09$ s, so still water is simulated from $t = 0$ s
to $t = 1.09$ s in the SPH model, which enables the SPH particles to settle in to a natural
configuration before the plate starts moving. Note that initially, at $t = 0$ s, the SPH par-
ticles are positioned on a regular grid, which is not a natural configuration for the SPH
particles. In figures 4.11 and 4.12 we see that in general, the SPH simulations correctly
model the increase in surface height and the movement of the wave along the wave tank.
There is agreement between the time at which the surface height starts to increase, and
4.2. SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

the peak height, at each of the wave gauge locations.

![Figure 4.11: SPH predictions of surface height at the eight different wave gauges.](image)

![Figure 4.12: Experimental measurements of surface height at the eight different wave gauges.](image)

Figures 4.13 to 4.20 show a direct comparison between the SPH predictions and experimental measurements of surface height at eight wave gauges. In figures 4.13 to 4.20 it is seen that there are differences between the SPH and experimental surface elevations when the surface level falls, with the depth of trough under predicted in the SPH simulations. We believe that this is because, in the experiments, some water could move around below the plate as it moves upwards, whereas in the SPH model, all fluid remains above the plate at all times. For this reason, the largest differences between the SPH and experimental surface height profiles shown in figures 4.13 to 4.20, occur between $t = 2.5$ s and $t = 3.5$ s.
4.2. SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

**Figure 4.13:** Comparison between SPH predictions and experimental measurements of surface height at WG1.

**Figure 4.14:** Comparison between SPH predictions and experimental measurements of surface height at WG2.

**Figure 4.15:** Comparison between SPH predictions and experimental measurements of surface height at WG3.
4.2. SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

Figure 4.16: Comparison between SPH predictions and experimental measurements of surface height at WG4.

Figure 4.17: Comparison between SPH predictions and experimental measurements of surface height at WG5.

Figure 4.18: Comparison between SPH predictions and experimental measurements of surface height at WG6.
4.2. SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

Figure 4.19: Comparison between SPH predictions and experimental measurements of surface height at WG7.

Figure 4.20: Comparison between SPH predictions and experimental measurements of surface height at WG8.
4.2. SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

Figure 4.21: SPH pressure solution near the plate rupture the times at which it is moving upwards.
The phase velocity, $V_{ph}$, or tsunami speed, of a shallow water gravity wave is defined in Truong (2012) as $V_{ph} = \sqrt{g(D + A)}$, where $D$ is the water depth and $A$ is the wave amplitude. Using the wave amplitude at wave gauge 4 from both the SPH simulations and the experiments, $V_{ph}$ is calculated as 1.78 m/s for the SPH simulations and 1.77 m/s for the experiments. We have calculated a tsunami wave speed using the peaks in the surface height profiles shown in figures 4.11 and 4.12. That is, we have used the time between the surface height peaks between two wave gauges (4 and 8), to calculate the speed at which the wave travels. These calculations give a SPH tsunami wave speed of 1.74 m/s and an experimental wave speed of 1.67 m/s. We have also calculated wave speeds by using the time difference in the zero crossing height between wave gauges 4 and 8. This gives a SPH tsunami wave speed of 1.74 m/s and an experimental wave speed of 1.74 m/s. These calculations show that there is reasonably good agreement between the wave speed obtained in the SPH simulations with that obtained in the experiments, and also with that predicted by the theory in Truong (2012).

Figure 4.21 shows the SPH pressure solution near the plate over the times at which it is moving upwards. Note that the plate moves upwards from $t = 1.09$ s to $t = 1.29$ s. In figure 4.21 it is seen that the initial hydrostatic pressure that was set at $t = 0$ s is maintained at $t = 1$ s and that there is an increase in pressure above the plate when it starts moving upwards at $t = 1.09$ s. Figure 4.22 shows the SPH predicted vertical velocity when the plate is half way up, at $t = 1.19$ s, and it is seen that in the section of water above the plate, it is all moving upwards at a very similar velocity. Figure 4.23 shows the SPH pressure solution, near the plate end of the SPH model tank, at various times after the plate has moved upwards, where it is seen that, as expected, the pressure settles to a hydrostatic solution. This figure also shows the shape of the wave and how it travels along the wave tank as time progresses.
4.2. SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

Figure 4.22: SPH vertical velocity solution when the plate is half way up, at $t = 1.19$ s.

Figure 4.23: SPH pressure solution at various times after the plate has moved upwards.
4.3 Including the Small Sloped Ramp in the Simulations of a Tsunami Wave Generated by a Flat Uplift Plate

The experimental wave tank has a sloped ramp, of height 0.06 m and length 0.224 m, on the base of the wave tank, immediately to the right of the plate, as shown in figure 4.25. This is so that the experimental tank can also be adapted for other experimental setups, with a slope on top of the moving uplift plate, which aligns with the sloped ramp fixed to the base, when the plate has moved upwards. A landslide can also then be created down the continual slope. We have carried out SPH simulations which include the sloped ramp in the model using ghost particles, and, in terms of the free surface position, very similar results were obtained both with and without the sloped ramp in the SPH model. However, the SPH simulations that include the sloped ramp had an unstable pressure solution, which we believe was caused by the SPH particles close to the sloped ramp moving into an irregular configuration, as the simulation progressed in time. Therefore, the results presented in section 4.2 did not include a small sloped ramp in the SPH model. In order to solve the unstable pressure problem, we have considered the use of ‘fixed’ ghost dummy particles to define the sloped ramp, and have changed the full tank boundary conditions from the use of ghost boundary particles to ‘fixed’ ghost boundary particles. The main advantage of the use of ‘fixed’ ghost boundary particles, instead of the standard ghost boundary particles, is that the boundary particles position is always fixed and does not depend on the fluid particles position. This should ensure a more regular particle configuration close to the sloped ramp and hence a more stable pressure solution.

In this technique, the ghost boundary particles are fixed and are created only once at the beginning of the simulation. To compute the properties of each fixed ghost particle, an interpolation point is associated to it. This interpolation point is obtained by reflecting, about the relevant boundary, the position of the fixed ghost particle into the fluid domain. The main fluid particles are equally spaced with a distance $dx$ and the fixed ghost boundary particles are created outside of the fluid domain in the normal direction to the boundary. The first line of fixed ghost boundary particles are at a distance $dx/2$ from the fluid boundary and subsequent lines are spaced a distance $dx$ apart.

A slip boundary condition has been used for the velocity along the solid boundaries.
4.3. INCLUDING THE SMALL SLOPED RAMP IN THE SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

For pressure, a Neumann boundary condition is used along the solid boundaries, that is \( \frac{\partial p}{\partial n} = 0 \), where \( n \) is in the normal direction to the solid boundary. On the slopes in the wave tank, the tangential velocity to the slope, at the fixed ghost particles, is in the same direction as the tangential velocity at the corresponding interpolation points. However, the normal velocity to the slope, at the fixed ghost particles, is in the opposite directions as the normal velocity at the corresponding interpolation points. This ensures that \( \mathbf{u} \cdot \mathbf{n} = 0 \) at the boundary.

Figure 4.24 shows the velocity components for fixed ghost particles and the corresponding interpolation points, on the slopes in the wave tank. The slopes are at 15° to the horizontal. So, until the moving uplift plate has finished moving upwards and is then in line with the top corner of the small sloped ramp, there is an internal 75° corner within the flow at the top of the small sloped ramp. The velocity values \( u \) and \( v \) at the interpolation point have been determined from an SPH summation of the velocity values at particles in the neighbouring domain of the interpolation point. Then, using these values, we calculate the tangential, \( u_T \), and normal, \( u_N \), velocity to the slope at the interpolation points, as defined in equation 4.1. The tangential and normal velocity of the fixed ghost dummy particles are denoted as \( u_{Td} \) and \( u_{Nd} \) respectively. Hence, the velocity of the fixed ghost dummy particles, \( u_d \) and \( v_d \), can be calculated. For a slip condition the values of \( u_{Td} \) and \( u_{Nd} \) are set as defined in equation 4.2. These values are then used to calculate the velocity components of the fixed ghost particles, \( u_d \) and \( v_d \), as defined in equation 4.3.

\[
\begin{align*}
u_T &= u \cos(\alpha) - v \sin(\alpha), \\
u_N &= u \sin(\alpha) + v \cos(\alpha)
\end{align*}
\] (4.1)

Then set
\[
\begin{align*}
u_{Td} &= u_T, \\
u_{Nd} &= -u_N
\end{align*}
\] (4.2)

then calculate \( u_d \) and \( v_d \) as:
\[
\begin{align*}
u_d &= u_{Td} \cos(\alpha) + u_{Nd} \sin(\alpha), \\
v_d &= u_{Nd} \cos(\alpha) - u_{Td} \sin(\alpha)
\end{align*}
\] (4.3)
where $u_T$ and $u_N$ are the tangential and normal velocity of the interpolation points. $u_{Td}$ and $u_{Nd}$ are tangential and normal velocity of the fixed ghost dummy particles.

Figure 4.24: Fixed ghost boundary particles on the slopes in the wave tank.

Figure 4.26 shows the use of fixed ghost boundary particles in SPH simulations near the flat moving plate, together with the sloped ramp at the edge of the moving uplift plate, with a different colour to represent each type of fixed ghost boundary particle. For boundary particle close to the $75^\circ$ corner, the corresponding interpolation points are dependent on the location of the moving uplift plate. The properties of the red coloured ghost particles will be determined from the green coloured interpolation points until the uplift plate is within a distance of $dx/2$ from the top. After this, because the green coloured interpolation points are not within the fluid, as the plate is now located here, the properties of the red coloured ghost particles will be determined from the blue coloured interpolation points. These points are positioned perpendicular to the corresponding slope boundary particles, as on the rest of the slope. The use of this fixed ghost boundary particles helps to have a stable pressure solution at the $75^\circ$ degree corner in the geometry.

Figures 4.27 to Figure 4.30 show the SPH predictions of surface height at four different wave gauges, in which the simulations without the slope use ghost particles and the simulations with the slope use fixed ghost particles. It is seen that there is a good agreement between both sets of predictions.

Figure 4.31 shows the SPH pressure solution near the flat plate and the sloped ramp over the times at which it is moving upwards. Note that the plate moves upwards from
4.3. INCLUDING THE SMALL SLOPED RAMP IN THE SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

t = 1.09 s to t = 1.29 s. In figure 4.31 it is seen that the initial hydrostatic pressure that was set at t = 0 s is maintained at t = 1 s and that there is an increase in pressure above the plate when it starts moving upwards at t = 1.09 s. Figure 4.32 shows the SPH predicted vertical velocity when the plate is half way up, at t = 1.19 s, and it is seen that in the section of water above the plate, it is all moving upwards at a very similar velocity. Figure 4.33 shows the SPH pressure solution, near the flat uplift plate and the small sloped ramp, at various times after the plate has moved upwards, where it is seen that, as expected, the pressure settles to a hydrostatic solution. This figure also shows the shape of the wave and how it travels along the wave tank as time progresses.
4.3. INCLUDING THE SMALL SLOPED RAMP IN THE SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

Figure 4.27: Comparison between SPH predictions of surface height at WG1 with and without the small sloped ramp.

Figure 4.28: Comparison between SPH predictions of surface height at WG3 with and without the small sloped ramp.

Figure 4.29: Comparison between SPH predictions of surface height at WG5 with and without the small sloped ramp.
4.3. INCLUDING THE SMALL SLOPED RAMP IN THE SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

Figure 4.30: Comparison between SPH predictions of surface height at WG7 with and without the small sloped ramp.

Figure 4.31: SPH pressure solution near the plate rupture the times at which it is moving upwards.
4.3. INCLUDING THE SMALL SLOPED RAMP IN THE SIMULATIONS OF A TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE

Figure 4.32: SPH vertical velocity solution when the plate is half way up, at $t = 1.19\, \text{s}$.

Figure 4.33: SPH pressure solution at various times after the plate has moved upwards.
4.4 The Effect of Plate Uplift Time and Water Depth on the Model-Scale Tsunami Waves

The main reason for this investigation is to consider the effect of changing the flat uplift plate time, by making it faster, (0.11 s) and slower (0.31 s and 0.41 s) than the main experiment flat uplift plate time of 0.21 s, on the free surface elevation results. This investigation has been carried out for SPH simulations of a tsunami wave generated by a flat uplift plate, moving a distance 0.06 m upwards, with a 5.5 m tank length, including a small sloped ramp, of height 0.06 m and length 0.224 m, on the base of the wave tank, immediately to the right of the plate. The comparison of SPH simulations results, for the flat uplift plate time 0.21 s, with the experiment data have been discussed in detail in section 4.3.

Figure 4.34 shows a plot for the uplift displacement profile, for various uplift times between 0.11 s and 0.41 s, where it moves up very rapidly by 0.06 m. Figures 4.35 to 4.38 show the SPH predicted surface height at wave gauges 1, 2, 4 and 8 respectively, for the four different uplift times. It is concluded from figures 4.35 to 4.38, that the change in the uplift time affects the surface elevation profile near the uplift plate, as seen at WG1 and WG2, but it is not affected further downstream, as seen at WG4 and WG8, where the surface elevation profiles for all uplift speeds are very similar. At WG1 the peak surface elevation is larger when the uplift plate moves up more rapidly, whereas at WG8 the peak surface elevation is approximately the same for all uplift times.

Also, SPH simulations have been carried out for four different water depths, D, (0.16 m, 0.3 m, 0.46 m and 0.6 m), in order to establish if there is any effect of the SPH tank depth on the free surface elevation. All of these simulations have been tested with a plate uplift time of 0.21 s and a plate displacement of 0.06 m. Therefore in each of these cases, the plate displacement to water depth ratio is different, ranging from a plate displacement of 10% of the water depth when $D = 0.6$ m to 37.5% of the water depth when $D = 0.16$ m. Figures 4.39 to 4.42 show the differences in surface height for various initial water depths at wave gauges 1, 2, 4 and 8 respectively. Hence, we conclude that for a shallower depth of water, with a given uplift plate time and displacement, the increase in surface elevation takes place over a longer time, as seen at all four wave gauges (WG1, WG2, WG4 and
4.4. THE EFFECT OF PLATE UPLIFT TIME AND WATER DEPTH ON THE MODEL-SCALE TSUNAMI WAVES

WG8). Also, as the wave travels downstream, the maximum increase in surface elevation is fairly constant (≈ 0.03 m) at all four wave gauges with a shallower depth of water (0.16 m). However, for the largest depth of water considered (0.6 m), the maximum increase in surface elevation reduces as the wave travels downstream from WG1 to WG8.

Table 4.1 contains the tsunami speed calculated from these simulations. The tsunami wave speed has been calculated using the peaks in the surface height profiles shown in figures 4.37 and 4.38 and figures 4.41 and 4.42. That is, we have used the time between the surface height peaks between two wave gauges (4 and 8), to calculate the speed at which the wave travels. We have also calculated wave speeds by using the time difference in the zero crossing height between wave gauges 4 and 8. These calculations show that there is reasonably good agreement between the wave speed obtained in all of these SPH simulations with that obtained in the experiments, and also with that predicted by the theory in Truong (2012) as $V_{ph} = \sqrt{g(D + A)}$, where $D$ is the water depth and $A$ is the wave amplitude at wave gauge 4 from the SPH simulations.

Figure 4.34: Plot of the uplift displacement profile, for various uplift times.
4.4. THE EFFECT OF PLATE UPLIFT TIME AND WATER DEPTH ON THE MODEL-SCALE TSUNAMI WAVES

**Figure 4.35:** Surface elevation at WG1 for various plate uplift times.

**Figure 4.36:** Surface elevation at WG2 for various plate uplift times.

**Figure 4.37:** Surface elevation at WG4 for various plate uplift times.
4.4. THE EFFECT OF PLATE UPLIFT TIME AND WATER DEPTH ON THE
MODEL-SCALE TSUNAMI WAVES

**Figure 4.38:** Surface elevation at WG8 for various plate uplift times.

**Figure 4.39:** Difference in surface elevation at WG1 for various water depths.

**Figure 4.40:** Difference in surface elevation at WG2 for various water depths.
4.4. THE EFFECT OF PLATE UPLIFT TIME AND WATER DEPTH ON THE
MODEL-SCALE TSUNAMI WAVES

Figure 4.41: Difference in surface elevation at WG4 for various water depths.

Table 4.1: Tsunami speed between WG4 and WG8

<table>
<thead>
<tr>
<th></th>
<th>Peaks between WG4 to WG8</th>
<th>Zero crossing between WG4 to WG8</th>
<th>Truong (2012) at WG4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plate displacement 0.11 s</td>
<td>1.6 (m/s)</td>
<td>1.6 (m/s)</td>
<td>1.788 (m/s)</td>
</tr>
<tr>
<td>Plate displacement 0.21 s</td>
<td>1.74 (m/s)</td>
<td>1.67 (m/s)</td>
<td>1.787 (m/s)</td>
</tr>
<tr>
<td>Plate displacement 0.31 s</td>
<td>1.6 (m/s)</td>
<td>1.6 (m/s)</td>
<td>1.788 (m/s)</td>
</tr>
<tr>
<td>Plate displacement 0.41 s</td>
<td>1.6 (m/s)</td>
<td>1.67 (m/s)</td>
<td>1.785 (m/s)</td>
</tr>
<tr>
<td>Tank depth 0.16 m</td>
<td>1.34 (m/s)</td>
<td>1.667 (m/s)</td>
<td>1.796 (m/s)</td>
</tr>
<tr>
<td>Tank depth 0.3 m</td>
<td>1.74 (m/s)</td>
<td>1.67 (m/s)</td>
<td>1.787 (m/s)</td>
</tr>
<tr>
<td>Tank depth 0.46 m</td>
<td>1.62 (m/s)</td>
<td>1.8 (m/s)</td>
<td>1.779 (m/s)</td>
</tr>
<tr>
<td>Tank depth 0.6 m</td>
<td>1.73 (m/s)</td>
<td>1.9 (m/s)</td>
<td>1.77 (m/s)</td>
</tr>
</tbody>
</table>

Figure 4.42: Difference in surface elevation at WG8 for various water depths.
4.5 Conclusions

In this chapter, we used the SPH method to simulate the flow that occurs in experimental models of tsunami waves generated by a fault rupture. In the experiments, the “fault rupture” is created by the rapid uplift of the horizontal plate on the bed of a wave tank. Simulations were carried out both including and not including the small sloped ramp that is in the experimental setup. In order to obtain an accurate pressure solution in the region around the small sloped ramp, careful consideration of the boundary conditions was required. It was established that “fixed” ghost particles must be used. The results show that, the SPH model gives generally good predictions of the free surface position. At each wave gauge location, there is good agreement between the SPH predictions and experimental measurements of the peak surface height and also the time it takes for the free surface height to increase and then return to the zero crossing height. There is also agreement between the calculated wave speeds from the SPH simulations and the experiments. However, relative to when the plate starts moving upwards, the surface height rises more quickly in the SPH simulations than in the experiments and the trough depths are underpredicted by the SPH model. We believe that these differences are due to some water being able to move around and below the moving plate in the experiments, whereas in the SPH model all of the water is always above the plate. The SPH simulations also show that, as expected, using a higher particle resolution decreases the noise in the predicted free surface location. The validated SPH model has then been used to consider the effect of plate uplift time and water depth on the surface elevation. It was shown that the surface elevation profiles, particularly at the downstream wave gauge locations, vary more with changes in water depth than with changes in plate uplift time. It is noted that the uplift plate displacement was constant, at 0.06 m, so as the depth of water changed, the displacement to depth ratio also changed.
Chapter 5

Tsunami Wave Generated by Fault Rupture Using an Inclined Uplift Plate (IU)

5.1 Introduction

In this chapter, we consider model-scale tsunami waves which are generated by an inclined uplift plate. The experiments are described, which have been carried out at University of Plymouth’s COAST laboratory. The experimental wave tank contains an inclined uplift plate and also a small sloped ramp, on the base of the wave tank. The SPH model tank is sufficiently long so that the predictions of surface elevation, at the wave gauge locations considered, are not influenced by the end wall of the tank. The SPH predicted surface elevations are then compared with the experimental measurements, and are generally in good agreement. There is also a discussion of possible reasons for any differences between the SPH simulations and the experiments. The surface profiles have also been used to calculate wave speeds, and there is reasonably good agreement between the SPH simulations, experiments and theory. After this discussion, a comparison is made between the surface elevation profiles obtained with the flat uplift plate (which is discussed in chapter 4) and those obtained with the inclined uplift.

5.2 Experimental Model with an Inclined Uplift Plate (IU)

The experiments with an inclined uplift plate were carried out in a horizontal water tank with, a water depth of 0.32 m and 20 m long. The inclined uplift plate is located at the
upstream end of the tank. Figure 5.1 shows the experimental inclined slope model. The inclined uplift plate is 0.6 m in length, and 0.6 m wide, i.e. the full width of the wave tank. The vertical side plates are joined by a cross-brace where the actuator is attached to perform the uplifting motion. All plates are made of aluminium. The horizontal plate is 0.005 m thick and the remaining elements are 0.01 m thick, to ensure that they do not bend during the uplift motion. Stiffening bars were added to the plate, to prevent it from bending due to the large uplift force. The distance between each wave gauge (16 in total) is 0.3 m, with the first wave gauge located in line with the downstream end of the inclined uplift plate. The tsunami wave is generated by the rapid uplift of the inclined uplift plate, which is located on the bed of the tank. The plate moves upwards by 0.06 m in approximately 0.2 s.

Figure 5.1: Inclined uplift plate used in the experiment.

5.3 SPH Simulation of Tsunami Waves Generated by an Inclined Uplift Plate (IU)

For the SPH simulations of an inclined uplift plate, the SPH model used was virtually the same as that for the flat uplift plate (Chapter 4). The only difference in the boundary condition at the location of the plate, which is now a slope. The fixed ghost particles boundary condition method used on the slope is as described in section (4.3).

Figure 5.2 shows a schematic representation of the wave tank. The SPH model consists of a truncated horizontal water tank which is 11 m long, with an inclined slope located
on the bed of the tank at the upstream end of the tank. The water depth of 0.32 m. The tsunami wave is generated by the rapid uplift of the inclined uplift plate, where it moves up very rapidly and the vertical displacement time history is shown in figure 5.3. The plate displacement profile is applied to the inclined uplift plate boundary in the SPH model. The SPH simulations used approximately 230,000 particles, with a particle spacing of 0.004 m. The tank contains 16 wave gauges (WG), and the distance between each wave gauge is 0.3 m, with the first located in line with the end of the moving inclined uplift plate. A slip boundary condition is used at the end wall in the SPH simulations.

Figure 5.2: Schematic representation of wave tank, including a sloped ramp.

Figure 5.3: Experiment plate displacement for inclined uplift.
5.3. SPH SIMULATION OF TSUNAMI WAVES GENERATED BY AN INCLINED UPLIFT PLATE (IU)

Figure 5.4 shows the SPH predicted surface height at the first wave gauge together with the plate displacement applied in the inclined uplift plate SPH simulations. In figures 5.4 and 5.5 it is seen that the free surface at WG1 starts moving almost instantly after the plate starts moving. From figures 5.4 and 5.5 we conclude that both the SPH simulation and experimental trends in free surface height at wave gauge 1 are similar. Also, for both the SPH simulations and the experiments, the surface height rises at the same time (1.07 s), almost immediately after the plate starts moving.

Figures 5.6 and 5.7 respectively show the SPH predictions and the experimental measurements of surface height at the 16 wave gauges. In the SPH simulations and the experiments the inclined slope uplift starts moving at $t = 1.07$ s, so still water is simulated from $t = 0$ s to $t = 1.07$ s in the SPH model, which enables the SPH particles to settle in to a natural configuration before the plate starts moving. Note that initially, at $t = 0$ s, the SPH particles are positioned on a regular grid, which is not a natural configuration for the
SPH particles. In figures 5.6 and 5.7 we see that in general, the SPH simulations correctly model the increase in surface height and the movement of the wave along the wave tank. There is agreement between the time at which the surface height starts to increase, and the general shape of the surface elevation profile, at each of the wave gauge locations. It is seen that, as the wave travels along the tank, the peak height recorded at each wave gauge location in the experiments, reduces more rapidly than is predicted by the SPH simulations. As with the flat uplift plate case, the troughs in the surface elevation profiles are under predicted by the SPH simulations. We expect that this for the same reasons as described in chapter 4, that is water being able to move around and below the moving uplift plate in the experiments.

Figures 5.8 to 5.16 show the SPH predicted surface height at wave gauges 1, 2, 4, 6, 8, 10, 12 and 16 respectively. From these figures, we conclude that for the time range considered, the SPH surface height predictions at the wave gauge locations, for an 11m tank length, are not influenced by the SPH end wall boundary condition. It is noted that, with the wave speeds obtained in the experiments and the SPH simulations, the wave does not reach the end of the 11 m length wave tank, within the time range shown in these figures.

Figures 5.8 to 5.16 show that there are differences between the SPH and experimental surface elevations when the surface level falls, with the height of peaks under predicted in the SPH simulations. We believe that this is because, in the experiments, some water could move around below the inclined slope as it moves upwards, whereas in the SPH model, all fluid remains above the inclined uplift at all times. In these figures, it is seen that there are oscillations in the experimental surface elevations profiles, after wave has passed a particular wave gauge. The amplitude of these oscillations increases in the downstream direction. These oscillations are not fully captured by the SPH predictions, but the SPH surface elevation profiles do show a small oscillation at the equivalent times. It is not fully understood what causing these oscillations From the length of the experimental and the calculated wave speeds, it is not a returning wave from the end of the tank.
5.3. SPH SIMULATION OF TSUNAMI WAVES GENERATED BY AN INCLINED UPLIFT PLATE (IU)

Figure 5.6: SPH predictions of surface height at the sixteen different wave gauges.

Figure 5.7: Experimental measurements of surface height at the sixteen different wave gauges.
5.3. SPH SIMULATION OF TSUNAMI WAVES GENERATED BY AN INCLINED UPLIFT PLATE (IU)

Figure 5.8: Comparison between SPH predictions and experimental measurements of surface height at WG1.

Figure 5.9: Comparison between SPH predictions and experimental measurements of surface height at WG2.

Figure 5.10: Comparison between SPH predictions and experimental measurements of surface height at WG4.
5.3. SPH SIMULATION OF TSUNAMI WAVES GENERATED BY AN INCLINED UPLIFT PLATE (IU)

**Figure 5.11:** Comparison between SPH predictions and experimental measurements of surface height at WG6.

**Figure 5.12:** Comparison between SPH predictions and experimental measurements of surface height at WG8.

**Figure 5.13:** Comparison between SPH predictions and experimental measurements of surface height at WG10.
5.3. SPH SIMULATION OF TSUNAMI WAVES GENERATED BY AN INCLINED UPLIFT PLATE (IU)

Figure 5.14: Comparison between SPH predictions and experimental measurements of surface height at WG12.

Figure 5.15: Comparison between SPH predictions and experimental measurements of surface height at WG14.

Figure 5.16: Comparison between SPH predictions and experimental measurements of surface height at WG16.
5.3. SPH SIMULATION OF TSUNAMI WAVES GENERATED BY AN INCLINED UPLIFT PLATE (IU)

The phase velocity, $V_{ph}$, or tsunami speed, of a shallow water gravity wave is defined in Truong (2012) as $V_{ph} = \sqrt{g(D + A)}$, where $D$ is the water depth and $A$ is the wave amplitude. Using the wave amplitude at wave gauge 8 from both the SPH simulations and the experiments, $V_{ph}$ is calculated as 1.84 m/s for the SPH simulations and 1.82 m/s for the experiments. We have calculated a tsunami wave speed using the peaks in the surface height profiles shown in figures 5.12 and 5.16. That is, we have used the time between the surface height peaks between two wave gauges (8 and 16), to calculate the speed at which the wave travels. These calculations give a SPH tsunami wave speed of 1.84 m/s and an experimental wave speed of 1.67 m/s. We have also calculated wave speeds by using the time difference in the zero crossing height between wave gauges 8 and 16. This gives a SPH tsunami wave speed of 2 m/s and an experimental wave speed of 1.71 m/s. These calculations show that there is reasonably good agreement between the wave speed obtained in the SPH simulations with that obtained in the experiments, and also with that predicted by the theory in Truong (2012).

Figure 5.17 shows the SPH pressure solution near the inclined uplift plate over the times at which it is moving upwards. Note that the inclined uplift plate moves upwards from $t = 1.07$ s to $t = 1.27$ s. In figure 5.17, it is seen that the initial hydrostatic pressure that was set at $t = 0$ s is maintained at $t = 1$ s and that there is an increase in pressure above the inclined slope when it starts moving upwards at $t = 1.07$ s, up to $t = 3$ s when the pressure settles to a hydrostatic solution. Figure 5.18 shows the SPH predicted vertical velocity when the inclined slope uplift is half way up, at $t = 1.17$ s, and it is seen that in the section of water above the inclined slope, it is all moving upwards at a very similar velocity. Figure 5.19 shows the SPH pressure solution over 11 m SPH model tank length, at various times after the inclined slope has moved upwards, where it is seen that, as expected, the pressure settles to a hydrostatic solution. This figure also shows the shape of the wave and how it travels along the wave tank as time progresses.
5.3. **SPH SIMULATION OF TSUNAMI WAVES GENERATED BY AN INCLINED UPLIFT PLATE (IU)**

**Figure 5.17:** SPH pressure solution near the inclined slope uplift over the times at which it is moving upwards.

**Figure 5.18:** SPH vertical velocity solution when the inclined uplift is half way up, at \( t = 1.17 \text{s} \).
Figure 5.19: SPH pressure solution at various times after the inclined uplift has moved upwards.
5.4 Comparison of Tsunami Wave Generated by a Flat Uplift Plate and an Inclined Uplift Plate

In this section, a comparison is made between the SPH predicted surface elevations obtained with a flat uplift plate and those obtained with an inclined uplift. The comparison between the experiments and the SPH simulations for the flat plate case is detailed in chapter 4, section 4.3, and for the inclined uplift case is in section 5.3. In the flat uplift plate case the wave gauges were at a distance of 0.1 m apart, whereas in the inclined uplift plate case the wave gauges were at a distance of 0.3 m apart. These were the wave gauges spacing used in the experiments. For the SPH simulations of the inclined uplift plate case, surface elevations have also calculated at wave gauges 0.1 m apart. This allows a direct comparison to be made between the surface elevations obtained with the two different types of uplift plate, flat or inclined. This figure shows the wave gauge locations used in this comparison of the flat and inclined uplift plate cases. Figures 5.20 shows a schematic representation of the wave tank for both the flat and inclined uplift plate cases. Figures 5.21 to 5.24 show a direct comparison between the SPH predictions, for both flat plate and inclined uplift simulations, of surface height predictions at the wave gauges 1, 2, 4 and 8.

![Figure 5.20: Schematic representation of the SPH wave tank for the inclined uplift model.](image-url)
5.4. COMPARISON OF TSUNAMI WAVE GENERATED BY A FLAT UPLIFT PLATE AND AN INCLINED UPLIFT PLATE

Figure 5.21: Comparison between SPH predictions of surface height for the flat plate and the inclined uplift at WG1.

Figure 5.22: Comparison between SPH predictions of surface height for the flat plate and the inclined uplift at WG2.

Figure 5.23: Comparison between SPH predictions of surface height for the flat plate and the inclined uplift at WG4.
5.5 Conclusions

In this chapter, we used the SPH method to simulate the flow that occurs in experimental models of tsunami waves generated by a fault rupture. In the experiments, the fault rupture is created by the rapid uplift of an inclined uplift plat on the bed of a wave tank. The results show that the SPH model gives generally good predictions of the free surface position. At each wave gauge location, there is reasonable agreement between the SPH predictions and experimental measurements of the surface elevation profiles. However, the peaks in these profiles are slightly lower in the experimental data than in the SPH predictions, and the difference between the experimental peak and SPH peak becomes larger as the wave travels downstream. Also, the troughs in the surface elevation profiles are underpredicted by the SPH simulations and in the experiments, the free surface height returns to the zero crossing height slightly earlier than in the SPH predictions. We believe that these differences are due to some water being able to move around and below the moving inclined uplift plat in the experiments. There is also a reasonable agreement between the calculated wave speeds from the SPH simulations, the experiments and the theory. The comparison of SPH predicted surface elevation profiles between the flat uplift plate case, and the inclined uplift plate case, showed that there was a little difference in the profiles obtained for these two types of uplift plate.
Chapter 6

Tsunami Wave Generated by Moving Landslide on a Fixed Inclined Plate

6.1 Introduction

In this chapter, we describe the SPH model used to simulate waves generated by a landslide block that moves down an inclined plate. The experiments, which have been carried out at University of Plymouth’s COAST laboratory, are described. We use an SPH model wave tank, consisting of a fixed inclined plate, and a moving landslide block which slides down the fixed inclined plate. There is also a comparison between the SPH simulations of two different landslide block heights (0.018 m and 0.036 m).

6.2 Experimental model of a Fixed Inclined Plate with Moving Landslide

The experimental model consists of a horizontal water tank with a water depth of 0.37 m and it is 20 m long. The landslide is modelled by a block moving down an inclined uplift plate, which is located at the upstream end of the tank. Figure 6.1 shows the experimental test rig, containing the inclined plate and the landslide block. The test rig could be used in different modes, including having a combined uplift followed by a landslide. In this chapter we consider the case when the uplift plate is fixed, in its elevated position and there is landslide only, down the continual slope which is seen in figure 6.1. Note that the wave gauge numbering system used in this section is not that used in the experiments, which is shown in figure 6.1. Instead we use the wave gauge numbering system shown
6.3 SPH Simulation of Tsunami Waves Generated by a Moving Landslide on a Fixed Inclined Plate

In this work we use the SPH model described to simulate the flow that occurs in experimental models of tsunami waves generated by a landslide. For the SPH simulations of the landslide, the SPH model was similar to that for the flat uplift plate (Chapter 4, section 4.2.2). The differences were in the boundary condition at the location of the plate, which is now a slope, and the inclusion of a landslide block which slides down the slope. As seen in figure 6.1, the shape of the landslide block used in the experiment has a curved top surface, and effectively points at each end. This shape is not easy to model using SPH boundary conditions. Therefore, due to the limited time available, we consider the simpler shape for the landslide block, which is a rectangle. Therefore, the SPH simulations in this section will not be compared directly with the experimental data. The aim of this work is to effectively show that the SPH code used and developed for the work in this thesis, can be adapted to model a landslide.

The SPH model consists of a truncated horizontal water tank, with a landslide block located on the fixed inclined plate at the upstream end of the tank. The landslide block is
6.3. SPH SIMULATION OF TSUNAMI WAVES GENERATED BY A MOVING LANDSLIDE ON A FIXED INCLINED PLATE

Figure 6.2: Schematic diagram of the SPH model showing the fault rupture (inclined plate) and landslide model.

0.3 m long and starts near the top of the inclined plate, which is 0.853 m long. Two different heights of landslide block have been considered, 0.018 m and 0.036 m. The water depth is 0.37 m. The tsunami wave is generated by releasing the landslide, on the fixed inclined plate, where it moves downwards with a recorded experimental velocity. The SPH simulations used approximately 120,000 particles, with a particle spacing of 0.004 m. The distance between each wave gauge is 0.3 m, with the first located at 0.6 m along the tank. A slip boundary condition is used at the end wall in the SPH simulations.

Figure 6.3 shows the use of fixed ghost particles to simulate the moving landslide on top of the fixed inclined plate, with a different colour to represent each type of fixed ghost particles. Note that we use the term “fixed ghost particles”, as the properties of these particles are calculated using the method described in section 1.14. However, these “fixed ghost particles” are not fixed as they represent the landslide block, which moves down the slope. The blue, green and orange particles in figure 6.3 represent the block in the SPH method. The properties of the blue coloured ghost particles will be determined from the red coloured interpolation points. The properties of the green coloured ghost particles in the corners, will be determined from the yellow coloured interpolation points. The properties of the lower particles on the left and right walls of the landslide block, which are orange coloured, will be determined from the purple coloured interpolation points.

Figures 6.4 and 6.5 show the SPH predictions of surface height at the first three wave gauges. Figure 6.4 is for the 0.018 m landslide block height and figure 6.5 is for the 0.036 m landslide block height. In the SPH simulations the landslide starts moving at $t = 1 \text{s}$, so still water is simulated from $t = 0 \text{s}$ to $t = 1 \text{s}$ in the SPH model, which enables the
SPH particles to settle into a natural configuration before the plate starts moving. Note that initially, at $t = 0$ s, the SPH particles are positioned on a regular grid, which is not a natural configuration for the SPH particles. The changes in surface height from $t = 0$ s to $t = 1$ s that are seen in figures 6.4 and 6.5 are due to particles settling to a natural configuration, as the landslide block does not start moving until $t = 1$ s. These changes in surface height are small, and less than the spacing between particles, $dx = 0.004$ m. The method for determining the free surface location is described in section 4.2.2, and as the particles settle to a natural configuration this leads to the small changes in the calculated surface elevation that are observed in figures 6.4 and 6.5. The surface elevation profiles in these figures are very different in shape than those obtained for the wave generated by an uplift plate, either the flat plate in figures 4.11 and 4.12 or the inclined case in figures 5.6 and 5.7. Figure 6.6 shows the experimental measurements of surface height at the first three wave gauges. As previously described, the shape of the landslide block in the simulations is rectangular, and so different to that in the experiments. It is therefore not appropriate to make direct quantitative comparison between the SPH predictions and experimental measurements of the surface height. However, from figures 6.4 and 6.6, it can be seen that the general variation in the surface elevation profiles at the wave gauges, is similar for both the SPH predictions and the experiment measurements. Note that the labelling system used for the wave gauges in figure 6.6 is consistent with that used in in figures 6.4
and 6.5.

**Figure 6.4:** SPH predictions of surface height at the three different wave gauges for the 0.018m landslide height.

**Figure 6.5:** SPH predictions of surface height at the three different wave gauges for the 0.036m landslide height.

**Figure 6.6:** Experimental measurements of surface height at the three different wave gauges for the 0.018m landslide height.
Figures 6.7 to 6.9 show a direct comparison between the SPH predictions, for both landslide block heights (0.018 m and 0.036 m) cases, of surface height at the first three wave gauges. There is agreement between the time at which the peaks and troughs occur for the two different landslide block height cases. The differences in the peak height and the trough depth is according to the landslide block height. That is, as expected, the larger landslide block height (0.036 m) leads to having a higher peak and a deeper trough.

Figures 6.10 and 6.11 show the moving landslide (0.018 m height) position, coloured in black, at various times, as it moves down the inclined plate. Note that the landslide moves downwards from $t = 1$ s to $t = 1.8$ s. In figures 6.10 and 6.11 it is seen that the initial hydrostatic pressure that was set at $t = 0$ s is maintained at $t = 1$ s and that there is an increase in pressure above the landslide when it moves downwards at $t = 1.2$ s. The movement of the landslide block affects the pressure solution further downstream, but at $t = 3$ s, some time after the landslide block has stopped moving, the pressure settles to an approximately hydrostatic solution. Figures 6.12 and 6.13 show the moving landslide position for the landslide block of 0.036 m height. It is seen that there is a similar behaviour in the pressure solution, to that obtained for the 0.018 m height case. These figures also show the shape of the wave and how it travels along the wave tank as time progresses. Figure 6.14 shows the SPH pressure solution near the moving landslide block at time $t = 1.4$ s, after it has started moving downwards. It is seen that, there is a large pressure around the landslide block.

Figure 6.7: Comparison between SPH predictions of surface height at WG1 for the 0.018 m and 0.036 m landslides.
6.3. SPH SIMULATION OF TSUNAMI WAVES GENERATED BY A MOVING LANDSLIDE ON A FIXED INCLINED PLATE

Figure 6.8: Comparison between SPH predictions of surface height at WG2 for the 0.018m and 0.036m landslides.

Figure 6.9: Comparison between SPH predictions of surface height at WG3 for the 0.018m and 0.036m landslides.
Figure 6.10: SPH pressure solution at various times after the 0.018 m landslide has moved downwards.
6.3. SPH SIMULATION OF TSUNAMI WAVES GENERATED BY A MOVING LANDSLIDE ON A FIXED INCLINED PLATE

Figure 6.11: SPH pressure solution at various times after the 0.018 m landslide has moved downwards.
6.3. SPH SIMULATION OF TSUNAMI WAVES GENERATED BY A MOVING LANDSLIDE ON A FIXED INCLINED PLATE

Figure 6.12: SPH pressure solution at various times after the 0.036 m landslide has moved downwards.
6.3. SPH SIMULATION OF TSUNAMI WAVES GENERATED BY A MOVING LANDSLIDE ON A FIXED INCLINED PLATE

Figure 6.13: SPH pressure solution at various times after the 0.036 m landslide has moved downwards.

Figure 6.14: SPH pressure solution after the 0.036 m landslide has moved downwards.
6.4 Conclusions

In this chapter, we used the SPH method to simulate the flow that occurs in experimental models of tsunamis generated by a landslide, by using a moving landslide block on a fixed inclined plate. Two different heights of landslide block (0.018 m and 0.036 m) have been simulated and compared with each other. Relative to when the landslide starts moving downwards, the surface height rises at the same time for both the 0.018 m and 0.036 m landslide height. However, the comparison of the two different landslide block height cases show that the peak height and the trough depth in the surface elevation profiles, are dependent on the height of the landslide block. That is, the larger landslide block height (0.036 m) produces a higher peak and a deeper trough.

In the limited time available, it was not possible to produce an SPH model for the shape of landslide block used in the experiments. However, these SPH simulations have shown that it is possible to adapt the SPH code, developed for the work in this thesis, to model a landslide.
Chapter 7

Summary of The Project

7.1 Conclusions

This chapter represents an overview of this research by providing the main achievements, as well as summarising the future work in SPH, that could be carried out, in simulating tsunami waves and also considering the effect of boundary conditions on the solution. A substantial amount of time was spent investigating different boundary condition methods, that would be used in the SPH code, which was written specifically for the simulations carried out in this project. This thesis has considered a new boundary treatment for the density using the smoothed particle hydrodynamics (SPH) method. Three different types of boundary conditions are used: the fixed boundary particle technique, the ghost boundary particle technique and the fixed ghost boundary particle technique. Numerical SPH results show that there is no difference between the fixed and ghost boundary particle techniques, but the fixed ghost boundary particle technique proved to give more reasonable results, especially when treating a complex geometry that included slopes, not aligned with the coordinate system, and an internal $75^\circ$ corner within the flow. The main advantage of using the fixed ghost boundary particles technique is that it produced a stable and accurate pressure solution in the geometry described.

In this work, we have presented lid-driven cavity flow WCSPH simulations at different Reynolds numbers, with uniform and staggered particle spacing. Numerical results give good agreement with Ghia et al. (1982) data for both types of particle configuration. Adding background pressure, solved the problem of a hole appearing in the particles, which appeared at the centre of the cavity when using the evolved density scheme. At the boundary condition, by setting the density of the dummy particles to be the same
7.1. CONCLUSIONS

as that of the solid wall particles, so effectively setting a zero normal density gradient, solved the problem of velocity oscillations in the simulation. This gives more accurate results for uniform particle spacing than with the staggered spacing. Finally, velocity profile plots show that by increasing the number of particles, the results become more accurate. Furthermore, the use of the Shepard filter technique for density correction will control the density fluctuation and give a more accurate solution. For the dam break flow, both uniform and staggered spacing give good predictions of the leading edge location and the rate of collapse of the water column by comparing the results with the numerical models represented by Hughes & Graham (2010), Greaves (2006) and the experimental data of Martin & Moyce (1952).

The main work of this research is to use the SPH method, with the fixed ghost boundary particle technique, to simulate the flow that occurs in the experimental models of tsunami waves which are generated by an earthquake fault rupture or a landslide. In the experiment, two different models of an earthquake fault rupture were considered: a flat uplift plate and an inclined uplift plate. The SPH model, for both the flat uplift plate and the inclined slope uplift, gives generally good predictions of the free surface position. At each wave gauge location, there is good agreement between the SPH predictions and experimental measurements of the peak surface height and also the time it takes for the free surface height to increase and then return to the zero crossing height. There is also agreement between the calculated wave speeds from the SPH simulations and the experiments. However, relative to when the plate starts moving upwards, the surface height rises more quickly in the SPH simulations than in the experiments and the trough depths are under predicted by the SPH model of the flat uplift plate, while it is more in agreement for the inclined slope uplift model. We believe that these differences are due to some water being able to move around and below the moving plate in the experiments, whereas in the SPH model all of the water is always above the plate. The SPH simulations also show that, as expected, using a higher particle resolution decreases the noise in the predicted free surface location.

The validated SPH model has been used to consider the effect of plate uplift time and water depth on the surface elevation. It was shown that the surface elevation profiles, par-
particularly at the downstream wave gauge locations, vary more with changes in water depth than with changes in plate uplift time. It is noted that the uplift plate displacement was constant, at 0.06 m, so as the depth of water changed, the displacement to depth ratio also changed.

It has been demonstrated that a moving landslide can be simulated, for a simple shape landslide, i.e. a rectangle, though in the time available it was not possible to produce an SPH model of the actual landslide block used in the experiment. The free surface elevation comparison shows that the differences in the peak height and the trough depth of the surface elevations is dependent on the landslide height. That is, the higher landslide leads to having a higher peak and a deeper trough.

7.2 Future Work

The numerical results obtained in this thesis have shown that SPH can be used to simulate a model-scale tsunami waves that are generated by either a fault rupture or a landslide. Therefore, the knowledge gained from this approach will be useful to include the following possible future work:

- Develop an SPH model of the moving landslide for the experimental landslide mass shape, which has a curved top surface, and effectively points at the end. This shape is not easy to do model using SPH boundary condition methods.

- Develop a parallelised version of the SPH code so that the full length of the experimental tank could be simulated, rather than using truncated numerical tank.

- Use the fluid-structure interaction model, in the forces exerted on the moving landslide by the fluid are used to predict its velocity rather than reading the recorded velocity from the experimental data. This fluid-structure interaction model would take into account the mass of the landslide and the friction between the moving landslide and also the inclined plate.

- Develop this research by considering different shapes and sizes of landslide and investigate the effect of these parameters on the free surface elevation.
• Simulate the experiments in which the landslide block is located on the inclined up-lift plate, and this plate is first moves upwards before the landslide block is released. These experiments model are combined fault rupture and landslide.

7.3 Notation

\( P = \) Pressure.
\( \rho = \) Density.
\( u = \) Velocity vector.
\( \Delta t = \) Time step.
\( m = \) Particle mass.
\( h = \) Smoothing length.
\( dx, dy = \) Particle spacing.
\( u = \) Velocity in \( x \)-direction.
\( v = \) Velocity in \( y \)-direction.
\( r = \) Particle position vector.
\( F_a = \) Acceleration.
\( W_{ab} = \) SPH interpolation function.
\( L = \) Cavity length.
\( u_{tid} = \) Top velocity.
\( \nu = \) Kinematic viscosity.
\( \mu = \) Dynamic viscosity.
\( C_s = \) Speed of sound.
\( Re = \) Reynolds number.
\( g = \) Gravity.
\( H = \) Dam length.
\( W = \) Dam width.
\( D = \) Dam depth.
\( WG = \) Wave gauge.
List of references


Dalrymple, R. A. (2007), ‘Particle methods and waves, with emphasis on SPH’, *Department of Civil Engineering, Baltimore* p. 32.


Verlet, L. (1967), ‘Computer” experiments” on classical fluids. i. thermodynamical properties of lennard-jones molecules’, *Phys. Rev.* **159**(1), 98.

