

Numerical Modelling of Viscous and Turbulent Free-Surface Flows using Smoothed Particle Hydrodynamics

Alice Catherine Ely

Department of Civil and Environmental Engineering
Imperial College London

Thesis submitted for the degree of Doctor of Philosophy

Declaration of originality

All the work presented herein is that of the author.
Any work or contributions made by others is referenced appropriately.

Copyright declaration

The copyright of this thesis rests with the author and is made available under a Creative Commons Attribution Non-Commercial No Derivatives licence. Researchers are free to copy, distribute or transmit the thesis on the condition that they attribute it, that they do not use it for commercial purposes and that they do not alter, transform or build upon it. For any reuse or redistribution, researchers must make clear to others the licence terms of this work.

Abstract

This thesis presents the formulation of a Smoothed Particle Hydrodynamics (SPH) model and its application to a range of engineering applications. The motivation for this research lies in the desire to accurately model viscous and turbulent free-surface flows, including those with complete break-up of the free-surface. At present, boundary element modelling is typically chosen to describe free-surface flows where viscous effects are not important. The Volume of Fluid method is able to model most flow phenomena, but the representation of the free-surface is insufficient for the most complex flows. Current SPH models have shown aptitude for modelling such flows, but there is a noticeable lack of validation carried out in the literature. This thesis includes a thorough investigation into established modelling techniques, extending or developing new techniques where necessary, in order to create a versatile and accurate SPH model for free-surface flows. Where possible, quantitative comparisons with experimental observations have been carried out to ensure a suitable level of accuracy has been achieved.

First, a fairly basic SPH model is constructed through testing its ability to generate and propagate solitary waves in a numerical wave flume. This is succeeded by a thorough investigation into solitary waves breaking on a 5° slope, through which further developments are added to the SPH model. The full process, including overturning, post-breaking behaviour, run-up, and the subsequent hydraulic jump are quantitatively compared with experimental measurements.

The work carried out in this thesis shows that the SPH model can successfully capture violent free-surface flows with large deformations from the initial surface geometry. Validation studies demonstrate that SPH can form an important part of model testing for engineering developments involving these types of flows.

Acknowledgements

First, I would like to thank my supervisor, Professor Chris Swan for suggesting such an interesting area of research, then providing support, guidance and insight throughout my studies. When I started working on my PhD, I initially needed some persuasion of the importance in validating numerical models to such a high level of detail. Having carried out the work for this thesis and through many discussions with Chris Swan and my fellow research students, this has been thoroughly installed in me and is something I will always aspire to in my current and future areas of work. I am also very grateful for the Doctoral Training Grant provided by the UK Engineering and Physical Sciences Research Council (EPSRC).

I would also like to thank everyone I met at the SPHERIC workshops. The opportunity to attend these workshops on two occasions provided me with inspiration, whilst allowing me to keep up with the latest developments in SPH modelling. It was also highly beneficial to meet so many others who share an enthusiasm for SPH modelling and the desire to exploit the methods full potential.

I wish to thank all the people who I shared the laboratory offices with, who provided me with a motivating and fun environment to work in. In particular, I would like to thank Marios, for so many useful and interesting discussions, and support with many technical matters. I am also very grateful to Stuart for his assistance with both programming and hardware related problems, along with his enthusiasm for progressing the computational side of the numerical models. I would also like to thank Johannes for his help in the laboratory. Thanks are also due to Rebecca, who has been immensely helpful over the years, and to the laboratory technicians.

Finally, but by no means least, I would like to thank my friends and family, who have been extremely supportive and patient throughout the duration. In particular, Pete has been amazing throughout the entire time and I honestly could not have done this without him. Likewise, thanks go to my colleagues at HR Wallingford, who were always very encouraging and supportive. I would also like to thank everyone who helped proof read my thesis – Pete, Matt, Jo, Jenny, Dad and Graham. This was greatly appreciated, especially at such short notice.

Contents

Declaration of originality	3
Copyright declaration	4
Abstract	5
Acknowledgements	6
Contents	7
List of Figures	11
List of Tables	22
1 Introduction	23
1.1 Motivations for research	24
1.2 Aims of the project	25
1.3 Outline of thesis	26
1.4 Principal achievements	27
2 Background	28
2.1 Review of numerical methods for free-surface flows	28
2.1.1 Analytical solutions and early numerical models for free-surface flows	28
2.1.2 Boundary Element Method	30
2.1.3 Numerical methods for two-phase flows	31
2.1.4 Meshless methods	33
2.2 Review of Smoothed Particle Hydrodynamics	34

2.2.1	Origins of SPH	34
2.2.2	An initial model for free-surface flows	35
2.2.3	Applications and further developments	37
2.2.4	Concluding remarks	45
3	Numerical formulation of Smoothed	
	Particle Hydrodynamics (SPH)	47
3.1	Basis for SPH	48
3.2	Governing equations	53
3.3	The equation of state	55
3.4	Viscosity	56
3.4.1	Viscosity model tests	58
3.5	Turbulence	77
3.5.1	Turbulence model tests	79
3.6	XSPH	83
3.7	Density smoothing techniques	85
3.8	SPH boundary representations	87
3.9	A new hybrid boundary formulation	92
3.9.1	The collapsing dam test case	96
3.10	The turbulent boundary condition	103
3.11	Time marching	104
3.12	Computational considerations	107
3.13	Conservation properties of SPH	109
3.14	Errors and accuracy in SPH	110
3.15	Other modelling techniques	111
3.16	Weakly compressible versus incompressible formulation	113
4	An SPH study of solitary wave propagation	116
4.1	Introduction	116
4.2	Background	117
4.2.1	Solitary wave theory	117
4.2.2	Solitary wave generation	120
4.2.3	Solitary wave propagation	122

4.3	Numerical simulations	124
4.3.1	SPH model	124
4.3.2	SPH wave gauges	125
4.4	Optimising the accuracy of the SPH model	125
4.4.1	Basic SPH model	125
4.4.2	Model testing	128
4.5	Comparison with established theories	145
4.5.1	$H/d = 0.3$	145
4.5.2	Varying H/d	152
4.6	Comparison with experimental observations	163
4.7	Effects of bed shear on the dissipation of propagating solitary waves . .	166
4.8	Conclusions	170
5	An SPH study of solitary waves breaking on a plane beach: quanti-	
	tative comparisons to laboratory observations	173
5.1	Introduction	173
5.2	Background	175
5.2.1	Solitary wave breaking criteria and run-up on slopes	175
5.2.2	Overturning solitary waves	178
5.2.3	Post-breaking behaviour	182
5.2.4	SPH solitary wave studies	184
5.3	Numerical simulations	186
5.3.1	Introduction	186
5.3.2	Turbulence modelling in SPH	187
5.3.3	The turbulent boundary condition	188
5.3.4	SPH model	189
5.4	Experimental investigations	192
5.4.1	Laboratory apparatus	192
5.4.2	Wave generation	193
5.4.3	Quantitative flow visualisation	194
5.5	Discussion of results	197
5.5.1	Preliminary results	197

5.5.2	Final model results	207
5.6	Conclusions	237
6	Conclusions and further work	241
6.1	Principal achievements	241
6.2	Engineering significance and practical implications	244
6.3	Suggestions for further work	244
	References	250

List of Figures

3.1	Influence domain for an SPH particle.	51
3.2	Numerical setup of the channel used to demonstrate Poiseuille flow and Couette flow.	61
3.3	Time varying Poiseuille flow tending towards a steady-state, comparisons between results obtained with the SPH model using the <i>Realistic</i> viscosity method and theory.	62
3.4	Time varying Poiseuille flow tending towards a steady-state, comparisons between results obtained with the SPH model using the <i>Artificial</i> viscosity method and theory.	63
3.5	Time varying Poiseuille flow tending towards a steady-state, comparisons between results obtained with the SPH model using the <i>Improved Artificial</i> viscosity method and theory.	64
3.6	Shear stress at the wall in time varying Poiseuille flow tending towards a steady-state, comparisons between results obtained with the SPH model using the <i>Realistic</i> viscosity method, the <i>Artificial</i> viscosity method, the <i>Improved Artificial</i> viscosity method and theory.	66
3.7	Time varying Couette flow tending towards a steady-state, comparisons between results obtained with the SPH model using the <i>Realistic</i> viscosity method and theory.	68
3.8	Time varying Couette flow tending towards a steady-state, comparisons between results obtained with the SPH model using the <i>Artificial</i> viscosity method and theory.	69
3.9	Time varying Couette flow tending towards a steady-state, comparisons between results obtained with the SPH model using the <i>Improved Artificial</i> viscosity method and theory.	70

3.10	Solution for $u_{\max}$ (Poiseuille Flow) obtained with varying SPH particle numbers across the width of the channel. $u_{\max}$ is plotted against particle number.	72
3.11	Solution for the velocity residual in $u_{\max}$ (Poiseuille Flow) obtained with varying SPH particle numbers across the width of the channel. The error in $u_{\max}$ compared to the exact solution is plotted against the initial particle spacing, Δx	73
3.12	Time varying Poiseuille flow tending towards a steady-state, comparisons between results obtained with the SPH model including a turbulence model and theory.	80
3.13	Time varying Couette flow tending towards a steady-state, comparisons between results obtained with the SPH model including a turbulence model and theory.	81
3.14	Steady-state turbulent plane channel flow at $Re \approx 500,000$; comparisons between the theoretical laminar solution (matched to u_{max}) and the results of the SPH model including a turbulence model.	82
3.15	Log profile of steady-state turbulent plane channel flow velocity with $Re \approx 500,000$, obtained with the SPH model including a turbulence model.	83
3.16	Open channel flow velocity profile with $Re \approx 220,000$, obtained using the SPH model with an effective turbulence model.	84
3.17	Pressure distribution associated with a propagating solitary wave: (a) without density smoothing and (b) with density smoothing.	85
3.18	Boundaries modelled with a repulsive force condition showing the peeling away of the fluid from the boundary close to the water surface.	89
3.19	SPH particle positions arising in the initial stages of a dam break problem. When water flows across a previously dry boundary the <i>Stationary Ghosts</i> boundary method leads to a gap developing between the bottom boundary and the water flow above.	93
3.20	SPH particle positions arising some time after a dam break problem. When the water level on the rear containing wall reduces, and the boundary effectively dries out, use of the <i>Stationary Ghosts</i> boundary method leaves water particles stuck to the left wall.	94

3.21	Numerical setup and progression of a collapsing dam.	97
3.22	Progression of the reduction in height of the remaining dam with time, normalised to the initial dam height.	99
3.23	SPH particle positions arising some time after a dam break problem. When the water level on the rear containing wall reduces, and the bound- ary effectively dries out, the <i>Hybrid</i> method provides a better qualita- tive representation of the water adjacent to the wall than the <i>Stationary</i> <i>Ghosts</i> boundary method.	100
3.24	Progression of the movement of the surge front, Z , over the dry bed with time, normalised to the initial dam width.	101
3.25	SPH particle positions arising in the initial stages of a dam break problem. When water flows across a previously dry boundary the <i>Repulsive Force</i> boundary method provides a better qualitative representation of the water immediately above the bed than the <i>Stationary Ghosts</i> boundary method.	102
3.26	SPH particle positions arising in the initial stages of a dam break problem. When water flows across a previously dry boundary, initially the <i>Hybrid</i> boundary method provides a better qualitative representation of the water immediately above the bed than the <i>Stationary Ghosts</i> boundary method.	103
3.27	Background grid for particle pair searching.	108
4.1	Numerical wave gauge.	126
4.2	Solitary wave propagation, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of numerical wave gauges using the <i>basic</i> SPH model.	127
4.3	Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons be- tween results based on different boundary models.	131
4.4	Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons be- tween results based on different viscosity models.	132
4.5	Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons be- tween results with and without the inclusion of the XSPH correction.	133

4.6	Predictions of the pressure field associated with an open channel flow: (a) no density smoothing applied and (b) Shepard filter applied, giving a hydrostatic pressure distribution.	135
4.7	Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons be- tween results based on different density smoothing techniques.	135
4.8	Predictions of SPH particle distribution: (a) cubic kernel, causing particle clumping, where particles move around in pairs and (b) quartic kernel, allowing particles to move around as individuals.	136
4.9	Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons be- tween results using different kernels.	137
4.10	Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons be- tween results based on different time stepping schemes.	138
4.11	Spatial variations in the wave heights over the length of the computational domain.	139
4.12	Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons be- tween results using different SPH particle resolutions.	141
4.13	Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons be- tween results produced with the <i>optimum</i> model and Boussinesq's first order solitary wave theory.	142
4.14	Solitary wave propagation, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of numerical wave gauges using the <i>optimum</i> SPH model.	143
4.15	Solitary wave propagation with $H/d = 0.3$, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of four numerical wave gauges, comparisons between results produced with the <i>optimum</i> SPH model and Boussinesq's first order solitary wave theory.	146
4.16	Water surface elevation, $\eta(x)$, predicted at four times during solitary wave propagation with $H/d = 0.3$, comparisons between results produced with the <i>optimum</i> SPH model and Boussinesq's first order solitary wave theory. 147	

4.17	Horizontal water particle velocities, $u(z)$, under the wave crest, predicted at four times during solitary wave propagation with $H/d = 0.3$, comparisons between results produced with the <i>optimum</i> SPH model and Boussinesq's first order solitary wave theory.	148
4.18	Solitary wave propagation with $H/d = 0.3$, a time-history of the water surface elevation, $\eta(t)$, predicted at $x = 18\text{m}$, comparisons between results produced with the <i>optimum</i> SPH model, Boussinesq's first order theory, McCowan's first order theory and Laitone's second order theory.	149
4.19	Solitary wave propagation with $H/d = 0.3$, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of four numerical wave gauges, comparisons between results produced with the <i>optimum</i> SPH model and McCowan's first order solitary wave theory.	150
4.20	Water surface elevation, $\eta(x)$, predicted at four times during solitary wave propagation with $H/d = 0.3$, comparisons between results produced with the <i>optimum</i> SPH model and McCowan's first order solitary wave theory.	151
4.21	Horizontal water particle velocities, $u(z)$, under the wave crest, predicted at four times during solitary wave propagation with $H/d = 0.3$, comparisons between results produced with the <i>optimum</i> SPH model and McCowan's first order solitary wave theory.	152
4.22	Solitary wave propagation with $H/d = 0.1$, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of four numerical wave gauges, comparisons between results produced with the <i>optimum</i> SPH model and McCowan's first order solitary wave theory.	153
4.23	Water surface elevation, $\eta(x)$, predicted at four times during solitary wave propagation with $H/d = 0.1$, comparisons between results produced with the <i>optimum</i> SPH model and McCowan's first order solitary wave theory.	154
4.24	Horizontal water particle velocities, $u(z)$, under the wave crest, predicted at four times during solitary wave propagation with $H/d = 0.1$, comparisons between results produced with the <i>optimum</i> SPH model and McCowan's first order solitary wave theory.	155

4.25	Solitary wave propagation with $H/d = 0.2$, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of four numerical wave gauges, comparisons between results produced with the <i>optimum</i> SPH model and McCowan's first order solitary wave theory.	156
4.26	Water surface elevation, $\eta(x)$, predicted at four times during solitary wave propagation with $H/d = 0.2$, comparisons between results produced with the <i>optimum</i> SPH model and McCowan's first order solitary wave theory.	157
4.27	Horizontal water particle velocities, $u(z)$, under the wave crest, predicted at four times during solitary wave propagation with $H/d = 0.2$, comparisons between results produced with the <i>optimum</i> SPH model and McCowan's first order solitary wave theory.	158
4.28	Horizontal water particle velocities, $u(z)$, under the wave crest, predicted at four times during solitary wave propagation with $H/d = 0.2$, comparisons between results produced with the <i>optimum</i> SPH model and McCowan's first order solitary wave theory.	159
4.29	Water surface elevation, $\eta(x)$, predicted at four times during solitary wave propagation with $H/d = 0.45$, comparisons between results produced with the <i>optimum</i> SPH model and McCowan's first order solitary wave theory.	160
4.30	Horizontal water particle velocities, $u(z)$, under the wave crest, predicted at four times during solitary wave propagation with $H/d = 0.45$, comparisons between results produced with the <i>optimum</i> SPH model and McCowan's first order solitary wave theory.	161
4.31	Solitary wave propagation with $H/d = 0.1$ and $H/d = 0.2$, a time-history of the water surface elevation, $\eta(t)$, predicted at $x = 18\text{m}$, comparisons between results produced with the <i>optimum</i> SPH model, Boussinesq's first order theory, McCowan's first order theory and Laitone's second order theory.	162
4.32	Solitary wave propagation with $H/d = 0.3$, time histories of the water surface elevation, $\eta(t)$, predicted at two spatial locations, comparisons between results produced with the <i>optimum</i> SPH model and laboratory observations.	164

4.33	Solitary wave propagation with $H/d = 0.3$, time-history of the water surface elevation, $\eta(t)$, predicted at $x = 11\text{m}$, comparisons between results produced with the <i>optimum</i> SPH model, laboratory observations, Boussinesq's first order theory and McCowan's first order theory.	165
4.34	Solitary wave propagation, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of numerical wave gauges using the <i>optimum</i> SPH model with comparison to the theoretical attenuation.	168
5.1	Diagram of the numerical wave flume, showing the location and angle of the slope, the wave generating paddle and the location of a numerical wave gauge.	190
5.2	Initial layout of SPH particles above the toe of the bed slope, the Cartesian grid being parallel to the flat bed on the left of the toe of the slope and parallel to the bed slope on the right of the toe of the slope.	191
5.3	Results arising from the quantitative flow visualisation applied to an overturning solitary wave: (a) An example photograph taken in the laboratory study, indicating the high quality of the images and (b) the defined water surfaces, including the air void, based on the image given in (a) above.	196
5.4	Time histories of the water surface elevation, $\eta(t)$, during solitary wave propagation with $H/d = 0.3$. Comparison between laboratory observations and initial SPH predictions based upon the <i>optimum</i> model at the wave gauge located 0.15m before the toe of the slope ($x = 0$).	199
5.5	Time histories of the water surface elevation, $\eta(t)$, during solitary wave propagation with $H/d = 0.3$. Comparison between laboratory observations and SPH predictions, with improvements to the model, at the wave gauge located 0.15m before the toe of the slope ($x = 0$).	200
5.6	Visualisation of a solitary wave with $H/d = 0.3$ at the breaking point. Comparison between results produced with an initial SPH simulation and laboratory observations.	201
5.7	Visualisation of a solitary wave with $H/d = 0.3$ at the breaking point. Comparison between results produced with an SPH simulation, with improvements to the model, and laboratory observations.	202

5.8	Visualisation of a solitary wave with $H/d = 0.3$ as the wave starts to overturn. Comparison between results produced with an initial SPH simulation and laboratory observations.	203
5.9	Visualisation of a solitary wave with $H/d = 0.3$ as the wave starts to overturn. Comparison between results produced with an SPH simulation, with improvements to the model, and laboratory observations.	203
5.10	Visualisation of a solitary wave with $H/d = 0.3$ as the overturning jet nears impact with the bed slope ahead. Comparison between results produced with an initial SPH simulation and laboratory observations. .	205
5.11	Visualisation of a solitary wave with $H/d = 0.3$ as the overturning jet nears impact with the bed slope ahead. Comparison between results produced with an SPH simulation, with improvements to the model, and laboratory observations.	205
5.12	Time histories of the water surface elevation, $\eta(t)$, during solitary wave propagation with $H/d = 0.3$. Comparisons between laboratory observations and high resolution SPH predictions at a wave gauge located 0.15m before the toe of the slope.	208
5.13	Visualisation of a solitary wave with $H/d = 0.3$ at the breaking point. Comparison between results produced with a high resolution SPH simulation and laboratory observations.	209
5.14	Visualisation of a solitary wave with $H/d = 0.3$ at breaking point: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations which have been spatially aligned with the SPH results.	210
5.15	Visualisation of a solitary wave with $H/d = 0.3$ as the wave starts to overturn: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations.	212
5.16	Visualisation of a solitary wave with $H/d = 0.3$ as the overturning jet nears impact with the bed slope ahead: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations.	213

5.17	Visualisation of a solitary wave with $H/d = 0.3$ as the wave starts to impact with the bed slope: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations.	214
5.18	Visualisation of a solitary wave with $H/d = 0.3$ as the flow begins to run up the bed slope: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations.	216
5.19	Visualisation of a solitary wave with $H/d = 0.3$ as the flow continues to run up the bed slope: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations.	217
5.20	Visualisation of the hydraulic jump resulting from a solitary wave with $H/d = 0.3$ washing back down the bed slope: (a) photograph of the experiment, (b) laboratory observations, and (c) results produced with a high resolution SPH simulation.	220
5.21	Time histories of the water surface elevation, $\eta(t)$, during solitary wave propagation with $H/d = 0.45$. Comparisons between laboratory observations and high resolution SPH predictions at a wave gauge located 0.15m before the toe of the bed slope.	221
5.22	Visualisation of a solitary wave with $H/d = 0.45$ at the breaking point. Comparison between results produced with a high resolution SPH simulation and laboratory observations.	222
5.23	Visualisation of a solitary wave with $H/d = 0.45$ at the breaking point: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations which have been spatially aligned with the SPH results.	223
5.24	Visualisation of a solitary wave with $H/d = 0.45$ as the wave starts to overturn: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations.	224

5.25	Visualisation of a solitary wave with $H/d = 0.45$ as the overturning jet nears impact with the bed slope ahead: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations.	226
5.26	Visualisation of a solitary wave with $H/d = 0.45$ as the wave starts to impact with the bed slope: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simu- lation and laboratory observations.	227
5.27	Visualisation of a solitary wave with $H/d = 0.45$ as the cavity starts to fill with water: (a) photograph of the experiment and (b) compari- son between results produced with a high resolution SPH simulation and laboratory observations.	228
5.28	Visualisation of a solitary wave with $H/d = 0.45$ as the flow begins to run up the bed slope: (a) photograph of the experiment and (b) compari- son between results produced with a high resolution SPH simulation and laboratory observations.	229
5.29	Visualisation of a solitary wave with $H/d = 0.45$ as the flow continues to run up the bed slope: (a) photograph of the experiment and (b) compari- son between results produced with a high resolution SPH simulation and laboratory observations.	230
5.30	Visualisation of the hydraulic jump resulting from a solitary wave with $H/d = 0.45$ washing back down the bed slope: (a) photograph of the experiment, (b) laboratory observations and (c) results produced with a high resolution SPH simulation.	232
5.31	Velocity vector plots of a solitary wave with $H/d = 0.3$ at three times when the wave is close to the breaking point, or the point at which the wave face becomes vertical: (a) 0.08s before reaching the breaking point, (b) at the breaking point and (c) 0.08s after reaching the breaking point.	233
5.32	Velocity vector plots of a solitary wave with $H/d = 0.45$ at three times when the wave is close to the breaking point, or the point at which the wave face becomes vertical: (a) 0.08s before reaching the breaking point, (b) at the breaking point and (c) 0.08s after reaching the breaking point.	234

5.33	Velocity vector plots (left column) and pressure plots (right column) of a solitary wave with $H/d = 0.3$ at four times during breaking: (a) velocity vectors at $t = 0.00s$, (b) velocity vectors at $t = 0.14s$, (c) velocity vectors at $t = 0.23s$, (d) velocity vectors at $t = 0.30s$, (e) pressure at $t = 0.00s$, (f) pressure at $t = 0.14s$, (g) pressure at $t = 0.23s$ and (h) pressure at $t = 0.30s$. Times are relative to breaking point.	236
5.34	Velocity vector plots (left column) and pressure plots (right column) of a solitary wave with $H/d = 0.45$ at four times during breaking: (a) velocity vectors at $t = 0.00s$, (b) velocity vectors at $t = 0.16s$, (c) velocity vectors at $t = 0.26s$, (d) velocity vectors at $t = 0.36s$, (e) pressure at $t = 0.00s$, (f) pressure at $t = 0.16s$, (g) pressure at $t = 0.26s$ and (h) pressure at $t = 0.36s$. Times are relative to breaking point.	237
5.35	Velocity vector plot of a solitary wave with $H/d = 0.3$ during the hydraulic jump.	238
5.36	Velocity vector plot of a solitary wave with $H/d = 0.45$ during the hydraulic jump.	238

List of Tables

3.1	Maximum velocity and percentage error relative to the theoretical predictions for steady-state Poiseuille flow simulations with varying SPH particle resolutions.	71
3.2	Results of the convergence analysis carried out on two sets of three simulations considering the maximum velocity obtained during steady-state Poiseuille flow.	75
4.1	Modelling techniques used in a case study for the development of the SPH model.	130
4.2	runtimes for all cases relative to <i>Case 1</i>	140
4.3	Properties of the propagating wave form, a comparison between the 11 test cases and first order Boussinesq solitary wave theory.	144
4.4	A comparison of the wave attenuation rate and phase speed between two gauge locations comparing both SPH and experiment to theory.	169
5.1	Maximum run-up on a 5° slope for a solitary wave with $H/d = 0.3$, comparison between results produced with a high resolution SPH simulation and laboratory observations.	218
5.2	Maximum run-up on a 5° slope for a solitary wave with $H/d = 0.45$, comparison between results produced with a high resolution SPH simulation and laboratory observations.	231

1

Introduction

Furthering our knowledge of fluid dynamics in the ocean, coastal areas and inland waterways is imperative if we are to continue to engineer structures and manage water effectively in a constantly changing climate. Aside from the predictions of a rise in extreme weather conditions, resulting in more intense cyclones and freak waves, the oil and gas industry is turning to increasingly treacherous waters to search for new reserves, whilst renewable energy solutions are also looking to generate power at sea. Throughout history, tsunami waves have been devastating for the people living in the effected regions. However, they became particularly topical after the Boxing Day Tsunami of 2004 caused so much desolation in south-east Asia. With numerous coastal developments taking place all over the world, both for leisure and industry, the destruction caused by tsunamis and intense storms is potentially set to become a greater problem. Predicted sea level rise and changing weather patterns will also bring increased flooding, sometimes to new areas and sometimes damaging property built on known flood plains. As well as continuing to manage reservoirs and waste water with increasingly innovative methods, considerable research will need to take place in order to plan flood solutions for future generations.

In the past, a lot of research into these areas of fluid dynamics has been undertaken using physical model tests. However, the use of numerical models for this purpose has increased dramatically over the last few decades. Computers have become more powerful, allowing higher resolution and more complexity, whilst research into computational

methods has constantly strived for improved accuracy. This has allowed numerical methods to diversify, with many focusing on specialist areas of application.

1.1 Motivations for research

At present, there is a wide range of numerical methods available for modelling free-surface flows. The most popular of these tend to solve either the shallow water equations or the full Navier-Stokes equations using finite difference, finite element or finite volume techniques. In many cases, the application of these methods involves additional models to capture the free-surface, such as the Volume of Fluid (VOF) method.

The VOF method was originally introduced by Hirt & Nichols (1981) and is used with grid-based methods where the entire domain is filled with water and air. The interface between the two fluids is calculated at each time step and in each cell through the use of a fraction function, related to the proportion of the cell occupied by water. This implies that, in two dimensions, the free-surface is formulated by a series of linear chords. Unfortunately, this gives an overly simplistic, first-order representation of the free-surface, which tends to be dissipative and results in unphysical smoothing, reducing the accuracy of the interface location.

For some fluid flows, where viscosity and turbulence are not of great importance, the Boundary Element Method (BEM) has been shown to provide very accurate results despite its restriction to potential flow. This has been demonstrated in the context of modelling wave-wave and wave-structure interactions for a wide range of coastal and offshore structures in recent developments of a multi-flux BEM model by Hague (2006), Christou (2008) and Archibald (2011). The success of the model has been established by providing excellent correlation with laboratory experiments. This version of the BEM model is an improvement on its predecessors, in part because it has no requirements for any re-gridding or smoothing. In particular, the model has been validated by Christou (2008) through its application to overturning solitary waves, wave groups corresponding to realistic wave spectra, focused irregular waves, and waves interacting with various structures. However, the nature of the BEM means that the model is only capable of simulating waves up to a point during the overturning stage and breaks down completely well before water in the over-turning wave re-enters the main body of water.

Many of the most useful and interesting studies of free-surface flows require a numerical model to have more flexibility in the water's movement than the BEM allows. For example, many extreme waves at sea undergo wave breaking, whilst in coastal regions, the majority of waves likely to have a significant impact onshore will have broken before running up the beach. There are also numerous examples of free-surface flows involving high levels of turbulence, aeration and water droplets or jets. In order to study these flows numerically, it is therefore essential that the chosen method is proficient in their simulation. Although the VOF method is able to capture most of these flows, the first-order representation of the free-surface and underlying grid formulation do not allow the desired accuracy and level of detail to be achieved for the most complex flows.

In recent years, particle methods have significantly increased in popularity, providing a viable alternative to the traditional grid-based methods. They are of particular interest because of their versatility in modelling flows with complex or greatly deformed free-surfaces, including surfaces which may have broken up altogether. The most widely used of these methods is Smoothed Particle Hydrodynamics (SPH). This was first proposed by Gingold & Monaghan (1977), originally as a tool for modelling the movement of fluid masses in astrophysics. The method, which was later adapted for free-surface flows by Monaghan (1994), solves the Navier-Stokes equations in Lagrangian form, using a kernel approximation. The fluid domain is discretized by dividing it into 'particles', or interpolation points, which are free to move around under the governing equations through interactions with one another, whilst carrying with them the material properties of the fluid. However, as a relatively new method in the field of free-surface flows, as well as exploring its full capabilities, there remains a necessity to validate any model predictions against both established theory and laboratory experiments. This is essential if the accuracy of the model predictions is to be established both throughout the flow field and over the duration of a simulation.

1.2 Aims of the project

The aim of this PhD is to develop a numerical model using the SPH method in order to allow a greater range of free-surface flows to be modelled than has previously been possible using a BEM model. This will involve establishing the best methods available

and extending them, or developing new techniques where necessary, to create a versatile and accurate SPH model for free-surface flows. Where possible, quantitative comparisons between the SPH model and experiments will be provided to ensure the results are suitably accurate and to provide an assessment of errors arising from the model application. Once this is ascertained, the model will be applied to the description of more complex flows which have previously been difficult to model with grid-based methods.

1.3 Outline of thesis

Descriptions of the subsequent chapters of the thesis are as follows:

- **Chapter 2** provides a review of current literature relevant to the research carried out for this thesis.
- **Chapter 3** gives a description of the SPH methods and modelling techniques that have been considered within the present study and are employed throughout the subsequent chapters.
- **Chapter 4** concerns various SPH modelling techniques, explored in the context of a first test case involving solitary waves propagating in a long flume. Quantitative comparisons are provided and the most effective schemes are selected and proposed for use in the following chapter.
- **Chapter 5** continues the investigation of solitary waves, but extends the study to the full process of solitary waves breaking on an inclined beach. This includes a detailed examination of the mechanisms involved as the wave begins to overturn and forms a jet which eventually touches down on the slope in front. The analysis continues as water from the wave runs up and washes down the beach, concluding with the formation of a transient hydraulic jump. At each stage of this process, thorough, quantitative comparisons are made with laboratory experiments.
- **Chapter 6** brings together the main conclusions from the thesis and discusses some suggestions for further work in this area of research.

1.4 Principal achievements

The work carried out for this thesis has:

1. Developed an SPH model for simulating a wide range of free-surface flows by selecting and contributing to the most appropriate methods found through a thorough examination of currently available modelling techniques in SPH.
2. Demonstrated the SPH model's ability to propagate solitary waves over long distances with only minimal and explainable changes in the wave height.
3. Implemented improvements to the modelling of solid boundaries with SPH in regions which had previously been dry or have dried out, having previously been wet.
4. Demonstrated the SPH model's ability to reproduce breaking solitary waves.
5. Validated the SPH model for solitary wave breaking on a slope by providing quantitative comparisons with laboratory experiments.

2

Background

This chapter begins with a brief review of methods other than SPH that have been used to model free-surface flows. This is followed by a detailed discussion of SPH, as presently outlined in the literature, including the origins of the SPH method, an initial model for free-surface flows and further developments in relation to common free-surface flow applications for SPH models. Further details concerning the SPH method are given in Chapter 3 and more discussion of SPH applications, specific to the work outlined in this thesis, are given in the relevant sections of each results chapter.

2.1 Review of numerical methods for free-surface flows

2.1.1 Analytical solutions and early numerical models for free-surface flows

The earliest methods for representing free-surface flows required analytical solutions to the governing equations. In the case of surface water waves, theories, such as Stokes (1847), were initially developed for regular, or steady waves, involving a single freely propagating frequency component. In their simplest form, these solutions were based upon a linearisation of the free surface boundary condition, with subsequent solutions (Stokes (1880)) providing higher order approximations based upon a small perturbation expansion. These theories were later built upon to solve more realistic unsteady wave problems. Typically, such solutions, commonly referred to as Linear Random Wave The-

ory (LRWT), involve superimposing linear wave components with various frequencies, amplitudes and directions of propagation. However, such solutions ignore the non-linear wave interactions and so are unsatisfactory for modelling phenomena such as extreme waves; the implication of this is that such waves are steep and therefore nonlinear, perhaps highly nonlinear. In terms of modelling random or unsteady waves, Longuet-Higgins & Stewart (1960) proposed a second-order model, again based upon a small perturbation expansion. Although this was a vast improvement on LRWT, it is still limited in terms of its description of real ocean waves, both in terms of its accuracy and its application. Further more, none of these theoretical solutions are able to produce solutions for overturning waves.

For free-surface flows, where viscosity and turbulence are not important, potential flow theory can suffice. For example, Cassidy (1965) studied flow over a spillway crest and was able to satisfactorily solve for the location of the free-surface and the pressure on the spillway crest using potential flow theory. Steady, potential free-surface flows can also be solved using the Bernoulli equation. However, the limitations of these theories mean they too are unable to provide solutions for the more complex flow problems studied in this thesis.

In terms of numerical methods, early attempts to model surface waves were based on Fourier Methods, the free-surface being represented by a summation of sine and cosine waves. However, due to the nature of these methods, they are only able to model waves in periodic domains and cannot be used to describe waves once they start to overturn and the water surface becomes multi-valued. It therefore follows that these methods are completely unsuitable for the types of flows studied in this thesis.

Other numerical methods derive their solution from the shallow water equations. These describe the flow below a surface and so provide an excellent representation of the free-surface on the assumption that the horizontal scale is much greater than the vertical scale and vertical velocities are small. This makes them good for modelling a large range of free-surface flows, but presents difficulties when the free-surface breaks up. Although phenomena such as wave breaking and hydraulic jumps can be included in problems being modelled by these equations, thus allowing the simulation to continue, they can only be captured as shocks or bores and hence the details of these important events are completely lost.

2.1.2 Boundary Element Method

The Boundary Element Method (BEM) belongs to a group of numerical methods that describe the fluid flow in a computational domain by considering the velocity potential and potential flux around the boundary of the domain. The boundaries are defined by specifying boundary conditions in terms of both the potential and the potential flux. Provided one of these quantities is defined, the other is calculated via the boundary integral equation (BIE) which typically involves the solution of a system of linear equations. Within such solutions, the potential at interior points can be calculated afterwards, if required. This means that for free-surface flows, the fully non-linear free-surface boundary conditions can be time-marched without requiring any information from the interior of the domain. Not only does this reduce the dimensionality of the problem by one, but it also provides a highly accurate representation of the evolving free-surface. The remainder of the boundary is represented by the sides and bottom of the computational domain. This allows for the possible inclusion of varying bathymetry or moving side sections, such as those used to incorporate wave generation using a numerical wave maker or wave paddle. Unfortunately, Green's second identity, which is used for the BIE, assumes a smoothly varying boundary. With calculations undertaken in the physical domain, specifically a numerical wave tank, this is often not the case. For example, geometrical discontinuities occur at the corners of a two-dimensional domain and at the edges and corners of a three-dimensional domain. These can lead to significant numerical instabilities due to the uncertainty in the definition of the outward pointing normal; the nature of the BEM solution ensures that any such instabilities rapidly affect the entire computational domain. As a result, the boundary conditions applied at the corners must be chosen very carefully (Hague (2006)). The method also suffers from being computationally expensive, both in terms of storage and simulation times, particularly when used in three dimensions. This is due to the need to solve large systems of matrices. However, modern computational techniques, such as iterative solvers and parallelisation, have gone some way to alleviating these problems (Archibald (2011)).

The BEM representation of a fluid flow ensures that unlike the Fourier methods, the solution is not restricted to periodic flows and can allow relatively severe deformations of the free-surface, such as overturning waves. However, as the method is based on

potential flow theory, viscous and turbulent effects cannot be modelled. Furthermore, since the method is inherently dependent on the description of the free-surface as a boundary, the free-surface can never be allowed to break up. As a results although the method has been shown to provide very accurate results for unbroken waves, it is clearly unsuitable for modelling many of the flows studied within this thesis.

Examples of studies carried out using the BEM include the work of Grilli *et al.* (1997) and Grilli *et al.* (2004); both studies investigate solitary waves overturning on various bed slopes. More recently, Hague (2006) studied the same problem with an improved BEM code. The advantage of this latter model over that described by Grilli *et al.* (1997) is that the model does not require any smoothing, filtering or re-gridding and overcomes the corner problem described above by using a system of multiple fluxes (Hague & Swan (2009)). This model was developed further by Christou (2008) and Archibald (2011); the resulting advances include improvements to the computational efficiency of the model and its application to the study of wave-wave and wave-structure interactions.

2.1.3 Numerical methods for two-phase flows

Marker and Cell methods

The Marker and Cell (MAC) method was originally introduced by Harlow & Welch (1965) to simulate the time-dependent motion of viscous free-surface flows. The method uses a finite difference method to solve the full Navier-Stokes equations on a Eulerian grid, with velocity and pressure boundary conditions applied at the free-surface. In addition, a set of marker particles are included to indicate which cells contain fluid and which are empty. The marker particles are moved according to the local velocities defined by the grid, with adjustments to the particles' velocity when moving into previously empty cells or emptying previously occupied cells. The marker particles are only used to indicate the location of the free-surface and do not take part in calculations. Harlow & Welch (1965) tested the method on a collapsing dam simulation and compared the results to the laboratory experiments of Martin & Moyce (1952), with the results shown to be in good agreement. In general, MAC methods are considered to be extremely accurate for many non-violent flows, but have difficulties predicting flows with highly complex motion, where marker particles are either pushed too close to one another or

stretched too far apart. This can cause instabilities which can be somewhat resolved by regridding, although this introduces new errors into the solution (Osher & Sethian (1988)).

Level set methods

Level set methods describe the interface between two fluids as a zero level set of a function of the distance from the interface. The distance function is advected with the local fluid velocity. An example of such a method is the Propagation of Surface under Curvature (PSC) method of Osher & Sethian (1988). This method was originally designed for modelling crystal growth and flame propagation, but can equally be applied to simulating free-surface flows with an interface between the water and the air. The scheme is based on a Eulerian grid and follows the motion of the surface by numerically solving the Hamilton-Jacobi equations including viscous terms. According to Gopala & van Wachem (2008), level set methods give accurate results when the interface is advected parallel to a coordinate axis, but suffer from loss of mass in cases where the interface becomes significantly deformed.

Volume of Fluid method

The Volume of Fluid (VOF) method, originally introduced by Hirt & Nichols (1981), is a technique for modelling the free-surface in grid-based numerical methods. It is often used in combination with the Finite Volume Method (FVM) to solve the Navier-Stokes equations for fluid flow, although it can also be used in a similar fashion with finite difference based methods. The interface between two fluids is calculated at each time step and in each grid cell through the use of a fraction function related to the proportion of the cell occupied by water. In older versions of the VOF method, the fluid in each interface cell was assumed to have a horizontal surface. However, more modern techniques, such as those used by Savage & Johnson (2001) and the Segment Lagrangian-Volume of Fluid method developed by Guignard *et al.* (2001), use a surface angle as well to improve the method's accuracy and, most importantly, its surface representation. However, despite these improvements, the free-surface continues to be represented by a series of linear chords in two dimensions or planes in three dimensions. This gives an overly simplistic, first-order representation of the free-surface, which tends to be dissipative

and results in unphysical smoothing, reducing the accuracy of the interface location.

Within the VOF methods, a number of volume advection techniques are available, several of which are considered by Gopala & van Wachem (2008). These include donor-acceptor methods, such as the original method of Hirt & Nichols (1981) and more recent complex versions of the technique, such as the Compressive Interface Capturing Scheme for Arbitrary Meshes (CICSAM) by Ubbink (1997) and the Inter-Gamma Differencing Scheme of Jasak & Weller (1995). In these methods scalar functions are advected through the interface between the donor cell and a downwind cell, known as the acceptor cell, using contributions of the same function from both of these cells as well as an upwind cell. The particular scheme used to define the scalar function depends on the angle of the interface with the face of the cells.

The VOF methods have several important advantages over the BEM. They are able to include the affects of viscosity and are often coupled with turbulence models. There are also no restrictions placed on the location and movement of the free-surface. As a result, it is allowed to break up and can include air bubbles within the main body of water. However, the description of the free-surface is not as accurate as that achieved using the BEM (Guignard *et al.* (2001)), the explanation for this being outlined above. Indeed, this provides one of the main incentives for developing an SPH model, which has the potential to approach the accuracy of the BEM whilst, at the same time, being capable of modelling flows with large deformations and break up of the free-surface.

Despite the reservation noted above, the VOF method has been used by Guignard *et al.* (2001) to successfully model solitary waves propagating and breaking on slopes. However, the calculations during propagation were shown to be less accurate than predictions based on the BEM. More discussion concerning VOF studies is given in the review sections of the relevant results chapters in this thesis.

2.1.4 Meshless methods

SPH is by far the most popular meshless method in the current literature and is the method of choice for the studies in this thesis. Accordingly, the method is reviewed and described in more detail in the following sections of this chapter and beyond. However, before doing so, it is important to note that a number of other meshless methods have been developed, such as the Moving Particle Semi-Implicit (MPS) method, originally

introduced by Koshizuka *et al.* (1995). The method is similar to SPH, in that the fluid domain is represented by particles and the governing equations are solved through the particles' interactions with one another using appropriate kernels. However, it differs from SPH in the calculation of pressure and density. Also, within the MPS method, physical properties are only considered on the particles themselves; their distribution is not considered as it is in SPH. According to Monaghan *et al.* (2003), the MPS method does not conserve momentum exactly. This is due to the construction of the pressure gradient and means larger pressure fluctuations can be expected since the kernels do not have continuous first derivatives. For this reason, SPH is the widely preferred method. Nevertheless, Gotoh *et al.* (2005) used the MPS method to model non-breaking and breaking waves overtopping a sea wall. The results, which were qualitatively compared with laboratory observations, were shown to be in good agreement for the non-breaking wave case. This confirms the particle method's ability to simulate flows with large deformations from the initial geometry of the free-surface. However, the method did not satisfactorily reproduce the experimental results in the breaking wave case. The authors suggest that this is due to the method not being able to resolve the sub-particle scale (SPS) features of the flow. Therefore, Gotoh & Sakai (2006) introduced an SPS turbulence model into the MPS method for modelling breaking waves. Snapshots from the simulations are provided, but they are not compared with experimental observations and the free-surface appears to be bumpy and rapidly degrades into spray as soon as the waves begin to break. This problem with the method was not discussed in any detail and since it is not a feature of SPH calculations, as will be shown in subsequent chapters, it is not considered further here.

2.2 Review of Smoothed Particle Hydrodynamics

2.2.1 Origins of SPH

Smoothed Particle Hydrodynamics was first proposed by Gingold & Monaghan (1977) as a tool for modelling the movement of fluid masses in astrophysics. For this purpose, it was desirable to design a numerical method that would be easy to work with and provide accurate results. SPH is a Lagrangian particle method, which solves the Navier-Stokes equations using a kernel approximation. The fluid domain is discretized by dividing

it into ‘particles’, or interpolation points, which are free to move around under the governing equations through interactions with one another, whilst carrying with them the material properties of the fluid.

Following its initial development, applications of SPH over the next decade were confined to astrophysical problems, and therefore, compressible flows. However, some developments of the method for these purposes later proved to be useful when dealing with terrestrial problems, such as free-surface flows. For example, shocks and penetration were encountered in Monaghan & Pongracic (1985) and this led to the introduction of an artificial viscous term. Shortly after, Monaghan (1989) introduced a velocity correction term (XSPH) into the calculation of particles’ positions in order to prevent unphysical penetration of particles.

Monaghan (1992) published a review of SPH detailing the numerical formulation of the method; this contribution still provides the basis for most SPH models today. The majority of the applications of SPH in this review paper were related to astrophysics, but towards the end, the concept of adapting SPH for nearly incompressible flows was discussed. In this respect, an artificial equation of state must be introduced to ensure that compressibility effects are kept to a minimum. With this new use for SPH, came the need to model solid boundaries, which still remains a considerable challenge today.

The equations and formulation of SPH for modelling free-surface flows are given in detail in Chapter 3. In the following subsection a brief introduction to the SPH method is provided, together with some key examples outlining both the success of the method and the outstanding difficulties.

2.2.2 An initial model for free-surface flows

The SPH method was fully extended to deal with free-surface flows by Monaghan (1994). This original model uses the basic SPH equations, adopting a kernel approximation to calculate the momentum and continuity equations. The conservation of mass is automatically satisfied in SPH, provided the number of SPH particles is kept constant. Both artificial viscosity and XSPH were included in the free-surface model and the artificial equation of state was chosen to be of the form of Batchelor (1974). The boundaries were constructed using a row of particles that exert a repulsive force on the fluid particles to avoid penetration. The repulsive force has a Lennard-Jones form and

was based on the known molecular forces exerted by a real boundary, but scaled to be close to the resolution length of the calculation. Using this method, both stationary and moving boundaries could be modelled and, if desired, a no-slip boundary condition could also be created by the inclusion of boundary particles in the viscous term. A predictor-corrector time-stepping scheme was used, although no reasons were given for this choice. Prior to initialising the main calculations for the problem, a damping term was introduced into the momentum equations to settle down any spurious energy created from the initial placing and allocation of physical properties to the particles. Depending on the particular aspects of the SPH model, it is common to find that, even if no external forces are applied, particles may move slightly from their initial position. This is due to the particles' desire to naturally fall into the most stable positions relative to one another. Therefore, despite the rare occurrence of a damping term in the literature, it seems plausible that such a term would be beneficial to the calculations.

Monaghan (1994) tested his free-surface SPH model by undertaking simulations of the evolution of an elliptical drop, a bursting dam, a propagating bore and a wave maker, the latter using a moving boundary. The tests showed that the SPH method was capable of simulating free-surface flows subject to the use of the XSPH velocity correction and assuming that the density was calculated through an approximation of its rate of change. However, significant problems were identified close to boundaries: first, the nature of the repulsive forces produces a discrete effect, so that the boundary force is not continuous, and second, close to the free-surface, the fluid peels away from the boundary because the hydrostatic pressure is weaker in this region. These problems are confined to the area within a few resolution lengths of the boundary and so do not affect the quality of the simulation as a whole. However, if detailed results are required at the boundary, it is crucial that improvements are made to the modelling of boundaries in SPH. The results of the bursting dam simulation, which were compared to the experiments of Martin & Moyce (1952), are discussed in § 2.2.3.

Since the initial adaption of SPH for modelling free-surface flows, many authors have contributed to the development of such models. Further details are given below in a number of brief, problem specific reviews.

2.2.3 Applications and further developments

SPH has been used to study many free-surface flow applications. These have included relatively simple tests, such as collapsing dams and sloshing tanks, as well as more complex problems involving waves generated by impacts, solitary waves, breaking waves and problems involving wave interactions with structures. More recently, SPH has been used to model real-life events, such as the flows resulting from landslides or flows over specific spillways. In some cases the SPH results have been compared with experimental data, theory and other numerical models. However, thorough quantitative comparisons are currently rare in the literature. This section provides a discussion on some of the free-surface applications present in the SPH literature. Further applications, specifically relevant to the work carried out in this thesis, are discussed in more detail in the Background sections of Chapters 4 and 5.

Collapsing dams

Collapsing columns of water, commonly referred to as the collapsing dam problem, represent a relatively straight forward application for SPH models, and therefore provide a good test case. For this reason, the successful modelling of a collapsing dam is one of the most frequent tests of a new or modified SPH model and is common in the literature. Within these numerical simulations, the dam is typically set up as a block of fluid particles with boundaries below and to the left. The dam is then allowed to collapse under gravity, such that the height of the remaining column decreases as the fluid surges forwards, forming a propagating bore. The literature presently includes examples of two-dimensional columns, axi-symmetric columns and cylindrical columns.

Both Monaghan (1994) and Doring *et al.* (2003) modelled dams in two dimensions with standard SPH models and compared their results to the experiments of Martin & Moyce (1952). In both cases the height of the column, throughout the collapse, was found to be in excellent agreement with the experiments. However, the position of the numerical surge front exceeded the experimental surge throughout the simulations. The most likely explanation for this lies in the effects of the drag forces present in the experiment, which neither Monaghan (1994) nor Doring *et al.* (2003) included in their models. According to Morris *et al.* (1997), using first-order no-slip boundaries can

lead to velocity errors. Therefore, an increase in particle number, and hence resolution, would be an appropriate way of minimising the errors associated with not modelling drag. Monaghan (1994) also noted difficulties with the modelling of boundaries and the timing of the experiment, both of which could effect the results. With these factors taken into account, it can be said that the SPH models used here are suitable for modelling collapsing dams.

Other SPH models which have been tested via their application to collapsing dams include the use of Reimann solvers to describe the velocity and pressure (Roubtsova & Kahawita (2006)); models based on the Shallow Water Equations (Ata (2005) and Rodriguez-Paz & Bonet (2005)); incompressible models, which solve the Poisson equation for pressure (Shao & Lo (2003)); and modelling air as well as water, thus creating an interfacial flow (Colagrossi & Landrini (2003)). All of these modifications reproduce the available experimental results with a similar level of accuracy to that achieved by the standard SPH models. However, in terms of modelling the collapse of a dam, none of these modifications substantially improve the accuracy of the numerical predictions. It therefore follows that the benefit of these, often complex, modifications remains to be seen.

In some cases the introduction of an obstacle, such as a ramp, downstream of the dam has been considered. These results are harder to quantify and comparisons to photographs of experiments are often unsatisfactory. Further more, the effects of a wet bottom in the tank are considered by Dalrymple & Rogers (2006). In experimental studies, the tank bed is often initially wet. This causes the advancing bore to push up fluid on the wet bed, thereby creating a wave moving backwards down the tank. Unfortunately, quantitative results are not given for this case, but it is clear that subtle differences between the laboratory observations and the numerical calculations can be very significant. Crespo *et al.* (2008) also used a test case involving a collapsing dam over a wet bed to validate their SPH model. Quantitative comparisons of the water surface profile throughout the dam breaking process were made with experiments, with statistical parameters being introduced to determine the accuracy of the SPH results. The wave front velocity is also quantitatively compared to experiments. The rigorous validation of the SPH model against experiments allowed other aspects of the model, which cannot easily be measured experimentally, to be investigated. These include the

mixing process of water originally located behind and in front of the dam gate, energy dissipation and vorticity.

More recently, Gomez-Gesteira *et al.* (2010) used the collapsing dam test case to investigate the accuracy of various aspects of their state-of-the-art classical SPH model including varying the smoothing length and a range of density filters. The numerical results were compared to experimental observations and two statistical parameters were employed to assess the accuracy of the SPH model, providing a thorough quantitative comparison. The SPH model is similar to that used in this thesis, with the method for testing various numerical techniques analogous to that described in 4.4 whilst considering solitary wave propagation, albeit with fewer techniques analysed. They conclude that the development of the SPH technique for modelling free-surface flows has reached a maturity where quantitative comparisons with experimental measurements can be carried out and an accuracy level similar to more conventional CFD methods should be expected.

In conclusion, modelling collapsing dams and producing results in good agreement with experiment is achievable with the standard SPH formulation and with various adaptations to this formulation. In general, the benefits to be gained by modelling improvements are not substantial. Nevertheless, due to the simplicity of the application, an SPH model's capability to reproduce a collapsing dam successfully is a good test of the validity of the model. Once an SPH model has been validated for a particular test case, the benefits of numerical modelling can be realised by allowing further analysis into properties of the flow which are difficult to measure experimentally.

Sloshing tanks

The sloshing tanks described in the SPH literature by Doring *et al.* (2003), who uses a standard SPH model, and Roubtsova & Kahawita (2006), who uses Reimann solvers to describe the velocity and pressure, are generated by moving a tank, 60% full of water, through prescribed horizontal oscillations. This provides another relatively simple test for SPH models, as the fluid motion is forced constantly and confined within the tank. In both cases, the models are compared to the water surface elevations arising in the experiments of Corrigan (1994.). The SPH results are in good agreement with the experiments, particularly when large numbers of particles were used to obtain a high

resolution. However, quantitative results are not available and so percentage errors cannot accurately be obtained.

More recently, Colagrossi *et al.* (2010) used an improved weakly-compressible SPH model to analyse violent sloshing wave impacts and compared the pressure at impact quantitatively with experiments. The results were in good agreement, showing that their SPH model is capable of producing a good pressure field. Pakozdi & Graczyk (2009) also made comparisons between SPH simulations and experiments of pressure measurements in sloshing tanks, but for non-impact fluid motion.

Both Doring *et al.* (2003) and Roubtsova & Kahawita (2006) tested their SPH models on collapsing dams and sloshing tanks, thus showing the versatility and validity of their formulation. However, forcing motion continuously means that the limits of an SPH model are not fully tested, as they are for solitary wave propagation, for example, in which the SPH model must sustain the correct fluid motion for a long period of time.

Waves generated by impact

This test case provides a more complicated problem in which viscous effects are present and no analytical solution is available to describe the whole process. However, such problems are closely related to practically important problems, not least the occurrence of tsunamis caused by underwater landslides. If SPH is able to provide accurate solutions to simplified cases of this nature, then it may provide a useful tool when applied to real-life events.

Monaghan & Kos (2000) looked at the generation of solitary waves formed by dropping a weighted box vertically into water using SPH. This event causes a jet to form below the box which forces the water in the tank upwards, resulting in a reverse plunging wave, followed by a solitary wave. As the reverse plunging wave collapses, a vortex is created which follows the solitary wave as it propagates down the tank. The SPH model was very similar to the standard SPH model for free-surface flows described by Monaghan (1994). Two dimensional simulations were carried out for a range of box sizes and water depths. Experiments were also carried out and the height of the solitary waves were used for comparison. Converged results for the SPH model were found to have errors in the range of 3%-18% when compared to experiment, depending on the water depth. This is not particularly accurate, but it was found that the small gap

behind the weighted box had a significant effect, and when this was modelled the errors were considerably reduced. However, it was also noted that a higher resolution in the SPH model would be required to model the gap properly. These results confirm the ability of SPH to model more complex flows with a good degree of accuracy as long as all aspects of the system are included in the model. Unfortunately, no further details were given in relation to the propagation of the solitary wave.

Fluid motion generated by impact with a solid body was also considered by Monaghan *et al.* (2003). In this case he considered boxes sliding down curved ramps to generate wave motion. Experimental observations were undertaken with boxes released from various heights to generate a range of impact velocities and the fluid motion was recorded by video. As the box entered the water a vigorous jet was produced. A large body of water was heaved up at the base of the jet and this settled on top of the box, so that as the box continued to move, this water moved with it, initiating a large wave. The wave then propagated down the tank, was reflected off the end wall and returned, approximately, as a solitary wave. Again, the SPH model was very similar to that described by Monaghan (1994), the main difference being the inclusion of three possible modelling techniques for friction: no friction, reduced friction and full friction. Unfortunately, the optimum model was found to depend on the local water depth. As a result, none of the friction models could be considered universally acceptable, ensuring that their practical application is very difficult, if not impossible. However, to put these results in context, it is important to note the simplifying assumptions on which the model was based. First, the model was two-dimensional, so no three-dimensional effects could be considered, including flow around the sides of the box. Second, the tank was reduced in size and, third, it was assumed that the effects of the overlying air could be neglected. Despite these assumptions, comparisons between the scatter plots of particle positions from the SPH model and the experimental video frames showed agreement to within just a few percent. Qualitatively, the SPH results were very good, reproducing the whole process well, but some details were undoubtedly lacking. For example, the resolution in the jet becomes very poor and was not reproduced accurately. Some improvement in this was achieved when the overall particle resolution was increased, but difficulties still remained. It is also noted that some aspects of the flow, including the air trapped under the water on top of the box, introduced disorder into the system which could not be

described by this SPH model. In order to improve upon the accuracy of these results, other physical effects would need to be included in the model, not least the effects of turbulence and aeration.

Ataie-Ashtiani & Shobeyri (2008) used an incompressible SPH formulation to simulate impulsive waves generated by landslides. The landslides were generated by allowing a rigid wedge, initially located just below the water surface, to freely slide down a plane inclined at 45° to the horizontal. A quantitative comparison of the resulting wave profiles was made between the SPH predictions and experimental data at two selected times. The wave profiles are in good agreement, showing that the incompressible SPH model is able to simulate this type of flow. However, the results, and in particular the comparisons to experimental data, are very limited and more analysis on the flow as a whole should be carried out in order to fully investigate the capability of incompressible SPH models for simulating this type of flow. A similar study was conducted by Das *et al.* (2009), who also looked at waves generated by landslides. Das *et al.* (2009) chose to use a more standard weakly compressible SPH model and compared results to both experiment and to a Reynolds-Averaged Navier-Stokes (RANS) VOF Method. Although the SPH results were found to qualitatively agree with both the experimental data and RANS VOF model, it was concluded that some of the error in the SPH predictions could be due to a lack of sufficient diffusion, since a turbulence model was not employed. Rogers & Dalrymple (2008) investigated landslide-generated tsunamis with both two and three-dimensional SPH models, concluding that the three-dimensional model was necessary to fully capture the landslide and flow. However, high performance parallel computing would be necessary to carry out the simulations at the desired resolution in three-dimensions.

The results arising from this test case have shown that SPH models can qualitatively provide good results. However, to increase the modelling accuracy more physical effects must be included in the models. This is always the case with the application of any numerical model and requires an acceptable level of accuracy to be defined. This test also showed that although a relatively low resolution may be acceptable when modelling some problems (for example, a collapsing dam), high resolution and a three-dimensional model may be necessary when a low number of particles is likely to be found in a particular region of the flow.

Solitary waves

The modelling of a propagating solitary wave with SPH provides a further challenge still, not least because the model must be able to maintain the wave motion for a long period of time without any external driving forces. Solitary wave run-up on slopes has also been the subject of several comparisons to laboratory data, most qualitative in nature.

Monaghan & Kos (1999) looked at the run-up and run-down of solitary waves on a dry beach. Experiments were carried out using a weighted box to create the solitary wave and the position of the wave was estimated from photographs. The process involves the wave running up the slope, hitting the end wall, forming a reverse plunging wave and returning as a disordered bore. The employed SPH model was very similar to that described in Monaghan (1994). The solitary wave was created by placing particles on a Cartesian grid, removing particles which lie above the solitary wave profile, and prescribing the remaining particles with velocities appropriate to first order solitary wave theory. Initially, a simple tank with no slope was used and the run-up on the vertical end wall was compared to experimental observations involving several incident wave heights. The SPH results compared well to the experiments, particularly at small wave heights. However, the tank length was very short, so that when the peak of the wave reached the end wall, it had only travelled approximately five times the depth of the water. Unfortunately, this length is insufficient for establishing whether or not the predicted solitary wave was capable of propagating without significant energy loss. With the beach included in the SPH model, the results from the simulation were compared to the photographs taken during the experiments. The wave shape and wave amplitude were well defined throughout the short experiment, the agreement between the two being within the estimated measurement error of the experiments. These results suggest that SPH provides a useful tool for modelling solitary wave run-up, but further tests are needed to properly assess the solitary wave motion prior to run-up. Again, some details of the reverse plunging wave were not captured well due to the poor resolution within this region of the flow.

The propagation of solitary waves was again studied by Vaughan *et al.* (2004), using a similar two-dimensional SPH model to that outlined by Monaghan & Kos (1999). The

solitary wave was initiated by removing particles above the solitary wave profile in the same way as that described by Monaghan & Kos (1999). However, in this case a second order solitary wave profile was employed. Once again, a variety of wave heights were used. A water depth of 0.3m was maintained and the waves were allowed to propagate over a distance of 8m. This arrangement ensured that the ability of the SPH model to conserve energy could be explored. As the solitary waves propagated down the tank, a significant loss in wave height, a marked dispersion or spreading out of the wave event and a substantial reduction in the phase velocity were observed. In particular, by the time the waves had travelled 8m along the tank, the wave heights were between 60% and 40% of their original heights. It is thought that this reduction was mainly due to numerical errors in the system and the application of a poor particle resolution. However, there are a number of other questionable techniques applied in this model, not least the absence of a viscosity model, which increases noise in the system, and a very large smoothing length compared to the water depth, which could lead to over-smoothing in the calculations. These aspects of the SPH model are fully described in Chapter 3. As a result of these tests, it is clear that this model must be greatly improved in order to model solitary waves with a satisfactory level of accuracy.

Lo & Shao (2002) also looked at the propagation of solitary waves, but made several important adjustments to the standard two-dimensional SPH model. First, SPH was coupled with large eddy simulation (LES) and, second, incompressibility was enforced by solving a Poisson equation for pressure rather than using the usual equation of state. The dynamic free-surface boundary condition, imposing constant pressure, was enforced on those particles found to be at the water surface, and a pressure gradient treatment was applied to overcome unavoidable errors arising from this pressure correction. Again, to initialise the solitary wave, the particles were set up on a Cartesian grid and those above the first order solitary wave were removed. The remaining particles were assigned velocities associated with first order solitary wave theory. Only one wave height was considered, corresponding to a value of $H/d = 0.15$. The wave was allowed to propagate for a distance of more than 100 times the water depth. Over this distance, comparisons to first order solitary wave theory were excellent, suggesting that the different modelling techniques are highly beneficial for this particular problem.

Within this series of tests (Lo & Shao (2002)), solitary wave run-up on the vertical

end wall was also looked at for a range of wave heights and comparisons were made with the results obtained by Monaghan & Kos (1999) and associated experiments. The SPH results compare very well to the experiments and some improvement over the SPH model used by Monaghan & Kos (1999) can be seen at larger wave heights. Solitary wave breaking on a mild slope was also studied with this numerical model and the wave profiles at various times were compared to the experimental results of Synolakis (1987). The agreement was said to be excellent, although the images of wave breaking do not show the free-surface profile to be clearly defined and there was some evidence that the resolution was not sufficiently high at the maximum run-up. It was also noted that after wave breaking has occurred, turbulent effects become important, and must be modelled. The SPH model outlined by Lo & Shao (2002) was able to incorporate viscosity, but for LES to capture the large-scale eddy motion accurately, the particle resolution must be high. Overall, this SPH model appears to be significantly better at capturing solitary wave propagation when compared to the standard SPH model, but there remains some ambiguity as to the quality of the results during the breaking process and subsequent run-up on the slope.

These results show that SPH does indeed provide a suitable tool for modelling solitary waves and can easily be employed for looking at run-up on slopes and subsequent breaking. Some problems were found in relation to wave propagation when using a fairly standard SPH model, but there is potential for improvements within this model. Certainly, these problems appear not to be significant when using the model by Lo & Shao (2002), but others remain.

2.2.4 Concluding remarks

SPH is a relatively new method for modelling free-surface flows, and model development is still very much ongoing. New techniques are constantly being introduced, their purpose being to improve the accuracy of SPH models. Taken as a whole, the proposed changes have had varying success in terms of physical accuracy, practicality and results. Certainly, further work is required to establish the best method, or combination of methods for creating an accurate and versatile SPH model for free-surface flows.

It has been shown in the literature discussed in this section, and elsewhere, that SPH is capable of modelling a wide range of free-surface flow problems including break-

ing waves, air entrainment and interaction with structures. In some cases, very good agreement has been found with experiment, emphasising the usefulness of SPH as a numerical tool. However, further quantitative comparisons are required to fully assess the accuracy of the method.

3

Numerical formulation of Smoothed Particle Hydrodynamics (SPH)

This chapter considers the mathematical basis of SPH and various aspects of SPH as a modelling tool for free-surface flows. The fundamental SPH equations are given in § 3.1. This is followed in § 3.2 by a presentation of the governing fluid dynamics equations in SPH form. The model is then broken down into its component parts: § 3.3 concerns the equation of state for calculating pressure, § 3.4 provides a description of varying representations of the viscosity term in SPH, whilst § 3.5 introduces turbulence modelling. Additional modelling techniques relating to particle penetration and density smoothing are described in § 3.6 and § 3.7 respectively, whilst § 3.8 and § 3.10 provide a discussion on the various techniques used to model boundaries and boundary conditions. § 3.11 looks at the SPH Courant Friedrichs Levy-type (CFL) condition and time marching schemes, whilst § 3.12 considers the computational effort involved in SPH modelling. This is followed by a discussion of the conservation properties of SPH in § 3.13, with the errors and accuracy of the method discussed in § 3.14. The final sections of this chapter, § 3.15 and § 3.16, round up some other methods used in SPH models, which are not considered elsewhere.

3.1 Basis for SPH

The aim of SPH, or any computational fluid dynamics method, is to describe the physical quantities of one or more fluids lying within a specified domain and predict their change or evolution over a period of time. Accordingly, the rate of change of physical quantities within the domain must be calculated, both in time and space. It therefore follows that spatial derivatives of the physical quantities are required. From a computational perspective, these must be approximated using information stored at a finite number of points. In finite difference methods, these points are the nodes of a mesh and spatial derivatives are calculated by interpolating between the nodes. In SPH, however, the points are the particles which move with the flow, making SPH a Lagrangian method. Physical quantities are calculated at any point in space by interpolating between the particles.

Interpolation of physical quantities is carried out using a kernel approximation. The kernel is best described by the following equation, which is used to approximate some physical quantity, A , at a point in space, $\mathbf{r}$, based upon known information concerning A at a point, or points, $\mathbf{r}'$. The equation is given in its integral form as

$$\tilde{A}(\mathbf{r}) = \int A(\mathbf{r}') W(\mathbf{r} - \mathbf{r}', h) d\mathbf{r}', \quad (3.1)$$

where $\tilde{A}(\mathbf{r})$ is the approximation of $A(\mathbf{r})$, $W(\mathbf{r} - \mathbf{r}', h)$ is the kernel, or a function connecting the point $\mathbf{r}$ with $\mathbf{r}'$. h is the smoothing length, which is a problem dependent parameter and determines the spatial resolution, just as the size of elements or grid cells do in finite difference methods. The kernel is an approximation to the dirac delta function and therefore has the following property:

$$\lim_{h \rightarrow 0} W(\mathbf{r} - \mathbf{r}', h) = \delta(\mathbf{r} - \mathbf{r}'). \quad (3.2)$$

This implies that when the smoothing length tends towards zero, Equation (3.1) tends towards the exact identity

$$A(\mathbf{r}) = \int A(\mathbf{r}') \delta(\mathbf{r} - \mathbf{r}') d\mathbf{r}'. \quad (3.3)$$

It is also important that the kernel function is normalised to 1. This is defined by the property

$$\int W(\mathbf{r} - \mathbf{r}', h) d\mathbf{r}' = 1, \quad (3.4)$$

which ensures that constants are interpolated exactly.

There are many possible kernels, the most popular of which are those based on Schoenberg (1946) M_n splines. These are piece-wise continuous functions with continuous derivatives up to $(n - 2)$ and compact support, ensuring that only nearby particles are taken into account in calculations. One commonly used example is the cubic spline kernel devised by Monaghan & Lattanzio (1985),

$$W(q) = \frac{\alpha_d}{h^d} \times \begin{cases} \frac{2}{3} - q^2 + \frac{1}{2}q^3 & 0 \leq q < 1 \\ \frac{1}{6}(2 - q)^3 & 1 \leq q < 2 \\ 0 & q \geq 2 \end{cases}, \quad (3.5)$$

where $q = |\mathbf{r} - \mathbf{r}'|/h$, d defines the dimension of the simulation and α_d is chosen to meet the normalisation condition given by Equation (3.4). For two-dimensional problems, $\alpha_2 = \frac{15}{7\pi}$. However, it can be preferable to use a higher-order spline-based kernel, as these more closely approximate a Gaussian function and reduce instabilities (Morris *et al.* (1997)). A disadvantage of the cubic kernel is that the second derivative, which the stability properties of the SPH solution depend upon, is a piecewise-linear function. Since higher-order kernels have smoother second derivatives, they are generally more stable. An example of a higher order kernel is the quintic kernel used by Morris *et al.* (1997). This is given as follows:

$$W(q) = \frac{\alpha_d}{h^d} \times \begin{cases} (3 - q)^5 - 6(2 - q)^5 + 15(1 - q)^5 & 0 \leq q < 1 \\ (3 - q)^5 - 6(2 - q)^5 & 1 \leq q < 2 \\ (3 - q)^5 & 2 \leq q < 3 \\ 0 & q \geq 3 \end{cases}, \quad (3.6)$$

where $q = |\mathbf{r} - \mathbf{r}'|/h$ and $\alpha_d = \frac{7}{478\pi}$ in two dimensions. The penalty for using smoother, higher-order kernels is an increase in the computational runtime. As a compromise between the shorter runtime associated with the cubic kernel and the superior properties of the quintic kernel, the quartic kernel used by Violeau & Issa (2007) can be employed. This is defined as follows:

$$W(q) = \frac{\alpha_d}{h^d} \times \begin{cases} \left(\frac{5}{2} - q\right)^4 - 5\left(\frac{3}{2} - q\right)^4 + 10\left(\frac{1}{2} - q\right)^4 & 0 \leq q < 0.5 \\ \left(\frac{5}{2} - q\right)^4 - 5\left(\frac{3}{2} - q\right)^4 & 0.5 \leq q < 1.5 \\ \left(\frac{5}{2} - q\right)^4 & 1.5 \leq q < 2.5 \\ 0 & q \geq 2.5 \end{cases}, \quad (3.7)$$

where $q = |\mathbf{r} - \mathbf{r}'|/h$ and $\alpha_d = \frac{96}{1199\pi}$ in two dimensions.

In order to apply the interpolation equation in SPH, Equation (3.1) must be approximated by a summation over the relevant particles. In this form the interpolation equation is given by

$$A_a = \sum_{b=1}^N A_b \cdot W_{ab} \, dV_b. \quad (3.8)$$

This implies that a physical quantity, A , for particle a (denoted by A_a), is calculated by summing over all the other particles (defined by $b = 1$ to N) using the kernel approximation. N is the total number of particles and W_{ab} is the kernel function for particles a and b , where $|\mathbf{r} - \mathbf{r}'|$ is the distance between the two particles. The differential volume element, $d\mathbf{r}'$, has been replaced by the volume, dV_b , of a particle and is defined by

$$dV_b = \frac{m_b}{\rho_b}, \quad (3.9)$$

where m_b and ρ_b are, respectively, particle b 's mass and density. An example of Equation (3.8) is the SPH density summation, given by

$$\begin{aligned} \rho_a &= \sum_{b=1}^N \rho_b \cdot W_{ab} \frac{m_b}{\rho_b} \\ &= \sum_{b=1}^N m_b \cdot W_{ab}. \end{aligned} \quad (3.10)$$

In Equation (3.8) all the particles within the domain are included in the summation, but in practice all SPH kernels have compact support. This means that only the particles lying within particle a 's compact support region, otherwise known as the particle's influence domain, are included in the summation. Outside this region, or domain, the value of the kernel falls to zero. The size of this region depends on the particular kernel. For example, the influence domain for the cubic kernel given by Equation (3.5) is the

region enclosed by a circle with a radius of twice the smoothing length. Accordingly, $W(q) = 0$ for $q \geq 2$. Figure 3.1 shows the influence domain for a particle a . When using the quintic kernel, given by Equation (3.6), the increase in the runtime is due to the influence domain operating over a region enclosed by a circle with a radius of three times the smoothing length, since $W(q) = 0$ for $q \geq 3$. With an increase in the area of the influence domain, more particle pairs must be included in each step of the calculation.

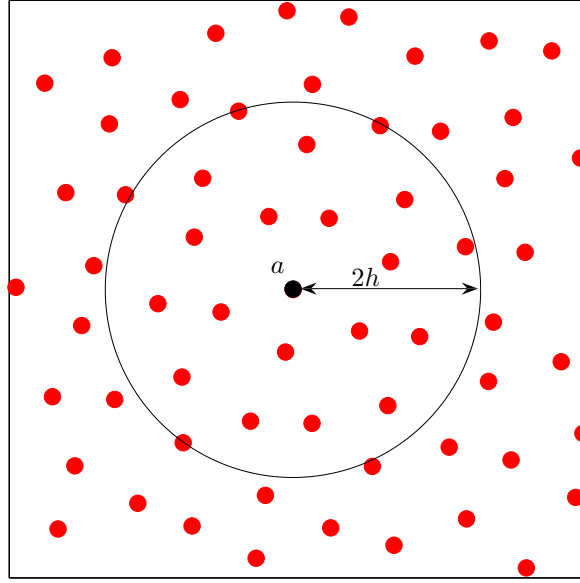


Figure 3.1: Influence domain for particle a

First derivatives can also be expressed in the SPH form. The general equation for a first derivative is given by

$$\nabla_a \cdot A_a = \sum_{b=1}^N A_b \cdot \nabla_a W_{ab} \frac{m_b}{\rho_b}, \quad (3.11)$$

so that the derivative of A is found by the exact derivative of the approximate kernel function. However, Equation (3.11) is unsuitable if A is constant, since the derivative does not vanish. A better equation for calculating first derivatives can be formed by introducing an arbitrary differentiable function, Φ , into the equation, such that

$$\nabla \cdot A = \frac{1}{\Phi} (\nabla \cdot (\Phi A) - A \cdot \nabla \Phi). \quad (3.12)$$

In SPH form this becomes

$$\nabla_a \cdot A_a = \frac{1}{\Phi_a} \sum_{b=1}^N \Phi_b (A_b - A_a) \cdot \nabla_a W_{ab} \frac{m_b}{\rho_b}. \quad (3.13)$$

It should be noted that in the transformation of Equation (3.12) to Equation (3.13), when Φ or A are inside an operator, they are taken in the SPH form as being properties of the particle being summed over (b). In contrast, when they are outside an operator, they are properties of the particle for which the property is being calculated (a). The function, Φ , could, for example, be taken as either 1, giving

$$\nabla_a \cdot A_a = \sum_{b=1}^N (A_b - A_a) \cdot \nabla_a W_{ab} \frac{m_b}{\rho_b}, \quad (3.14)$$

or ρ , giving

$$\begin{aligned} \nabla_a \cdot A_a &= \frac{1}{\rho_a} \sum_{b=1}^N \rho_b (A_b - A_a) \cdot \nabla_a W_{ab} \frac{m_b}{\rho_b} \\ &= \frac{1}{\rho_a} \sum_{b=1}^N m_b (A_b - A_a) \cdot \nabla_a W_{ab}. \end{aligned} \quad (3.15)$$

Since both of these expressions now include the term $(A_b - A_a)$, the derivative will vanish if A is constant. Examples of these two forms of derivative can be seen in the continuity equation

$$\frac{d\rho}{dt} = -\rho \nabla \cdot \mathbf{v}, \quad (3.16)$$

where t is time and $\mathbf{v}$ is the velocity vector. Using the first form of the SPH derivative, given by Equation (3.14), the continuity equation becomes

$$\frac{d\rho_a}{dt} = \rho_a \sum_{b=1}^N \mathbf{v}_{ab} \cdot \nabla_a W_{ab} \frac{m_b}{\rho_b}, \quad (3.17)$$

where $\mathbf{v}_{ab} = \mathbf{v}_a - \mathbf{v}_b$. Using the second form of the SPH derivative, given by Equation (3.15), the continuity equation becomes

$$\frac{d\rho_a}{dt} = \sum_{b=1}^N m_b \mathbf{v}_{ab} \cdot \nabla_a W_{ab}. \quad (3.18)$$

Both Equations (3.17) and (3.18) are acceptable forms of the continuity equation.

3.2 Governing equations

The equations of motion for a fluid flow can be expressed by the Navier-Stokes equations. In Lagrangian form these can be represented by

$$\frac{d\mathbf{v}}{dt} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\mathbf{v} + \mathbf{g}, \quad (3.19)$$

where $\mathbf{v}$ is again the velocity vector, t is time, ρ is density, p is pressure, ν is the kinematic viscosity coefficient and $\mathbf{g}$ is an externally applied force per unit mass, corresponding to the vector $(0, 0, -g)$ and representing gravity. Equation (3.19) must be solved subject to a continuity equation, discussed above, which is defined by

$$\frac{d\rho}{dt} = -\rho\nabla \cdot \mathbf{v}. \quad (3.20)$$

The continuity equation has already been given in two variations of SPH form, details being provided in Equations (3.17) and (3.18). Although it is possible to calculate the density using the summation method given in Equation (3.10), this causes problems at walls and free-surfaces. Such difficulties arise because particles near a boundary are surrounded by fewer other particles when compared to those away from a boundary. As a result, the summation is over fewer particles and therefore the calculated density is significantly lower than it should be for these particles. In order to avoid such difficulties, it is preferable to use the continuity equation to calculate the rate of change of density and use this information to time-march the density. This means that the initial density that is prescribed to each particle only changes when particles are in relative motion to one another. The usual form for the rate of change of density is given by Equation (3.18).

The momentum equation (Equation (3.19)) can be transcribed into SPH form in a similar way to the continuity equation. The body force, $\mathbf{g}$, is an external force, so does not need to be calculated by interpolation between particles. Therefore, it can simply be added on to the momentum equation for each particle after the other calculations have been performed. In the following derivation, both the viscosity and body force terms shall be ignored.

Originally, the momentum equation was written in SPH form using the method for first derivatives given in Equation (3.11) by Gingold & Monaghan (1977). This is given by

$$\frac{d\mathbf{v}_a}{dt} = -\frac{1}{\rho_a} \sum_{b=1}^N p_b \nabla_a W_{ab} \frac{m_b}{\rho_b}. \quad (3.21)$$

However, the derivative does not vanish, as it should, when the pressure is constant. In addition, this form does not conserve linear or angular momentum, since it does not ensure that the force exerted on particle a from particle b is equal and opposite to the force exerted on particle b from particle a . This can be demonstrated by considering Equation (3.21) for a single particle pair and multiplying by each particle's mass to equate the forces between the two particles:

$$\frac{m_a m_b p_a}{\rho_a \rho_b} \nabla_a W_{ab} = -\frac{m_a m_b p_b}{\rho_a \rho_b} \nabla_b W_{ab}. \quad (3.22)$$

This equality cannot be true unless $p_a = p_b$, since $\nabla_a W_{ab}$ is always equal to $-\nabla_b W_{ab}$. Since the pressure calculated on each of the two particles is not necessarily the same, Equation (3.21) is not suitable for determining the acceleration. Instead, the momentum equation can be reconstructed by using the identity

$$\frac{\nabla p}{\rho} = \nabla \left(\frac{p}{\rho} \right) + \frac{p}{\rho^2} \nabla \rho, \quad (3.23)$$

which has been chosen to keep pressure forces symmetrical and preserve linear and angular momentum. In SPH form, this gives the momentum equation as

$$\frac{d\mathbf{v}_a}{dt} = -m_b \sum_{b=1}^N \left(\frac{p_a}{\rho_a^2} + \frac{p_b}{\rho_b^2} \right) \nabla_a W_{ab}, \quad (3.24)$$

which is the form in which it is usually written in SPH. Another form which is sometimes used, for example by Colagrossi & Landrini (2003) when studying interfacial flows, is

$$\frac{d\mathbf{v}_a}{dt} = -m_b \sum_{b=1}^N \left(\frac{p_a + p_b}{\rho_a \rho_b} \right) \nabla_a W_{ab}. \quad (3.25)$$

Finally, reintroducing the viscosity and body force terms gives the SPH momentum equation in full as

$$\frac{d\mathbf{v}_a}{dt} = -m_b \sum_{b=1}^N \left(\frac{p_a}{\rho_a^2} + \frac{p_b}{\rho_b^2} - \Pi_{ab} \right) \nabla_a W_{ab} + \mathbf{g}, \quad (3.26)$$

where Π_{ab} is an SPH expression for the viscosity, which is dealt with in § 3.4. The momentum equation is integrated through a time-marching procedure to calculate the

particles' velocities. The velocities are then integrated by a similar method to calculate the particles' positions.

3.3 The equation of state

In SPH the pressure is calculated through an equation of state. This is the last equation needed to close the system of equations and defines the pressure in terms of the local density. In most computational fluid dynamics methods water is treated as an incompressible fluid. This is appropriate because the bulk motions in the flow are usually very small compared to the speed of sound, implying a very low Mach number. However, in SPH the particles are driven by local density gradients and so it is necessary for water to be treated as a slightly compressible fluid. If the real speed of sound for water was used, the time steps would need to be prohibitively small in order for the model to remain stable. In practice, the size of the time steps is limited by a CFL condition which is discussed in § 3.11. Therefore, an artificial speed of sound is introduced. This is chosen so that it is high enough to keep the density fluctuations negligible, but low enough that the time step is sufficiently large to be practical.

An equation of state commonly used for free-surface flows, and which will be used herein, is that given by Batchelor (1974). When the atmospheric pressure is negligible the equation of state is given by

$$p = B \left(\left(\frac{\rho}{\rho_0} \right)^\gamma - 1 \right), \quad (3.27)$$

where ρ_0 is the reference density, taken to be 1000kgm^{-3} for water, and B is a parameter chosen to ensure that the speed of sound is large enough to keep the density fluctuations small. γ is an empirically determined constant relating to the stiffness of the fluid, taken as $\gamma = 7$ for most SPH applications involving free-surface water flows. Since the speed of sound, c , in a compressible fluid is given by

$$c = \sqrt{\partial p / \partial \rho}, \quad (3.28)$$

differentiating Equation (3.27) with respect to the density gives

$$c^2 = \frac{\gamma B}{\rho_0} \quad (3.29)$$

at the reference density ($\rho = \rho_0$).

By considering the balance of forces in the momentum equation (Morris *et al.* (1997)), it can be shown that the density fluctuations are proportional to the square of the Mach number,

$$\frac{|\delta\rho|}{\rho} \sim \frac{v_{\max}^2}{c^2}, \quad (3.30)$$

where $v_{\max}$ is an estimate of the maximum speed of the flow. Previous work by Monaghan (1994) showed that it is usually desirable to keep density fluctuations below 1%, and therefore $v_{\max}/c$ must be kept under 0.1. Combining this constraint on the density fluctuations with Equations (3.29) and (3.30), implies that

$$B = \frac{100\rho_0 v_{\max}^2}{\gamma}. \quad (3.31)$$

3.4 Viscosity

Monaghan & Gingold (1983) first introduced a term, known as artificial viscosity, when dealing with clouds impacting on each other supersonically. The term helped to stop unrealistic penetration and improved the handling of simulations involving shocks. Artificial viscosity was also used by Lucy (1977) to reduce a build up of acoustic energy from integration errors in SPH. In free-surface flows, applying an artificial viscous term is beneficial as it increases the stability of a simulation. The term is added into the momentum equation (Equation (3.26)), with $\Pi_{ab}\nabla_a W_{ab}$ being given by

$$\Pi_{ab}\nabla_a W_{ab} = \begin{cases} \frac{-\alpha h(c_a+c_b)}{\rho_a+\rho_b} \frac{\mathbf{v}_{ab} \cdot \mathbf{r}_{ab}}{\mathbf{r}_{ab}^2 + \epsilon \bar{h}_{ab}^2} \nabla_a W_{ab} & \mathbf{v}_{ab} \cdot \mathbf{r}_{ab} < 0 \\ 0 & \mathbf{v}_{ab} \cdot \mathbf{r}_{ab} > 0 \end{cases}. \quad (3.32)$$

where $\mathbf{v}_{ab} = \mathbf{v}_a - \mathbf{v}_b$, $\mathbf{r}_{ab} = \mathbf{r}_a - \mathbf{r}_b$, where $\mathbf{r}$ is the position vector, c_a and c_b represent the speed of sound for particles a and b respectively, α is a small constant described in more detail below, and $\bar{h}_{ab}^2$ is the average smoothing length for particles a and b . The term, $\epsilon \bar{h}_{ab}^2$, is added into Equation (3.32) to prevent errors occurring when $\mathbf{r}_{ab}$ is close to

zero. The particle symmetry ensures that linear and angular momentum are conserved and that for rigid rotation the viscosity disappears.

The artificial viscosity equation, Equation (3.32), has been constructed so that as the number of SPH particles tends to infinity and the summations become integrals, the artificial viscosity tends towards the form of the real viscosity, as it is in the Navier-Stokes equations. An analysis of this is given by Monaghan & Kos (1999), where it is shown that $\alpha h c$ is proportional to the real kinematic viscosity coefficient, ν . α is usually set in the range 0.001-0.1, but different authors give different acceptable ranges. Varying α within the acceptable range should not have a significant effect on the flow.

The above formulation of the artificial viscosity term was originally developed for flows involving shocks, or step changes in pressure. However, for flows where the shear viscosity dominates, Equation (3.32) can be too dissipative. Balsara (1995) suggested that the generation of spurious entropy and hence, artificial viscous diffusion, could be reduced by modifying Equation (3.32) to give

$$\Pi_{ab} \nabla_a W_{ab} = \frac{-\alpha h (c_a + c_b)}{\rho_a + \rho_b} \frac{k_a + k_b}{2} \frac{\mathbf{v}_{ab} \cdot \mathbf{r}_{ab}}{\mathbf{r}_{ab}^2 + \epsilon \bar{h}_{ab}^2} \nabla_a W_{ab}, \quad (3.33)$$

where

$$k_a = \frac{|\nabla \cdot \mathbf{v}_a|}{|\nabla \cdot \mathbf{v}_a| + |\nabla \times \mathbf{v}_a| + 10^{-4} c_a / h} \quad (3.34)$$

and k_b is defined similarly. More recently, Colagrossi & Landrini (2003) replaced Equation (3.34) with

$$k_a = \frac{|\nabla \cdot \mathbf{v}_a|}{|\nabla \cdot \mathbf{v}_a| + \sqrt{S_a^{ij} S_a^{ij}} + 10^{-4} c_a / h}, \quad (3.35)$$

where S_a^{ij} is written in tensor notation and represents the rate of strain tensor defined by

$$S_a^{ij} = \frac{1}{2} \left(\frac{\partial v_a^i}{\partial x_a^j} + \frac{\partial v_a^j}{\partial x_a^i} \right). \quad (3.36)$$

Colagrossi & Landrini (2003) found that this gave better results for slightly compressible fluids. The SPH form of Equations (3.35) and (3.36) can be found by applying Equation (3.14) (whilst adopting tensor notation), noting that

$$\frac{\partial v_a^i}{\partial x_a^j} = \sum_{b=1}^N \left(v_{ab}^i \frac{\partial W_{ab}}{\partial x_a^j} \right) \frac{m_b}{\rho_b}. \quad (3.37)$$

An alternative representation of viscosity in SPH was given, in a more realistic form, by Morris *et al.* (1997), who originally used it for modelling low Reynolds number flows without a free-surface. Here, $\Pi_{ab} \nabla_a W_{ab}$ in the momentum equation (Equation (3.26)) is replaced by

$$\Pi_{ab} \nabla_a W_{ab} = \frac{\mu_a + \mu_b}{\rho_a \rho_b} \frac{\mathbf{v}_{ab}}{\mathbf{r}_{ab}^2 + \epsilon \bar{h}_{ab}^2} \mathbf{r}_{ab} \cdot \nabla_a W_{ab}, \quad (3.38)$$

where μ_a and μ_b are the real dynamic viscosity coefficients for particles a and b respectively. For water $\mu = 10^{-3}$ for all particles. In this form, linear momentum is conserved exactly, but angular momentum is only approximately conserved (Morris *et al.* (1997)). As a result, this approach is only appropriate for flows in which conserving angular momentum exactly is not of vital importance. Fortunately, this is true for all flows with large deformations in the free-surface, and certainly for all flows considered within this thesis. Finally, it is also important to note that the viscosity model introduced in Equation (3.38) was the preferred solution adopted by Violeau & Issa (2007) in their studies of related free-surface flows.

3.4.1 Viscosity model tests

In order to quantify the effectiveness of the viscosity models described in the previous section, simple benchmark test cases have been carried out. The first of these considers Poiseuille flow, in which a constant driving force is applied to flow between two infinite parallel walls. The second test concerns Couette flow, where the flow between the two infinite walls is instead driven by the movement of one of the walls with a constant velocity. The test cases are carried out at a low Reynolds number to ensure that the flow is laminar. In both cases, the fluid between the walls is initially at rest. This allows both the development of the flow and the final steady-state predictions from the SPH model to be compared with theory. In addition, a convergence test has been carried out to establish the order of accuracy for the SPH model whilst simulating Poiseuille flow.

Poiseuille flow

Poiseuille flow occurs between two infinite parallel walls, located at $y = 0$ and $y = l$, when a constant driving force, F , is applied to the flow between them. The flow is initially at rest, but the driving force (equivalent to a pressure gradient along the channel) causes the flow to move. Shear stress arising from the no-slip conditions on the two walls ensures that flow at the walls remains stationary, whilst the flow velocity in the centre of the channel increases. Eventually the driving force is balanced by the opposing viscous force and a steady-state is reached in which a parabolic velocity profile is established across the channel. For the SPH simulations, the accuracy of the viscosity model is therefore fundamental in defining the correct velocity profile.

Theoretically, the velocity profile can be obtained from the Navier-Stokes equations. The only forces present are the viscous force and the driving body force. This gives

$$0 = \nu \frac{\partial^2 u}{\partial y^2} + F \quad (3.39)$$

where ν is the kinematic viscosity of water and u is the velocity in the x -direction. Since both walls ensure a no-slip boundary condition, the flow velocity, u , is zero at both $y = 0$ and $y = l$. Integrating Equation (3.39) twice and applying the boundary conditions gives the final steady-state solution,

$$u(y) = -\frac{F}{2\nu}y(y-l), \quad (3.40)$$

with the maximum velocity at the centre of the channel given by

$$u_{\max} = \frac{F}{8\nu}l^2. \quad (3.41)$$

A time-dependent solution is also available, defining the velocity profile across the channel before the steady-state is established. This is given by Morris *et al.* (1997) as

$$u(y) = -\frac{F}{2\nu}y(y-l) - \sum_{n=0}^{\infty} \frac{4Fl^2}{\nu\pi^3(2n+1)^3} \sin\left(\frac{\pi y}{l}(2n+1)\right) \exp\left(-\frac{(2n+1)^2\pi^2\nu}{l^2}t\right). \quad (3.42)$$

For the viscosity tests carried out herein, a channel width of $y = 0.01\text{m}$ was used, with a driving force per unit mass of $F = 8 \times 10^{-6}\text{ms}^{-2}$. According to Equation (3.41)

this gives the maximum velocity at the centre of the channel as $u_{max} = 10^{-4}\text{ms}^{-1}$. Taking the kinematic viscosity as the standard value for water ($\nu = 10^{-6}\text{m}^2\text{s}^{-1}$) and using the channel width as a length scale, this gives the Reynolds number, Re , as

$$\text{Re} = \frac{u_{max}l}{\nu} = 1.0. \quad (3.43)$$

In the SPH simulations, the water in the channel is represented by 39 SPH particles across the width. The model uses periodic boundaries conditions, such that when SPH particles leave the domain at one end, they return at the other end. Effectively, this implies that the channel is infinitely long and consequently the actual length of the channel in the model was not important. The length chosen for this model setup was 0.01m, making the visible section of channel square. An initial particle spacing of $\Delta x = \Delta y = 2.5 \times 10^{-3}\text{m}$ was used with the particles being placed on a Cartesian grid, giving a total SPH particle number of 1560. The initial setup is shown in Figure 3.2 below. Besides from varying the viscosity model, all other numerical parameters used for these SPH simulations were kept constant with key elements being the stationary ghost boundary particle representation of Violeau & Issa (2007), a quartic kernel (Violeau & Issa (2007)), a Shepard filter for smoothing the density field (Dalrymple & Rogers (2006)) and a time marching scheme described by Monaghan (2007). This combination of parameters is identified in Chapter 4 as being optimal for the description of free-surface flows.

First, the realistic viscosity model of Morris *et al.* (1997) (subsequently referred to as the *Realistic* formulation) was tested on this flow. Since this model uses the real viscosity of water, there was no need to tune any parameters. The velocity profile obtained using the *Realistic* viscosity model is presented in Figure 3.3. The results are shown at four times before reaching the steady-state, $t = 2\text{s}$, $t = 5\text{s}$, $t = 10\text{s}$ and $t = 20\text{s}$, and once after reaching the steady-state, $t = 120\text{s}$. The SPH velocity profile remains very close to the theoretical profile throughout the development to the steady-state, eventually settling into the steady-state with just a 2.2% error in the maximum velocity. The *Realistic* viscosity model did not require any tuning of parameters. This is considered to be very important, since it implies the model can be used on other flows without first having to adapt the model to the particular flow. This is imperative in more complex flows, where

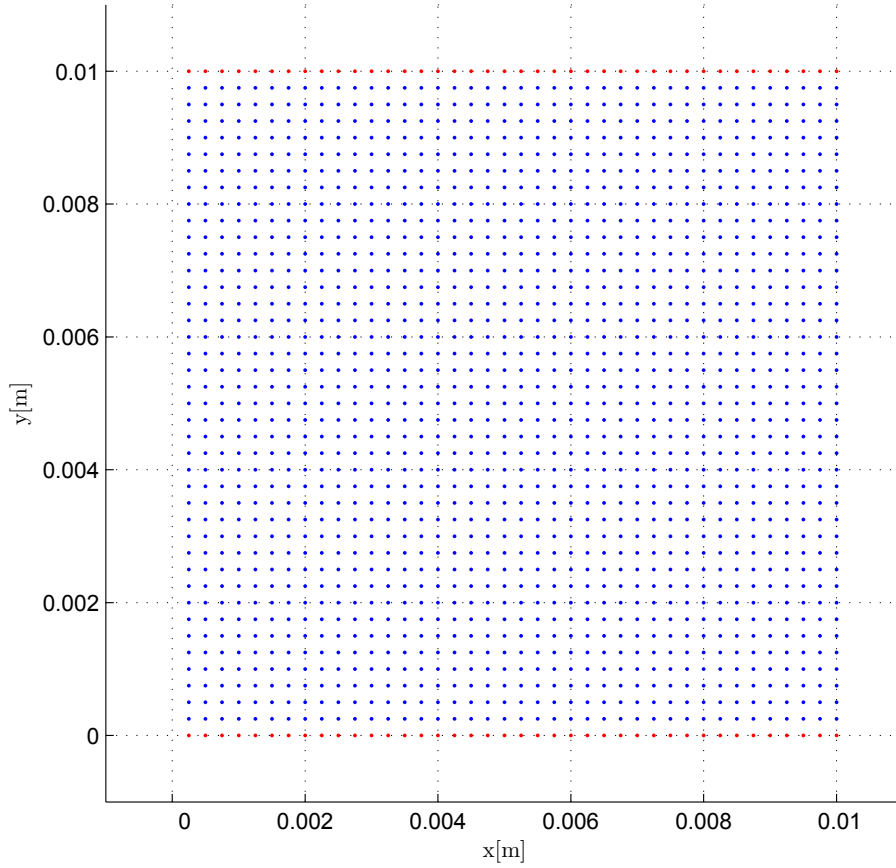


Figure 3.2: Numerical setup of the channel used to demonstrate Poiseuille flow and Couette flow. — SPH particles and — boundary particles.

different flow regimes may occur simultaneously within a single numerical domain.

In a second set of tests, the artificial viscosity model of Monaghan & Gingold (1983) (subsequently referred to as the *Artificial* formulation) was employed. It was stated in the previous section of this chapter that the parameter, α , should be set in the range 0.001-0.1. However, this viscosity model was originally designed to model strong shocks and is ideally suited to fast moving free-surface flows (Morris *et al.* (1997)). In order to obtain a good velocity profile for the Poiseuille flow at low Reynolds number it was necessary to substantially increase and tune the value of α . Indeed, Morris *et al.* (1997) also found that this method produced inaccurate velocity profiles for low Reynolds number flows. Eventually, using $\alpha = 5.5$ was found to produce a velocity profile close to the theoretical profile. The SPH model predictions are presented and compared to theory in Figure 3.4. This is presented at the same four times before reaching the

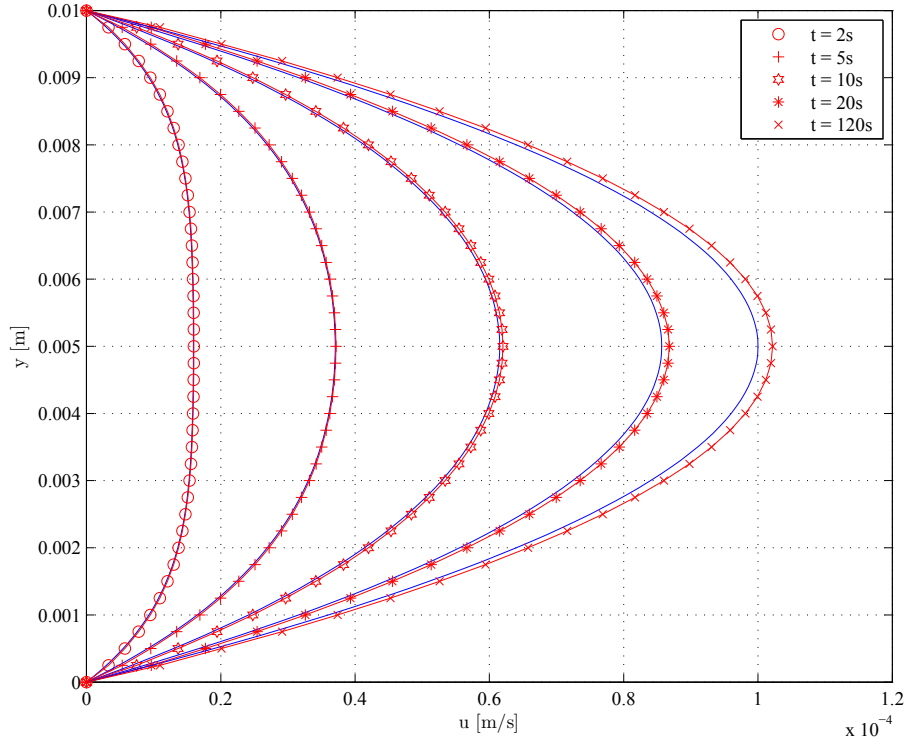


Figure 3.3: Time varying Poiseuille flow tending towards a steady-state, comparisons between results obtained with the SPH model using the *Realistic* viscosity method and theory: — SPH model, — theory. Results are presented at five times, the final time being after the steady-state was established.

steady-state and again after the steady-state has established as in the previous test. As the velocity profile begins to develop, the SPH model follows the developing theoretical profile very closely. However, by the time $t = 10\text{s}$, the SPH particles have fallen behind the theoretical velocity, suggesting that the viscosity model is overly opposing the driving force and is too dissipative. This continues through to $t = 20\text{s}$. However, once the steady-state is established, the SPH velocity profile is actually slightly faster than the theoretical profile, with only a 0.9% error in the maximum velocity at the centre of the channel. The variability in the comparison throughout time is probably due to the model not being particularly appropriate for this type of flow, especially as such a high value of α was necessary to obtain a steady-state value of the maximum velocity close to the theoretical value. Nevertheless, once α had been tuned, this viscosity model was able to establish the Poiseuille flow velocity profile in reasonably close agreement with the theoretical profile.

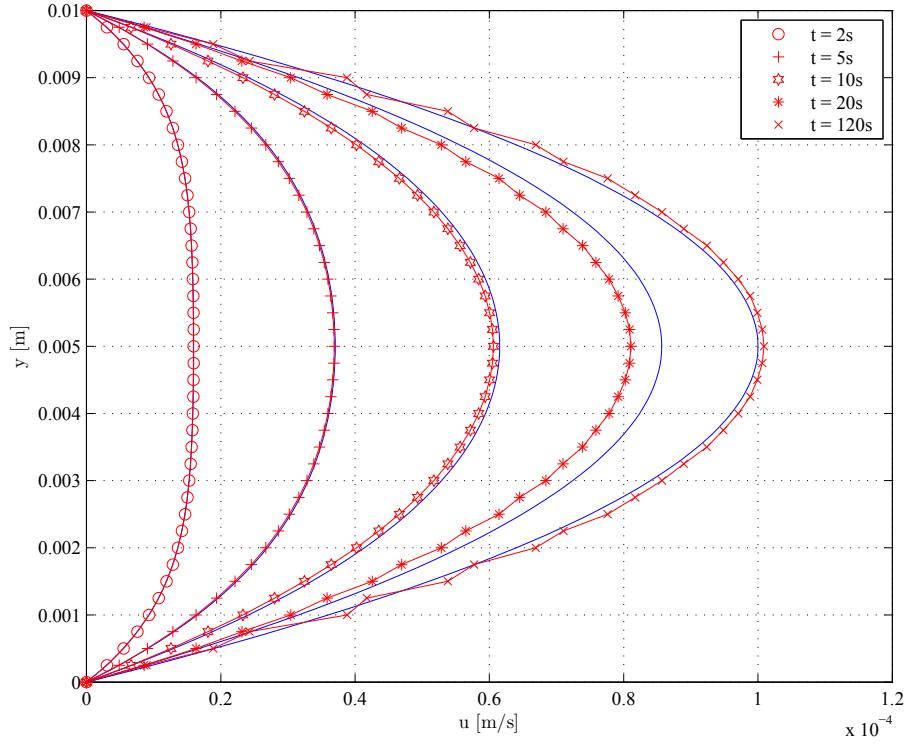


Figure 3.4: Time varying Poiseuille flow tending towards a steady-state, comparisons between results obtained with the SPH model using the *Artificial* viscosity method and theory: — SPH model, — theory. Results are presented at five times, the final time being after the steady-state was established.

Finally, the improved artificial viscosity model of Colagrossi & Landrini (2003) (subsequently referred to as the *Improved Artificial* formulation) was put to the test. This viscosity model had been shown to give improved results for weakly compressible fluids during free-surface flow simulations by Colagrossi & Landrini (2003). Despite this, a problem arose when attempting to use this model for enclosed channel flow simulations, such as Poiseuille flow. The errors were caused by the introduction of the k parameters in Equation (3.35) which uses the divergence of the velocity field as the numerator. This term should only be non-zero because the SPH representation treats the water as being weakly compressible. However, this term was found to be extremely small for the slow-moving Poiseuille flow, perhaps because the SPH particles remain in positions similar to their initial formation throughout the simulation, rather than becoming disordered as they would for higher Reynolds number flows. This made it exceptionally difficult to use this viscosity model at all, and in order to obtain a reasonable representation of

the Poiseuille flow it was necessary to increase the value of α to 160, several orders of magnitude higher than the recommended range. The velocity profile obtained using the *Improved Artificial* viscosity model is shown in Figure 3.5 at the same five times as in the previous viscosity tests. The development of the parabolic velocity profile is clearly not quite right for this viscosity model. Although it eventually reaches a steady-state with close to the same maximum velocity as the theory, the shape of the profile is not identical. This is probably due to the huge increase in α necessary to achieve the correct maximum velocity, making the viscosity model behave quite differently to the way in which it was intended. Rather than dismissing this viscosity model altogether, it can merely be said that it is even less suited to this sort of flow than the original *Artificial* viscosity method. Since Colagrossi & Landrini (2003) found the method gave improved results for free-surface flows, the *Improved Artificial* viscosity model will be considered further in Chapter 4.

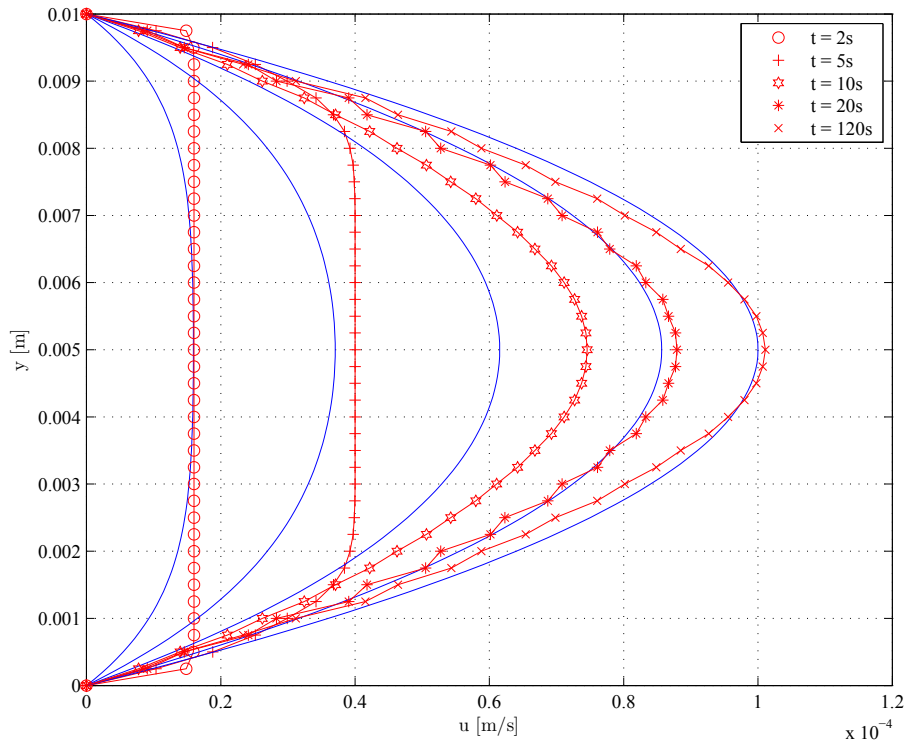


Figure 3.5: Time varying Poiseuille flow tending towards a steady-state, comparisons between results obtained with the SPH model using the *Improved Artificial* viscosity method and theory: — SPH model, — theory. Results are presented at five times, the final time being after the steady-state was established.

The development of the shear stress at the wall, τ_w , as the Poiseuille flow tends towards a steady-state was also considered with calculations based upon each of the three viscosity models. The results of this are presented in Figure 3.6 where the shear stress is normalised and compared to classical theory. In making these comparisons it should be noted that the normalised value of the theoretical steady-state shear stress is slightly below 1.0 since the calculations were performed using the velocity one particle spacing away from the wall. Although the shear stress predicted by the *Artificial* viscosity method is the closest of the three viscosity models to the theoretical shear stress during the development phase, it should be remembered that the model parameter, α , was heavily tuned to achieve this result. The graphs also show that neither the *Artificial* nor the *Improved Artificial* viscosity methods allow the Poiseuille flow to reach a truly steady-state; instead, the shear stress oscillates around the theoretical steady-state value. In contrast, the *Realistic* viscosity method settles into a steady-state at a similar time to the theory predicts, albeit at a slightly higher value. The explanation for this increased value is likely to be the relatively low resolution of the SPH particles, particularly close to the boundary.

The *Realistic* viscosity model can certainly be considered the most favourable SPH viscosity model for this type of flow and, importantly, did not require any tuning of parameters. Nevertheless, all three viscosity models have been shown to be able to produce reasonably accurate representations of steady-state Poiseuille flow, provided that the parameter, α , is tuned correctly for the *Artificial* and *Improved Artificial* viscosity models. In addition, it is possible that the artificial viscosity methods may be better suited to free-surface flows with higher Reynolds numbers. Since these latter flows are the main focus of the work presented herein, the three viscosity models will be further investigated in Chapter 4 where their performance is considered in respect of solitary wave propagation. Taken together, the results presented on Figures 3.3 to 3.6 confirm that provided the present SPH model incorporates an accurate viscosity model, Poiseuille flow can be accurately reproduced.

Couette flow

In common with Poiseuille flow, Couette flow also occurs between two infinite parallel walls located at $y = 0$ and $y = l$. However, instead of driving the flow with a constant

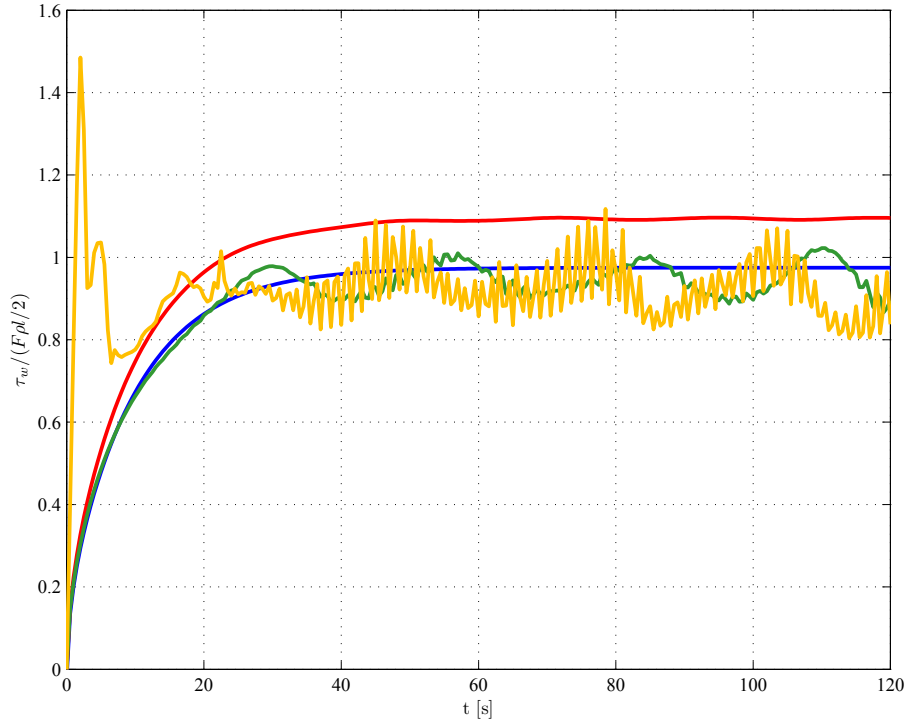


Figure 3.6: Shear stress at the wall in time varying Poiseuille flow tending towards a steady-state, comparisons between results obtained with the SPH model using the *Realistic* viscosity method, the *Artificial* viscosity method, the *Improved Artificial* viscosity method and theory: — theory, — SPH model with the *Realistic* viscosity method, — SPH model with the *Artificial* viscosity method and — SPH model with the *Improved Artificial* viscosity method.

force, flow which is initially at rest is driven through the movement of the top wall with a constant velocity. Numerically, this movement begins instantaneously at the start of the simulation and allows the development of the velocity profile to be compared to theory. Shear stress is generated between the top wall and the water below, resulting in the water on this boundary moving at the same speed as the wall, whilst water at the stationary wall on the opposite side of the channel remains still. Eventually this leads to a linear velocity profile across the width of the channel. As this steady-state develops, the time-varying velocity profile is influenced by the water's viscosity, so again the SPH viscosity model is put to the test. The theoretical velocity profile can again be obtained from the Navier-Stokes equations. In this case, the only remaining term is the viscous term, since there is no flow across the channel, no change of velocity in the x -direction, no pressure gradient and no body forces. The governing equation then reduces to

$$0 = \frac{\partial^2 u}{\partial y^2} \quad (3.44)$$

and the flow is entirely shear driven. Integrating twice and applying the boundary conditions, which state that the fluid velocity on the top wall, $y = l$, is the same as the instantaneous velocity, u_0 , and on the bottom wall is zero, gives the velocity profile across the channel as

$$u(y) = \frac{u_0 y}{l}. \quad (3.45)$$

Again, a time-dependent solution is available, defining the velocity profile across the channel before the steady-state is established. This is given by Morris *et al.* (1997) as

$$u(y) = \frac{u_0 y}{l} + \sum_{n=1}^{\infty} \frac{2u_0}{n\pi} (-1)^n \sin\left(\frac{n\pi}{l}y\right) \exp\left(-\nu \frac{n^2\pi^2}{l^2}t\right). \quad (3.46)$$

This equation shows that although the final steady-state solution only depends on the shear profile generated from the movement of the top boundary, the time-dependent solution also depends on the viscosity. This implies that the viscosity model used within the SPH model influences the development of the steady-state.

For the Couette flow simulations carried out herein, the channel dimensions and all the SPH parameters are the same as those used for the Poiseuille flow simulations described above. The velocity of the top wall was set to 10^{-4}ms^{-1} , giving $\text{Re} = 1.0$.

Initially, the performance of the *Realistic* viscosity model was assessed for this flow. The resulting velocity profile is presented in Figure 3.7. This is presented at four times before reaching the steady-state, $t = 2\text{s}$, $t = 5\text{s}$, $t = 10\text{s}$ and $t = 20\text{s}$, and once after reaching the steady-state, $t = 120\text{s}$. This shows that the SPH velocity profile perfectly matches the theoretical profile at all the times presented. Since the velocity profile is accurately predicted, this also implies that the shear stress is correct. Again, this emphasises the quality of the *Realistic* viscosity model for use with this type of flow, especially as no parameters require tuning.

Second, the *Artificial* viscosity model was also tested for Couette flow. The velocity profile is presented at the same five times again in Figure 3.8. Since the Reynolds number is the same as that used for the Poiseuille flow, it was assumed that the value of α could be left the same. Indeed, this seems to have provided a satisfactory representation of the

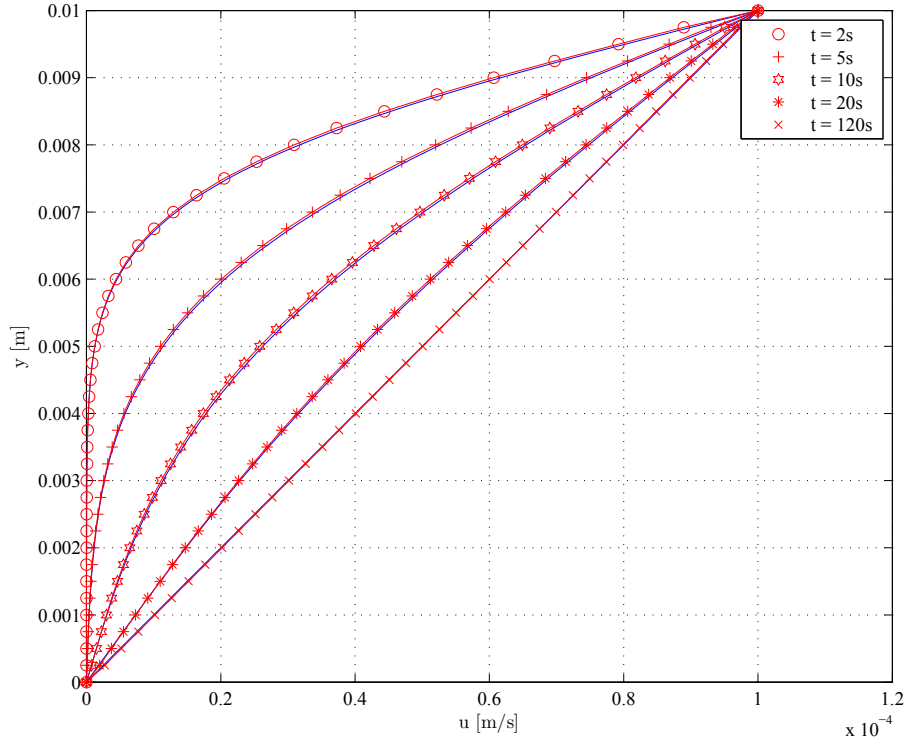


Figure 3.7: Time varying Couette flow tending towards a steady-state, comparisons between results obtained with the SPH model using the *Realistic* viscosity method and theory: — SPH model, — theory. Results are presented at five times, the final time being after the steady-state was established.

velocity profile as the steady-state develops, with only a slight lag behind the theoretical profile. The steady-state solution matches the theoretical solution as expected.

The *Improved Artificial* viscosity model was shown to perform badly for the Poiseuille flow case due to the divergence of velocity term, which is not suited to this type of flow. Nevertheless, this viscosity model was tested again for Couette flow, keeping the value of α the same as in the Poiseuille flow test case. The velocity profile is presented at the same five times as for the previous tests in Figure 3.9. As for Poiseuille flow, the development of the velocity profile with time does not compare at all well with the theoretical velocity profile. More surprisingly, the final steady-state solution is not correct. This is probably because α was not re-tuned specifically for this test case and the extreme high value is not really appropriate for use in the model. However, it should be remembered that this viscosity model was not designed for this type of flow. Overall, the data presented on Figures 3.7 to 3.9 confirm that with an appropriate viscosity model, the present SPH

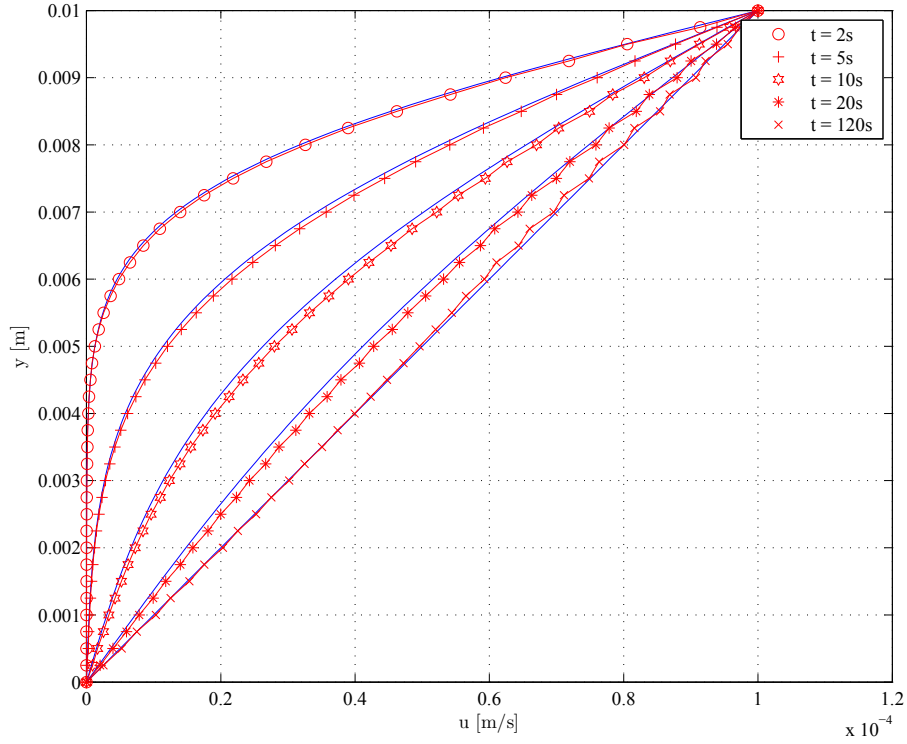


Figure 3.8: Time varying Couette flow tending towards a steady-state, comparisons between results obtained with the SPH model using the *Artificial* viscosity method and theory: — SPH model, — theory. Results are presented at five times, the final time being after the steady-state was established.

code can accurately predict Couette flow.

Convergence test

A series of simulations have also been carried out to establish the order of convergence for the Poiseuille flow test case. This involved varying the number of SPH particles used to represent the flow across the width of the channel, systematically reducing the particle spacing. Since the *Realistic* viscosity model performed best in the viscosity tests described above, this model was used again here. All other SPH model parameters remain unchanged. The convergence test was carried out with SPH particle numbers across the channel ranging from 20 to 160, in increments of 20 particles. The order of convergence was assessed by examining the maximum velocity reached at the centre of the channel once the flow had reached a steady-state. Since the driving force was kept at $F = 8 \times 10^{-6} \text{ms}^{-2}$, the theoretical maximum velocity remained $u_{max} = 10^{-4} \text{ms}^{-1}$.

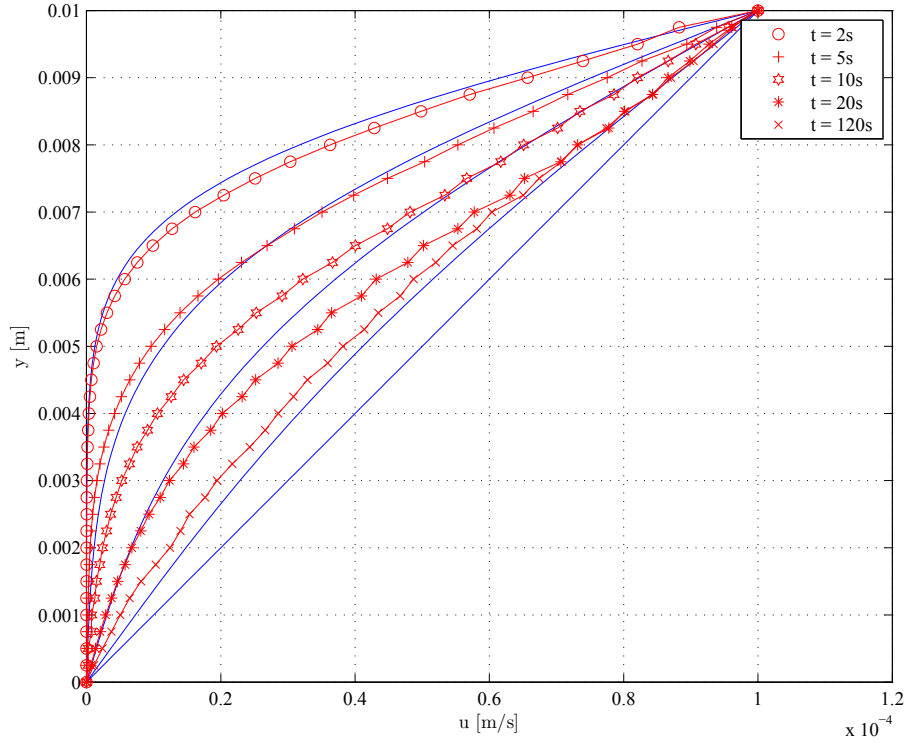


Figure 3.9: Time varying Couette flow tending towards a steady-state, comparisons between results obtained with the SPH model using the *Improved Artificial* viscosity method and theory: — SPH model, — theory. Results are presented at five times, the final time being after the steady-state was established.

The first stage in analysing the order of convergence for the Poiseuille flow test case was to check that the error in the maximum velocity reduces as the particle spacing was reduced. For each simulation, the number of SPH particles used to represent the water across the channel width and their initial particle spacing, Δx , are given in Table 3.1. The maximum velocity obtained at the centre of the channel and the percentage error compared to the theoretical maximum velocity are also presented in this table.

Data describing the maximum velocity are also presented in Figures 3.10 and 3.11. In Figure 3.10 the maximum velocity is plotted against the number of SPH particles across the channel width. This shows a logarithmic relationship between the SPH particle number and the predicted value of the maximum velocity; as the SPH particle number is increased, the predicted value of the maximum velocity tends towards an asymptote. This value is known as the asymptotic numerical solution or the continuum solution.

In Figure 3.11 the velocity residual (the error in the SPH predicted maximum ve-

Number of SPH particles across the channel width	SPH particle spacing [m]	Maximum velocity at the centre of the channel [m/s]	Percentage error
20	5.00×10^{-4}	1.0314×10^{-4}	3.14%
40	2.50×10^{-4}	1.0216×10^{-4}	2.16%
60	1.67×10^{-4}	1.0184×10^{-4}	1.84%
80	1.25×10^{-4}	1.0168×10^{-4}	1.68%
100	1.00×10^{-4}	1.0158×10^{-4}	1.58%
120	8.33×10^{-5}	1.0152×10^{-4}	1.52%
140	7.14×10^{-5}	1.0147×10^{-4}	1.47%
160	6.25×10^{-5}	1.0144×10^{-4}	1.44%

Table 3.1: Maximum velocity and percentage error relative to the theoretical predictions for steady-state Poiseuille flow simulations with varying SPH particle resolutions.

locity compared to the exact solution) is plotted against initial particle spacing, Δx . The predicted values of the velocity residual form a linear trend when plotted against the particle spacing, suggesting that the order of convergence for this test case is approximately equal to one. The dashed extension of this straight line, also plotted on Figure 3.11, extrapolates the results to the hypothetical case with zero particle spacing, representing the continuum solution. The velocity residual of the continuum solution is approximately 0.012, giving a value of $1.012 \times 10^{-4} \text{ms}^{-1}$ for the maximum velocity. These results will also be confirmed through a thorough numerical analysis, using Richardson extrapolation to calculate the continuum solution.

First, the order of convergence, p , can be confirmed by calculating its value from subsets of the simulations described in Table 3.1. This is most easily done by considering sets of three simulations where the particle spacing has reduced by a factor of two between each of the simulations in the set. Two such subsets exist, the first consisting of the simulations with 20, 40 and 80 particles, and the second consisting of the simulations with 40, 80 and 160 particles. Although it is only necessary to perform the calculations on one set of simulations, the second will provide confirmation that the results from the first set are representative of the test case as a whole. The method for calculating

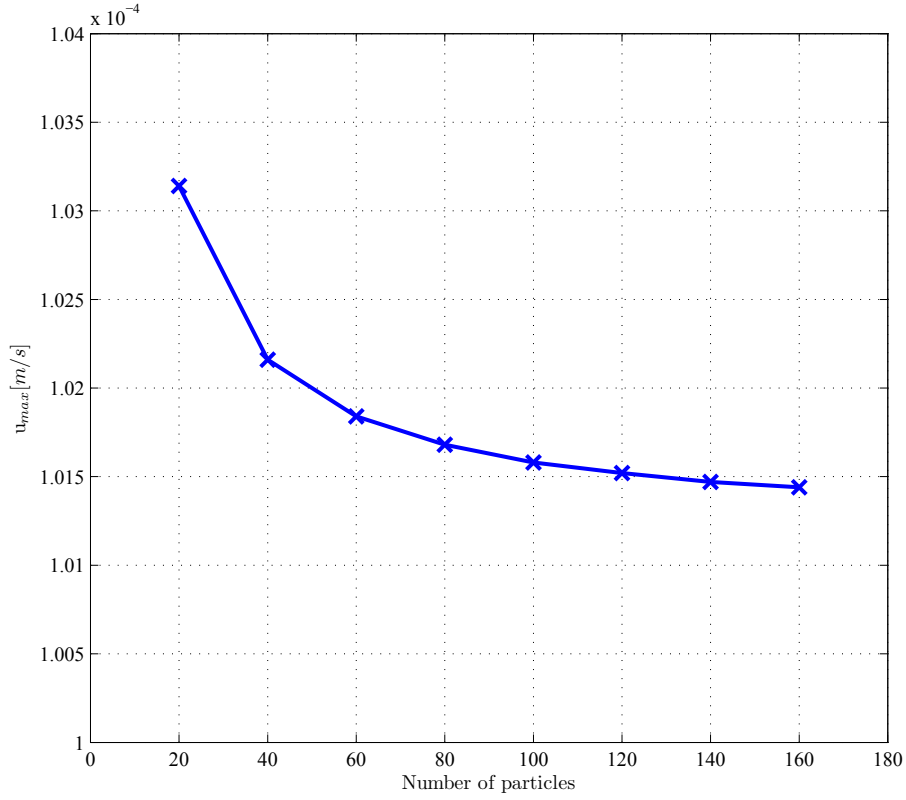


Figure 3.10: Solution for u_{max} (Poiseuille Flow) obtained with varying SPH particle numbers across the width of the channel. u_{max} is plotted against particle number. — SPH results.

the order of convergence for grid based methods can also be applied to SPH since the flow equations (Equations (3.18) and (3.26)) are also discretized onto a set of discrete points. For sufficiently fine particle resolutions, the discretization method results in a truncation error proportional to the leading term in the Taylor series. This error can be approximated as

$$E(s) \approx C s^p + H, \quad (3.47)$$

where s is a measure of the particle spacing, p is the order of the scheme, C is a constant independent of s and H stands for higher order terms (Ferziger & Peric (2002)). The discretization error, $E(s)$, is a measure of the difference between the exact solution of the flow equations, Φ , and the solution of the discretized equations, ϕ_s , referred to above as the continuum solution. This implies that the exact solution is given by

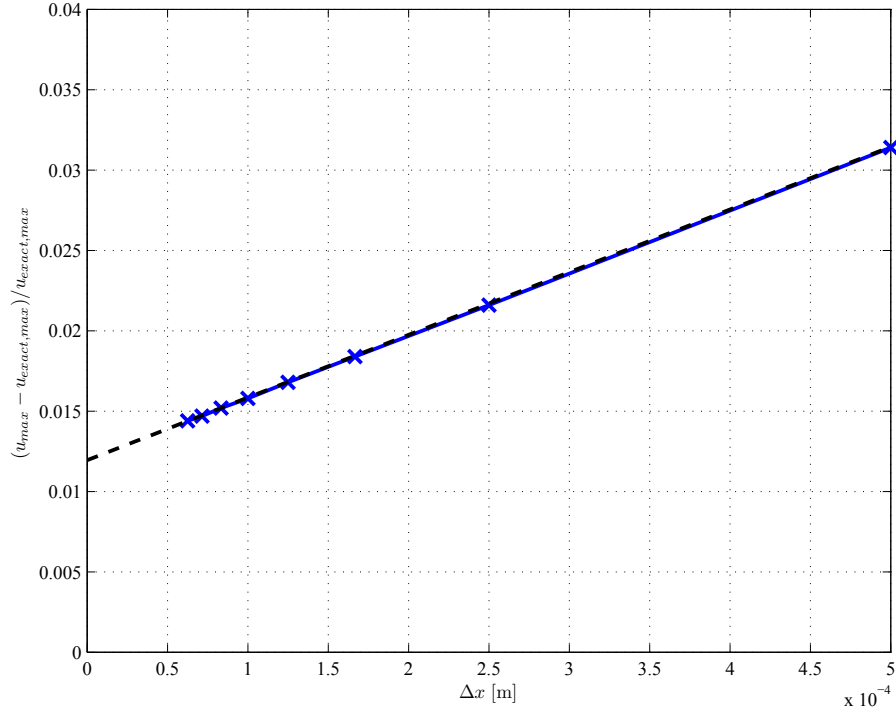


Figure 3.11: Solution for the velocity residual in $u_{\max}$ (Poiseuille Flow) obtained with varying SPH particle numbers across the width of the channel. The error in $u_{\max}$ compared to the exact solution is plotted against the initial particle spacing, Δx . — SPH results, — Trend line for SPH results showing the predicted solution for $\Delta x = 0$.

$$\Phi = \phi_s + Cs^p + H = \phi_{2s} + C(2s)^p + H. \quad (3.48)$$

For the following series of equations, it is assumed that the ratio between particle spacings on consecutive simulations is 2. The order of convergence, p , can then be calculated from

$$p = \frac{\log \left(\frac{\phi_{2s} - \phi_{4s}}{\phi_s - \phi_{2s}} \right)}{\log 2} \quad (3.49)$$

and the discretization error can be approximated by

$$E(s) \approx \frac{\phi_s - \phi_{2s}}{2^p - 1}. \quad (3.50)$$

where ϕ_{2s} and ϕ_{4s} represent solutions with twice and four times the particle spacing of ϕ_s respectively. Equation (3.49) can be used to calculate the order of convergence for each set of three simulations. For both of these sets of simulations the order of convergence

was found to be very close to one, as was indicated by the straight line shown on Figure 3.11.

Now that the order of convergence has been calculated, Richardson extrapolation can be used to estimate the theoretical continuum solution for each set of three simulations. The continuum solution, ϕ_0 , can be calculated from

$$\phi_0 = \phi_s + E(s) = \phi_s + \frac{\phi_s - \phi_{2s}}{2^p - 1} \quad (3.51)$$

for any two simulations where the difference in the particle spacings between them have a ratio of two (American Institute of Aeronautics and Astronautics (1998)). When the order of convergence is one, this equation reduces to

$$\phi_0 = \phi_s + E(s) = 2\phi_s - \phi_{2s}, \quad (3.52)$$

giving a linear relationship between the predicted solutions and the continuum solution. For both sets of simulations considered here, this analysis gives a value of the maximum velocity for the continuum solution of $1.012 \times 10^{-4} \text{ms}^{-1}$, confirming the result derived from the graph shown on Figure 3.11.

The particle spacing conversion index (PSCI), equivalent to the grid convergence index for grid-based methods, can also be checked and used to provide a measure of uncertainty of the particle spacing convergence (American Institute of Aeronautics and Astronautics (1998)). If sufficiently fine particle resolutions are used in all the simulations, such that all the simulations fall within the asymptotic range of the numerical solution, then the PSCI will be small. This was shown to be true graphically in Figures 3.10 and 3.11, but will be checked again by calculating the PSCI for each set of three simulations. The PSCI is defined as

$$PSCI = \frac{F|\epsilon|}{2^p - 1}. \quad (3.53)$$

where F is a safety factor, recommended by Roache (1998) as $F = 1.25$ for comparisons between three grids, and ϵ is the relative error between two consecutive solutions given by

$$\epsilon = \frac{\phi_{2s} - \phi_s}{\phi_s}. \quad (3.54)$$

The PSCI should be checked for all simulations used to calculate the order of convergence to ensure that they are all within the asymptotic range of convergence. The solutions can be said to be within the asymptotic range of convergence if

$$\frac{PSCI_{coarse}}{2^p PSCI_{fine}} \approx 1. \quad (3.55)$$

Data relating to the two sets of simulations are presented in Table 3.2, giving the order of convergence, the calculated continuum solution using Richardson extrapolation, the PSCI for each pair of simulations within the sets and the results of the asymptotic range check as the solution of Equation (3.55). The two sets of simulations give very similar results throughout. The order of convergence, p , for the SPH model during the Poiseuille flow test case is approximately 1.0. The results of the Richardson extrapolation confirm that the continuum solution for the maximum velocity seen on the graphs is indeed $1.012 \times 10^{-4} \text{ms}^{-1}$, giving a percentage error from the theoretical solution of 1.2%. Finally, the PSCI values and asymptotic range check confirm that all the simulations are within the asymptotic range of convergence, implying that the convergence test is valid.

Number of SPH particles	Order of convergence	Calculated continuum solution	$PSCI_{fine}$	$PSCI_{coarse}$	Asymptotic range check
80, 40, 20	1.00	1.012×10^{-4}	0.59%	1.18%	0.995
160, 80, 40	1.02	1.012×10^{-4}	0.29%	0.58%	0.998

Table 3.2: Results of the convergence analysis carried out on two sets of three simulations considering the maximum velocity obtained during steady-state Poiseuille flow.

The calculated order of convergence, p , can be compared to the theoretical order of accuracy of the SPH method. The kernel approximation and its derivative, which are used to derive the SPH equations, are of second order accuracy (Monaghan (2005), Liu & Liu (2003)). This can be observed by performing a Taylor series expansion of Equation (3.1). However, second order accuracy does not hold true close to the boundaries, where the support domain of the SPH particles overlaps the boundaries and the order of accuracy is reduced. There are also a number of errors that can build up

within numerical simulations and propagate throughout the domain. These contribute to a lowering of the accuracy and predominantly involve discretization errors, which have been tested through this convergence study. According to the American Institute of Aeronautics and Astronautics (1998), taking into consideration the propagation of these errors often results in the actual order of accuracy being one degree lower than the theoretical order. For these reasons, the calculated order of convergence found through this test of $p = 1.0$ is reasonable. The calculated order of accuracy is of greater importance than the theoretical order of accuracy, since this value gives the rate at which errors reduce as the particle spacing is refined (Ferziger & Peric (2002)).

It is important to note that the numerically calculated continuum solution is not necessarily expected to be the same as the true physical solution; when the particle spacing is refined, an error may still exist between the two solutions. The consistency of the SPH model for the Poiseuille flow test case has also been checked through the above calculations. If the model is consistent, the numerical solution would tend towards the theoretical solution as the particle spacing decreases. However, a small error of 1.2% was observed between the numerical asymptotic value and the theoretical solution. This is likely to be due to physical errors in the SPH model arising from the approximation of the Poiseuille flow within the SPH simulations. Although the Poiseuille flow theory is well understood, approximations in some of the numerical parameters, such as the boundary representation and viscosity model, will inevitably lead to small errors. Nevertheless, the error observed here is very small and the Poiseuille flow is well represented by the SPH model.

It should also be noted that the numerical solution is affected by the movement of the SPH particles relative to one another during the simulation. Since the Poiseuille flow simulations have been carried out at a very low Reynolds number and operate in a simple domain, the SPH particles move along lines parallel to the channel boundaries and remain evenly spaced. However, it has been observed that when the SPH particles become disordered in more complex flows, the accuracy of the model may be reduced. This is discussed further in Section 3.14 and by Monaghan (2005).

3.5 Turbulence

When modelling flows with high Reynolds numbers, a turbulence model should be included. Most of the standard turbulence models used in grid-based methods have been adapted for use with SPH. One of the most commonly used models, given by Dalrymple & Rogers (2006), is based on the large eddy simulation (LES) method and employs a sub-particle scaling (SPS) technique similar to the sub-grid scaling used in mesh-based methods. The governing equations are averaged over a length scale comparable to the initial particle spacing, Δx , which is sufficient to solve for large-scale eddies, whilst a closure scheme is necessary to resolve eddies smaller than the initial particle spacing. A Favre-averaging scheme for compressible flows, such that for an arbitrary function, f , $\tilde{f} = \overline{\rho f} / \bar{\rho}$, is chosen to average the governing equations, as this does not introduce additional terms into the continuity equation. The governing equations, (Equations (3.20) and (3.19)) can then be written as

$$\frac{d\bar{\rho}}{dt} = -\bar{\rho} \nabla \cdot \tilde{\mathbf{v}} \quad (3.56)$$

and

$$\frac{d\tilde{\mathbf{v}}}{dt} = -\frac{1}{\bar{\rho}} \nabla \bar{p} + \nu \nabla^2 \tilde{\mathbf{v}} + \mathbf{g} - \frac{1}{\bar{\rho}} \nabla \cdot \tilde{\boldsymbol{\tau}} \quad (3.57)$$

respectively, where $\tilde{\boldsymbol{\tau}}$ is the SPS scale stress tensor. Employing a variable-density form of the Smagorinsky model (Blin *et al.* (2003), Moin *et al.* (1991)) and adopting tensor notation gives

$$\tau^{ij} = 2\bar{\rho}\nu_t(\tilde{S}^{ij} - \frac{1}{3}\tilde{S}^{kk}\delta^{ij}) - \frac{2}{3}\bar{\rho}C_I(\Delta x)^2|\tilde{S}|^2\delta^{ij}, \quad (3.58)$$

where ν_t is the eddy viscosity, δ^{ij} is the Kronecker delta, C_I is a model constant set to 0.0066 and Δx is the initial particle spacing, used to determine the spatial filter width. The third term in Equation (3.58) represents the isotropic part of the SPS stress tensor. In incompressible flows, this term can be absorbed into the pressure and ignored, leaving the standard Smagorinsky model. However, since the SPH model described herein is weakly compressible, this third term should be included. The Favre-averaged rate of strain tensor, $\tilde{S}^{ij}$, is given by

$$\tilde{S}^{ij} = \frac{1}{2} \left(\frac{\partial \tilde{v}^i}{\partial x^j} + \frac{\partial \tilde{v}^j}{\partial x^i} \right). \quad (3.59)$$

and the eddy viscosity, ν_t , is given by

$$\nu_t = (C_s \Delta x)^2 |\tilde{S}|, \quad (3.60)$$

where C_s is the Smagorinsky constant, taken to be 0.12, and $|\tilde{S}|$ is the scalar mean rate of strain given by $|\tilde{S}| = \sqrt{2\tilde{S}^{ij}\tilde{S}^{ij}}$. The components of $\tilde{S}^{ij}$ can be calculated in SPH using Equation (3.14). In the case of $\partial u / \partial y$ this yields

$$\frac{\partial u}{\partial y} = - \sum_{b=1}^N u_{ab} \frac{m_b}{\rho_b} \frac{\partial W_{ab}}{\partial y}. \quad (3.61)$$

Once τ has been calculated, it is included in the Favre-averaged momentum equation (Equation (3.57)) using a standard SPH representation.

Violeau & Issa (2007) carried out an investigation into the success of various turbulence models for use with free-surface flows in SPH. These included the $k - \epsilon$ model, explicit algebraic Reynolds stress models (EARSIM) and LES modelling (in three dimensions only). They conclude that all three closure models appear to be attractive, although further validation should be undertaken. In terms of modelling complex free-surfaces, EARSIM models are highlighted for further consideration as they are likely to be more successful for modelling this type of flow than the $k - \epsilon$ model. LES was only carried out with three-dimensional SPH as, strictly speaking, eddies larger than the sub-particle scale cannot be explicitly modelled in two dimensions. For this reason, although LES may be promising, they were unable to draw any conclusions due to the computational time needed to carry out three-dimensional simulations. Although LES may not be strictly appropriate in two-dimensional flows, it undoubtedly provides a useful tool for modelling the sub-particle size eddies. In particular, it is far less complex to program and provides faster run times.

More recently, Monaghan (2011) developed a turbulence model specifically for SPH. The model is based on a Lagrangian and uses a smoothed velocity, which reduces the magnitude of the velocity spectrum at shorter length scales. Simulations on an enclosed box showed good agreement with experiment. However, since the model has not yet

been validated for free-surface flows of the type described in this thesis, this turbulence model was not investigated for use herein.

3.5.1 Turbulence model tests

For the studies carried out in the later chapters, it was necessary to introduce a turbulence model to better capture the violent and fluctuating nature of the flows. The turbulence model chosen for use herein is that proposed by Dalrymple & Rogers (2006) and described above. A full discussion of the reasons for choosing this turbulence model is given in § 5.3.2. In this section, the effects and accuracy of the turbulence model are assessed. This was achieved by first repeating the Poiseuille flow test case (used to test the viscosity models in § 3.4.1) at a higher Reynolds number to ensure turbulent flow and, second, by considering free-surface flow in a continuous uniform channel.

In the first instance, it is demonstrated that the turbulence model does not affect the results for the Poiseuille and Couette flows at low Reynolds number, as is expected. In order to do this, the simulations used to produce the results presented in Figures 3.3 and 3.7 were repeated with the turbulence model switched on. The velocity profiles for Poiseuille and Couette flows are given in Figures 3.12 and 3.13 respectively. In both cases it is clear that, as expected, switching the turbulence model on has not had any effect on the low Reynolds number flow. This is important, as although the turbulence model is included to enhance the model results at high Reynolds numbers, undesirable effects at low Reynolds numbers must be avoided.

The benefits of introducing the turbulence model to higher Reynolds number flows are now demonstrated by repeating the simulation used to produce Figure 3.12 at a Reynolds number of $Re \approx 500,000$ in a plane channel with a width of $y = 1.0\text{m}$. The number of SPH particles representing the water across the channel was kept at 39, giving an initial particle spacing of $\Delta x = \Delta y = 0.025\text{m}$. In order to maintain the flow at this Reynolds number a driving force per unit mass of $F = 0.002\text{ms}^{-2}$ was applied along the channel. All other model parameters were kept the same as in previous simulations, with the *Realistic* viscosity model employed. Instead of initialising the flow from a stationary condition, it was preferable to move the SPH particles at a constant velocity close to the expected maximum velocity from the outset of the simulation. This reduced the amount of time required for the flow to reach a steady-state condition.

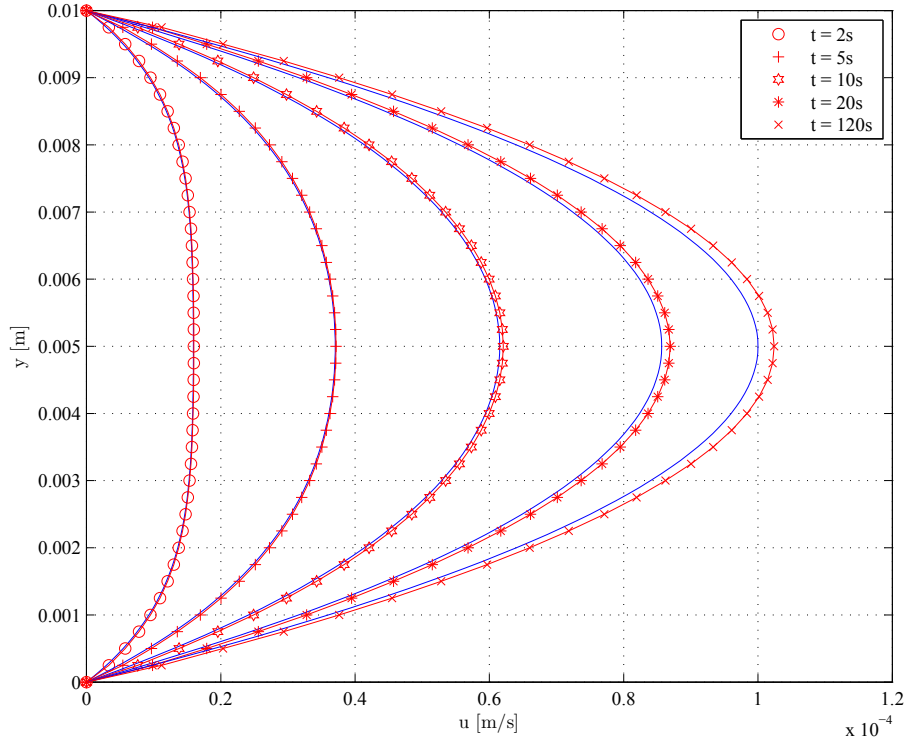


Figure 3.12: Time varying Poiseuille flow tending towards a steady-state, comparisons between results obtained with the SPH model including a turbulence model and theory: — SPH model, — theory. Results are presented at five times, the final time being after the steady-state was established.

Figure 3.14 shows the SPH predicted velocity profile after reaching the steady-state. With an increased Reynolds number, the profile across the channel has a different shape to the theoretical laminar solution. In general terms the SPH predicted profile is in accordance with expectations; the turbulent flow leads to increased mixing and hence reduced velocity gradients, except in the immediate vicinity of the wall. According to turbulent flow theory, the velocity profile should follow a log-law. If this theory holds true, then the velocity plotted against the log of the distance from the wall should form a straight line. The data shown in Figure 3.14 is re-plotted, for half of the channel width, in log form in Figure 3.15 and compared to the theoretical velocity deficit law. Note that the value y_0 used in the x -axis label refers to half the channel with: $y_0 = l/2$. The velocity deficit law considers the reduction in velocity across the channel from the maximum value occurring at the centre. The value of the friction velocity, u_* , used in the theory has been chosen to match that observed in the SPH model results. Since this

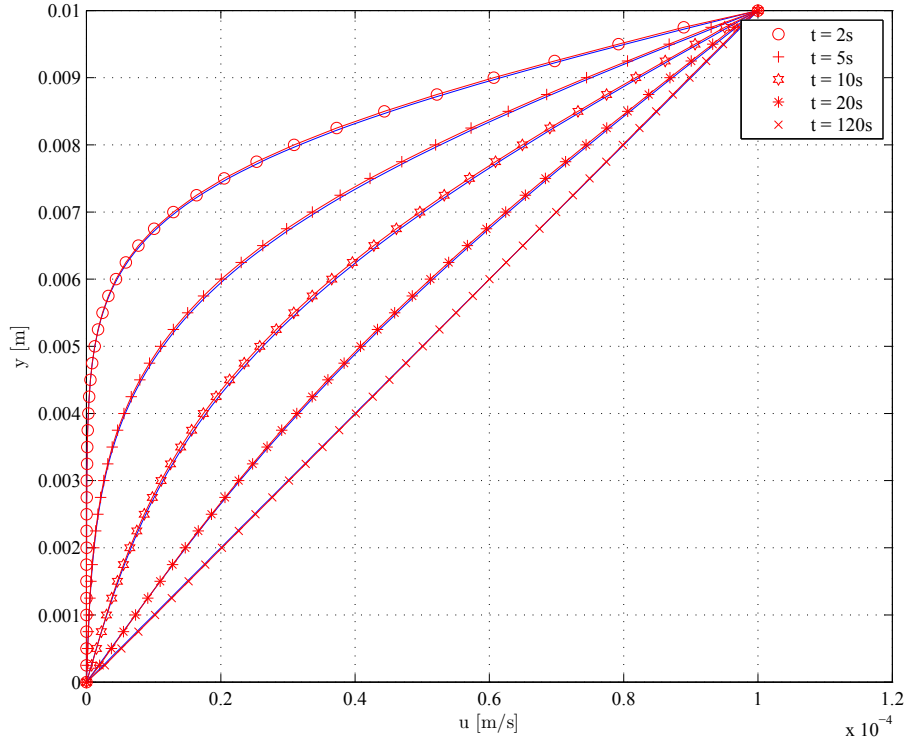


Figure 3.13: Time varying Couette flow tending towards a steady-state, comparisons between results obtained with the SPH model including a turbulence model and theory: — SPH model, — theory. Results are presented at five times, the final time being after the steady-state was established.

plot forms a straight line over the majority of the channel, this implies that the log-law profile is well described and confirms that the turbulence model is working appropriately. In considering this plot it is important to note that the measurement taken one particle spacing away from the wall is omitted from Figure 3.15, since the velocity at this location is heavily influenced by conditions at the wall. This can be seen in Figure 3.14 where the velocity profile close to the walls is slightly distorted. The SPH particle resolution on the boundaries appears to affect the water particles in the layer closest to the wall; it is expected that increasing the particle resolution would improve the appearance of the velocity profile in this region. Details of the turbulent boundary condition used in the SPH model, including the friction velocity, u_* , will be given in § 3.10 and discussed further in § 5.3.3.

Finally, the turbulence model is put to test on a free-surface flow. Flow along a uniform channel was used to demonstrate that the correct vertical velocity profile could

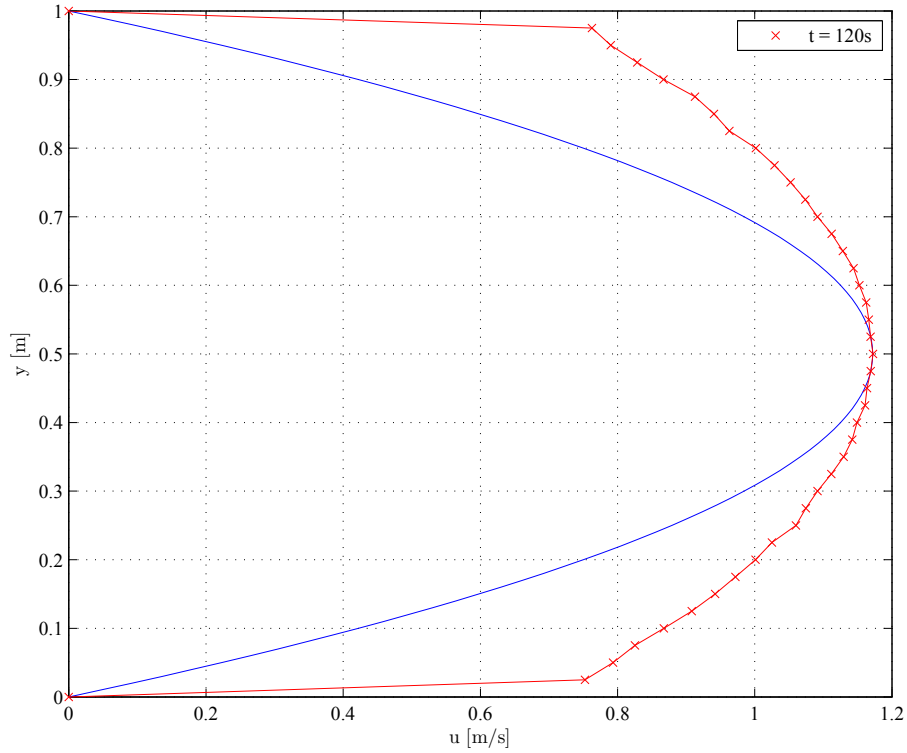


Figure 3.14: Steady-state turbulent plane channel flow at $Re \approx 500,000$; comparisons between the theoretical laminar solution (matched to u_{max}) and the results of the SPH model including a turbulence model: — SPH model, — theory.

be established, since this more closely represents the flows considered in the remainder of this thesis. The water depth was set to $d = 0.4\text{m}$ and the initial particle spacing was $\Delta x = \Delta y = 0.005\text{m}$ with 80 SPH particles placed throughout the depth. A driving horizontal force per unit mass of $F = 0.004\text{ms}^{-2}$ was applied throughout the simulation. This gave a Reynolds number of $Re \approx 220,000$. The flow was initialised with the full water column moving with a uniform velocity of 0.5ms^{-1} . All other SPH parameters were identical to those described for the Poiseuille flow simulations. The vertical velocity profile is presented in Figure 3.16, along with a comparison to the velocity distribution theory for wide channels given by Subramanya (2001). The SPH model is able to reproduce the log-profile of the velocity with reasonable accuracy.

Figures 3.12 to 3.16 demonstrate that the turbulence model selected for modelling the flows described in this thesis performs as expected in the simple benchmark tests described herein. Specifically, the turbulent plane channel flow simulation shows that

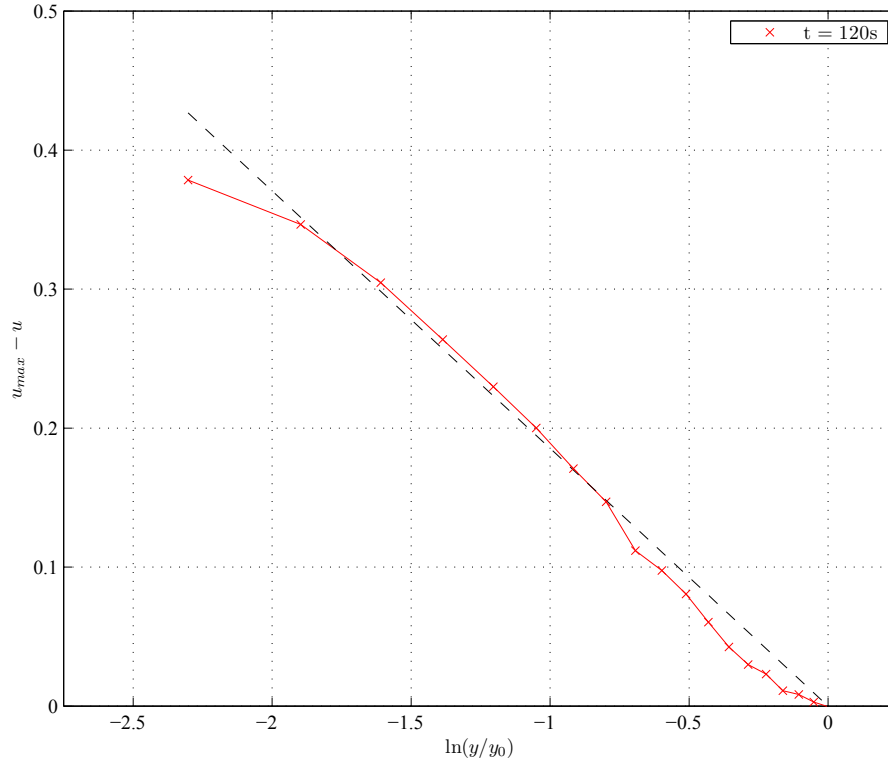


Figure 3.15: Log profile of steady-state turbulent plane channel flow velocity with $Re \approx 500,000$, obtained with the SPH model including a turbulence model: — SPH model, -- velocity deficit theory.

the turbulence model ensures a log-law velocity profile is correctly established across the channel width, while the free-surface channel flow calculations confirm that the correct vertical velocity profile develops in free-surface flows; the latter is in reasonable agreement with establish theory. Further discussion of the turbulence model is given in § 5.3.2.

3.6 XSPH

In order to prevent particle penetration in high speed flows and to keep the particles more orderly, Monaghan (1989) introduced a velocity correction, commonly referred to as XSPH. For each particle, the correction uses the sum of the velocity differences between the particle itself and the surrounding particles within its compact support region, to ‘smooth’ the particle’s velocity. This brings the velocity of individual particles closer to the average velocity of particles in the local area and has proved to be particularly

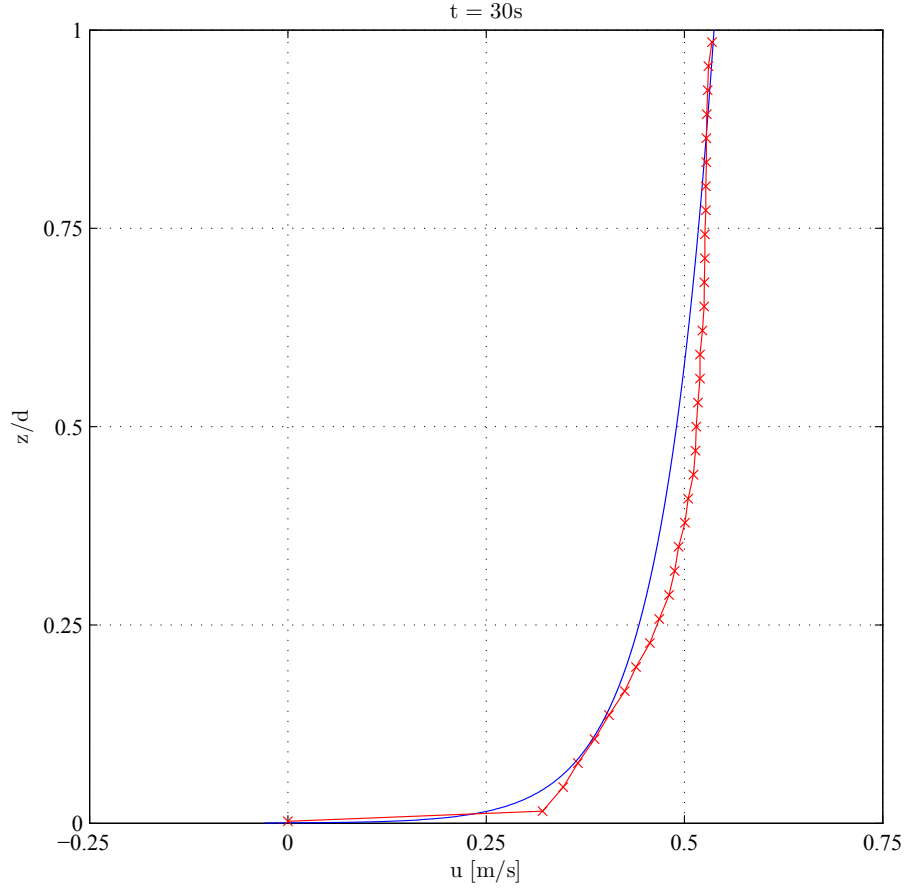


Figure 3.16: Open channel flow velocity profile with $\text{Re} \approx 220,000$, obtained using the SPH model with an effective turbulence model: — SPH model, — theory.

important in free-surface flows. The new velocity is defined by

$$\langle \mathbf{v}_a \rangle = \mathbf{v}_a + \Delta \mathbf{v}_a \quad (3.62)$$

where $\Delta \mathbf{v}_a$ is the velocity correction, given by

$$\Delta \mathbf{v}_a = \epsilon \sum_{b=1}^N \frac{m_b \mathbf{v}_{ab}}{\bar{\rho}_{ab}} W_{ab}. \quad (3.63)$$

The factor, ϵ , determines the strength of the smoothing effect and can be set between 0 and 1. However, it is usually chosen to be 0.5. The corrected velocity is used to update the particle positions within the time marching scheme (see § 3.11) and also in the continuity equation (Equation (3.18)) for consistency. However, it is not used in the momentum equation. Linear and angular momentum are unaffected by XSPH.

3.7 Density smoothing techniques

When using the continuity equation (Equation (3.18)) to calculate the density and the equation of state (Equation (3.27)) to calculate the pressure, small variations in these variables throughout the flow can be magnified throughout the flow field. This can be seen in Figure 3.17a which shows the pressure field under a solitary wave. It can be seen that on the macro scale, the pressure distribution without any density smoothing is hydrostatic, as it should be. However, there are small, but noticeable, variations between neighbouring particles. Using a density smoothing technique removes these variations, improving the quality of the pressure field and helping to maintain a smooth free-surface, as can be seen in Figure 3.17b.

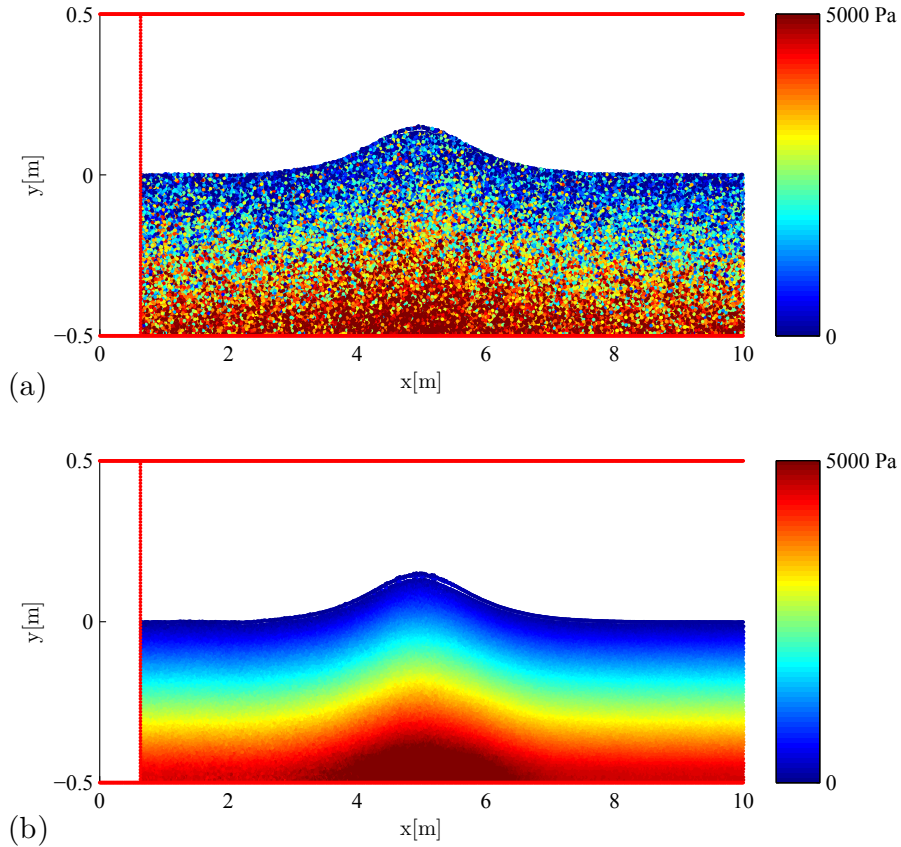


Figure 3.17: Pressure distribution associated with a propagating solitary wave: (a) without density smoothing and (b) with density smoothing.

Colagrossi & Landrini (2003) introduced a density re-initialisation technique to improve the quality of the pressure field in their SPH model. In doing so, they found

that their approach also proved useful for restoring consistency between the particles' mass, density and occupied space. Unfortunately, this is not necessarily satisfied exactly when the density is calculated through the continuity equation. The adopted approach involves calculating the density once every twenty to fifty time steps using Equation (3.10), but with a specially chosen kernel. The kernel is chosen to alleviate the problem which arises in the calculation of the density near the boundaries, and is constructed using a moving-least-squares technique. Details of the adapted procedure are as follows:

$$\langle \rho_a \rangle = \sum_b \rho_b W_b^{MLS}(\mathbf{x}_a) dV_b = \sum_b m_b W_b^{MLS}(\mathbf{x}_a), \quad (3.64)$$

where W_b^{MLS} is defined by

$$W_b^{MLS}(\mathbf{x}_a) = [\beta_0(\mathbf{x}_a) + \beta_1(\mathbf{x}_a)(x_a - x_b) + \beta_2(\mathbf{x}_a)(y_a - y_b)] W_{ab} \quad (3.65)$$

and

$$\boldsymbol{\beta}(\mathbf{x}_a) = \begin{pmatrix} \beta_0 \\ \beta_1 \\ \beta_2 \end{pmatrix} = \mathbf{A}^{-1}(\mathbf{x}_a) \begin{bmatrix} 1 \\ 0 \\ 0 \end{bmatrix}, \quad (3.66)$$

$$\mathbf{A}(\mathbf{x}_a) = \sum_b W_b(\mathbf{x}_a) \tilde{\mathbf{A}}_{ab} \quad (3.67)$$

and

$$\tilde{\mathbf{A}}_{ab} = \begin{bmatrix} 1 & (x_a - x_b) & (y_a - y_b) \\ (x_a - x_b) & (x_a - x_b)^2 & (y_a - y_b)(x_a - x_b) \\ (y_a - y_b) & (y_a - y_b)(x_a - x_b) & (y_a - y_b)^2 \end{bmatrix}. \quad (3.68)$$

As an alternative to the method proposed by Colagrossi & Landrini (2003), Dalrymple & Rogers (2006) suggested the use of a Shepard Filter, applied once every forty time-steps. In this case the density is re-initialised according to

$$\rho_a = \frac{\sum_{b=1}^N \rho_b W_{ab} V_b}{\sum_{b=1}^N W_{ab} V_b}. \quad (3.69)$$

The most significant advantage of this approach is that it is computationally quicker than the density re-initialisation technique of Colagrossi & Landrini (2003).

In considering these approaches, Dalrymple & Rogers (2006) commented that density smoothing techniques still allow the nonlinearities of the physics to be captured as the governing equations are not linearised before being converted to a form appropriate to SPH. Furthermore, since the filtering only takes place locally around each particle, highly-nonlinear processes, such as wave breaking, can still be modelled successfully.

3.8 SPH boundary representations

In SPH the representation of boundaries has proved to be an on-going problem. At present, boundaries are always represented by some form of additional particles which interact with the fluid particles in such a way that penetration is avoided. However, different authors choose to do this by different methods.

In an extensive series of papers, Monaghan has chosen to model boundaries as lines of particles that exert repulsive forces on the fluid particles. If a no-slip condition is required, then the boundary particles are included in the calculation of the artificial viscosity. Originally, Monaghan (1994) used forces based on the Lennard-Jones form, based upon molecular forces. However, Monaghan & Kos (1999) found this to be problematic because the size of the SPH particles means that the boundary produces a corrugated effect with ripples on the scale of the particle spacing. Hence, Monaghan & Kos (1999) proposed a new boundary model in which the repulsive forces are of the form

$$\mathbf{f} = \mathbf{n}\Gamma(y)\chi(x), \quad (3.70)$$

where $\mathbf{n}$ is a local unit normal vector pointing from the boundary into the fluid, and $\Gamma(y)$ and $\chi(x)$ are respectively functions normal and tangential to the boundary. Both x and y are taken to be positive distances. The function, $\Gamma(y)$, is defined such that it decreases as the distance from the boundary increases, reducing to zero within a few particle spacings from the boundary. In contrast, the function, χ , is chosen so that when a fluid particle is located between two boundary particles, the force contributions arising from both boundary particles combine to make the boundary force constant along the entire boundary. According to Monaghan & Kos (1999), $\Gamma(y)$ and $\chi(x)$ can be defined by

$$\Gamma(y) = \begin{cases} A \frac{1}{\sqrt{q}}(1 - q) & q < 1 \\ 0 & \text{otherwise} \end{cases}, \quad (3.71)$$

where $q = y/(2\Delta X_0)$, where ΔX_0 is the initial particle spacing,

$$A = \frac{1}{h}(0.01c^2 + \beta c \mathbf{v}_{ab} \cdot \mathbf{n}_b), \quad (3.72)$$

and $\beta = 1$ if the particles are approaching the boundary and zero otherwise.

More recently, Monaghan *et al.* (2003) proposed a slightly different form for the boundary force. This was based upon the concept of equal and opposite forces between particles. The force per unit mass exerted on fluid particle, a , by boundary particle, b , is given by

$$\mathbf{f}_{ab} = \frac{m_b}{m_a + m_b} \Gamma(y) \chi(x) \mathbf{n}_b. \quad (3.73)$$

Similarly, the force per unit mass exerted on boundary particle, b , by fluid particle, a , is given by

$$\mathbf{f}_{ba} = -\frac{m_a}{m_a + m_b} \Gamma(y) \chi(x) \mathbf{n}_b. \quad (3.74)$$

Within this solution, $\Gamma(y)$ and $\chi(x)$ are defined as follows:

$$\Gamma(y) = \begin{cases} \frac{2}{3}\beta & 0 < q < \frac{2}{3} \\ \beta(2q - \frac{3}{2}q^2) & \frac{2}{3} < q < 1 \\ \frac{1}{2}\beta(2 - q)^2 & 1 < q < 2 \\ 0 & \text{otherwise} \end{cases}, \quad (3.75)$$

where $q = y/h$ and $\beta = 0.02c^2/y$. β provides an estimate of the maximum force needed to stop a particle moving at the estimated maximum speed, including a factor of $1/y$ to ensure a faster particle can be stopped if necessary.

$$\chi(x) = \begin{cases} \left(1 - \frac{x}{\Delta X_0}\right) & 0 < x < \Delta X_0 \\ 0 & \text{otherwise} \end{cases}, \quad (3.76)$$

where ΔX_0 is the particle spacing. This form of $\Gamma(y)$ and $\chi(x)$ opposes the pressure gradient as it uses the gradient of a kernel. However, according to Monaghan *et al.*

(2003), the application of these two different forms of repulsive force makes little difference to the calculated results. In both cases, if there are corners in the boundary then special treatment is necessary in this region; a description of a suitable method is given by Monaghan *et al.* (2003). Furthermore, a disadvantage of using either of these forms of repulsive force is that they are known to cause fluid particles to peel away from the boundary, particularly near the free-surface, where the pressure is low. Evidence of this effect is shown in Figure 3.18.

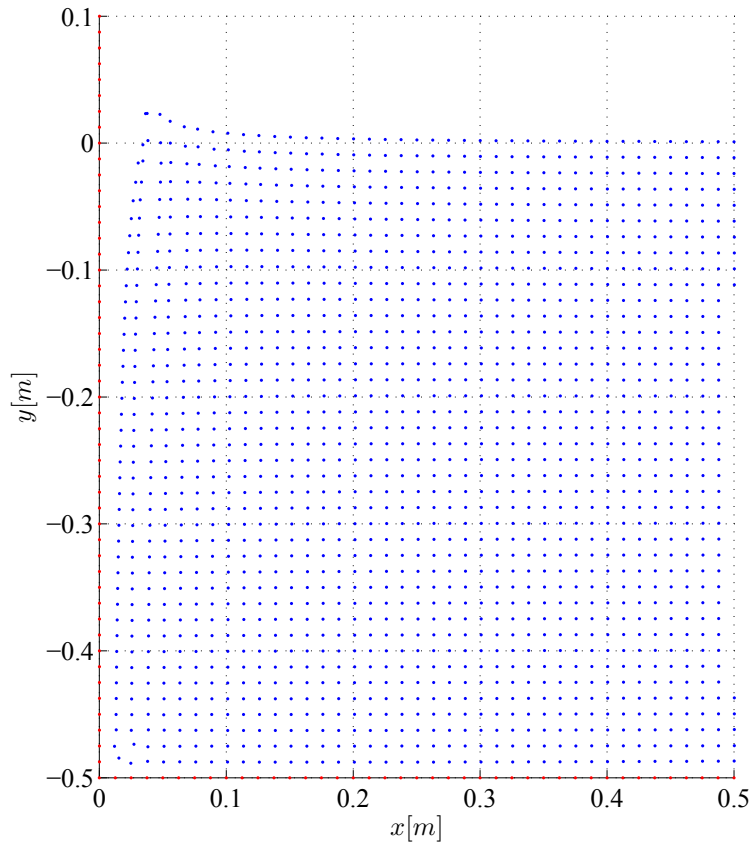


Figure 3.18: Boundaries modelled with a repulsive force condition showing the peeling away of the fluid from the boundary close to the water surface: ● boundary particles, ● fluid particles.

In contrast, Violeau & Issa (2007) chose to represent boundaries with a combination of fixed wall particles and several rows of mirror particles, known as ‘ghost’ particles. In this case, the fixed wall particles do not provide any form of repulsive force and the ghost particles are stationary and placed in several rows behind the wall particles. Although the ghost particles do not have a velocity, in order to create a no-slip condition on the wall, their density, and therefore pressure, is allowed to evolve and is prescribed by

symmetry through pairs formed with the fluid particles. This means that when a ghost particle contributes to the density evolution of a fluid particle, the same term is added to the density evolution of the ghost particle. This method has the advantage of keeping pressure forces symmetric with respect to the wall, which ensures an artificial Neumann condition for pressure. Violeau & Issa (2007) argue that this method is preferable to those methods requiring an artificial repulsive force, as the pressure condition on the wall provides perfect impermeability, even when modelling rapid dynamics phenomena. Furthermore, it is not necessary to introduce any *ad hoc* terms. This method was also found to cause virtually no disturbance to nearby fluid particles at the start of simulations, as opposed to the repulsive force type boundaries with which a significant problem remains.

Other authors use various forms of ghost particles to represent solid boundaries. For example, Morris *et al.* (1997) filled solid objects with static ghost particles and then created a no-slip boundary by defining their velocity to be equal and opposite to the nearest fluid particles. Liu & Liu (2003) described a line of repulsive particles, similar to those used by Monaghan (1994), coupled with ghost particles created by reflecting any fluid particles located within the compact support region of the boundary particles across an imaginary line at the boundary. The ghost particles were given the same density and pressure, but an opposite velocity to create a no-slip condition. Colagrossi & Landrini (2003) used ghost particles to create free-slip boundaries. The position of any fluid particle within the compact support region of a boundary was reflected across it, with the position, velocity and pressure prescribed as follows:

$$\mathbf{x}_{a_G} = 2\mathbf{x}_w - \mathbf{x}_a, \quad (3.77)$$

$$\mathbf{u}_{na_G} = 2\mathbf{U}_{nw} - \mathbf{u}_{na}, \quad (3.78)$$

$$\mathbf{u}_{sa_G} = \mathbf{u}_{sa}, \quad (3.79)$$

$$p_{a_G} = p_a, \quad (3.80)$$

where the subscript a refers to a fluid particle, G refers to the ghost particle being defined, w refers to the boundary itself, and n and s refer to the normal and tangential components to the boundary, respectively. As the tangential velocity is the same for both the ghost particle and the fluid particle (Equation (3.79)), a free-slip boundary

condition is created. Although Colagrossi & Landrini (2003) did not attempt to create no-slip boundaries, this is simply achieved by prescribing the tangential component of the velocity of the ghost particles in a similar way to the normal component of velocity. For plane boundaries, the free-slip condition is satisfied exactly. However, for arbitrarily shaped boundaries, the particles are reflected point-wise across the local tangent plane. Unfortunately, this method can become difficult to use when the boundaries are complex shapes. In particular, corrections may be needed to avoid having too many or too few ghost particles next to curved boundaries. Modelling convex corners also causes problems, as it is unclear how the properties of fluid particles should contribute to the ghost particles.

In order to avoid the problem of using the mirrored ghost particle method described above on complex boundaries, Ferrari *et al.* (2010) introduced the Virtual Boundary Particle (VBP) method. In this formulation, local point-symmetry is used to determine the location of ghost particles. A particular ghost particle is only used to interpolate the physical properties of the particular fluid particle that has been reflected. Boundary particles are positioned along walls at intervals of half the initial fluid particle spacing. When a fluid particle moves within two smoothing lengths of a particular boundary particle, its position is simply reflected beyond the boundary particle's location and a ghost particle is introduced. Since neither the wall normal or tangent are required, complex geometries pose no problem to this method. The VBP method was tested on dam break and impact flows, as well as flows with complex geometries, and comparisons with experimental data or numerical reference solutions were found to be satisfactory. The method was extended to form the Modified Virtual Boundary Particle (MVBP) method by Vacondio *et al.* (2012). In this case, a further ghost particle is introduced, located on the internal side of the wall at twice the distance from the fluid particle as the original ghost particle. Further boundary particles were also introduced for corners of less than 180° . For test cases using an SPH formulation based on the shallow water equations, the MVBP showed improvements over a ghost particle method with wall mirrored ghost particles. Unfortunately it was not possible to test this formulation within the scope of this thesis.

Many other boundary representations exist and have been developed by a range of authors. For example, Gong *et al.* (2009) chose to add a sponge layer of fluid particles

close to solid boundaries, in which the density is altered to absorb sound waves and, consequently, improve the pressure field. They tested the boundary treatment on a wedge entry simulation with the results being compared to theory and experiment. The pressure distribution and force on a wedge were found to be in good agreement with the analytical results and the sponge layer was observed to absorb the sound wave effectively. More recently, Ferrand *et al.* (2011) employed a boundary formulation with a renormalising factor (similar to that used to reinitialise the density field as described in § 3.7), along with wall segments to correct the strain rate and flux conditions for the dissipation of turbulent energy. Mahmood *et al.* (2011) also used this boundary representation and compared it to a standard ghost boundary formulation for a two-dimensional hydraulic jump flow, showing that the newer method better predicted the bed friction.

3.9 A new hybrid boundary formulation

It will be demonstrated later in this thesis that the stationary ghost boundary representation of Violeau & Issa (2007), described above, is preferable to the repulsive boundary representation of Monaghan *et al.* (2003) for general free-surface flow simulations. However, problems with the stationary ghost boundary representation arise when water particles either move over a previously dry boundary or move away from a boundary which will subsequently dry out. A solution to this problem, combining the two boundary modelling techniques to form a new hybrid boundary representation, is presented herein. In the following sections, the stationary ghost boundary representation will be referred to as the *Stationary Ghosts* method and the repulsive boundary formulation as the *Repulsive Force* method.

The problem encountered with the *Stationary Ghosts* method when water particles move over a previously dry boundary is related to the boundary and ghost particles changing their density and pressure as soon as they form a pair with a water particle. Using the quartic kernel and a smoothing length of $1.3\Delta x$ means that particles initially form pairs when they are $3.25\Delta x$ away from each other. As a water particle approaches a dry boundary, the repellent boundary force felt by the water particle becomes too strong, well before it actually passes the respective boundary particle. This has the

effect of holding the water particle too far away from the boundary. Tests have shown that once this distance is established it becomes impossible to penetrate. This effect can be seen in some plots shown in Crespo *et al.* (2008), who used a boundary model similar to the *Stationary Ghosts* method, although the problem is not discussed in the text. The problem was later recognised by Gomez-Gesteira *et al.* (2010), but a solution was still not provided. Figure 3.19 shows this effect as water from a dam break problem flows across a previously dry bed, the original dam front being located at $x = 0.25\text{m}$.

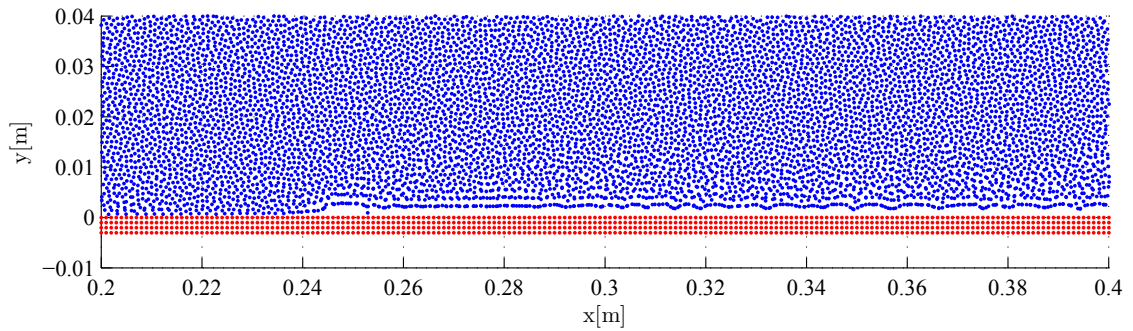


Figure 3.19: SPH particle positions arising in the initial stages of a dam break problem. When water flows across a previously dry boundary the *Stationary Ghosts* boundary method leads to a gap developing between the bottom boundary and the water flow above. • SPH water particles and • ghost particles.

The *Stationary Ghosts* boundary model also causes problems as boundaries dry out. Again, this was recognised by Gomez-Gesteira *et al.* (2010), but a solution was not provided. This effect is obvious when looking at the vertical wall behind a simple setup of a collapsing dam, as shown in Figure 3.20 for a dam originally 0.25m high. The water particles are unable to detach from the wall and appear to be stuck in their initial locations. This can also be seen in some plots shown in Violeau & Issa (2007), although, again, it is not discussed in the text.

Given the problems, it was decided that for simulations involving wetting and drying boundaries a combination of the two boundary modelling techniques would be used: *Stationary Ghosts* will be used on fully wet boundaries and the *Repulsive Force* technique will be used on dry and partially dry boundaries. This meant introducing criteria to describe a boundary particle as being either wet or dry. In order to avoid the problem discussed above, with previously dry boundaries becoming wet, it was not enough to

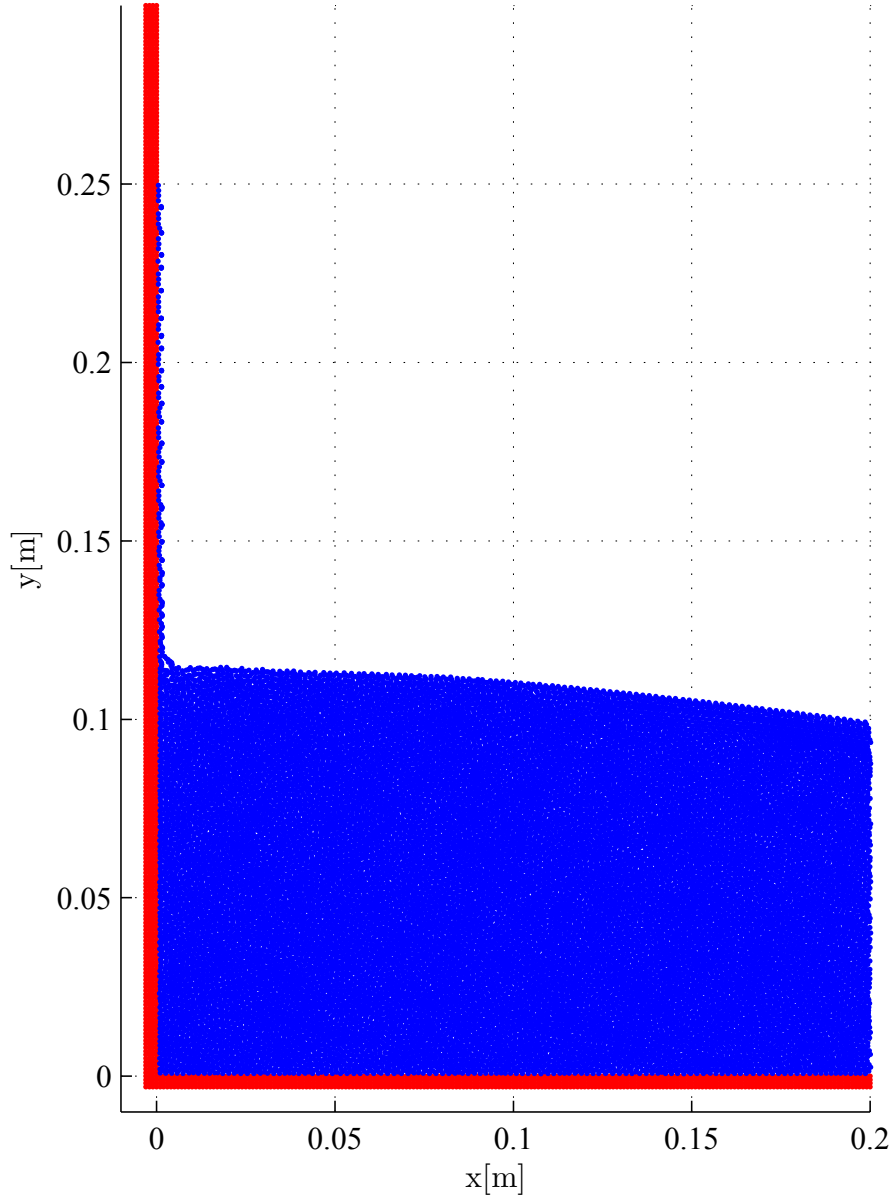


Figure 3.20: SPH particle positions arising some time after a dam break problem. When the water level on the rear containing wall reduces, and the boundary effectively dries out, use of the *Stationary Ghosts* boundary method leaves water particles stuck to the left wall. • SPH water particles and • ghost particles.

consider a boundary particle wet as soon as it forms a pair with a water particle, as the same problem would still occur. Following a number of preliminary tests, it was found that the best criteria was to assign boundaries particles a number of pairs that they must form with water particles before they are considered wet. This number depends

on the kernel and smoothing length being used, and also on the boundary particle's location relative to any corners. An explanation of this latter effect is given as follows. In the middle of a flat plate, water particle pairs would be expected to form in the region 180° around the boundary particle. In contrast, in a right-angled corner, the boundary particle would only be expected to form pairs with water particles in a 90° range. It was found that the *Repulsive Force* technique from Monaghan & Kos (1999), adapted to only begin repelling water particles closer than Δx , worked better for this purpose than the method outlined by Monaghan *et al.* (2003); both variations of the technique are fully described in § 3.8.

In terms of water particles moving onto a previously dry boundary, the proposed method works by using the *Repulsive Force* method for a given dry boundary particle until that boundary particle forms its required number of pairs with the water particles. It then switches to behaving as it would with the *Stationary Ghosts* method. When behaving according to the *Repulsive Force* method, the density and pressure of the boundary particle are kept at the reference density with zero pressure and the ghost particles are ignored. When the boundary particle switches to the *Stationary Ghosts* method, once the required number of water particles are surrounding it and are an appropriate distance away from it, the boundary particle and its associated ghost particles are allowed to adapt their density and pressure as they would do normally.

When water particles move away from a boundary, beginning the process of drying out, the boundary particles stop behaving as they would with the *Stationary Ghosts* method when the boundary particle's number of pairs has fallen below its given allocation. At this point, the boundary particle's density is set back to the reference density, the pressure to zero and the ghost particles are ignored.

This method, which will subsequently be referred to as the *Hybrid* method, allows a smooth transition between the *Stationary Ghosts* and *Repulsive Force* methods of modelling boundaries. The superior properties of the *Stationary Ghosts* method are retained in the majority of the flow, but the disadvantages of the method are avoided at those locations where the boundary either dries out or becomes wet after starting dry.

3.9.1 The collapsing dam test case

In order to further quantify the problems associated with the *Stationary Ghosts* method during wetting and drying, and demonstrate the capability of the new *Hybrid* method, tests have been carried out on a simple collapsing dam experiment. This considers both the decrease in the height of the remaining dam as the adjacent wall dries out, and the progression of the surge front moving over a previously dry bed. The SPH model results are quantitatively compared with the experiments of Martin & Moyce (1952). Images showing the qualitative nature of the boundary representations are also presented.

In the numerical tests, the dam is initially represented by a two-dimensional block of water, 0.25m high and 0.25m wide. The block of water comprised of 62500 SPH particles placed on a Cartesian grid with an initial particle spacing of $\Delta x = \Delta y = 0.001\text{m}$. The dam water was allowed to collapse under gravity, with the surge front moving forwards along a previously dry bed. The domain was 2.5m long and the simulations were allowed to run until the surge front reached the end wall in each case. The numerical setup is shown in Figure 3.21a and two images taken part way through the collapsing process are shown in Figures 3.21b and 3.21c. Aside from varying the boundary representations, all other numerical parameters used for these SPH simulations were kept constant with key elements being the viscosity formulation of Morris *et al.* (1997), a turbulence model based on that described by Dalrymple & Rogers (2006), a quartic kernel (Violeau & Issa (2007)), a Shepard filter for smoothing the density field (Dalrymple & Rogers (2006)) and a time marching scheme described by Monaghan (2007). This combination of parameters is identified in Chapter 4 as being optimal for the description of free-surface flows, with the addition of the turbulence model introduced in Chapter 5.

Initially, three numerical simulations were carried out. The first used the *Stationary Ghosts* method on its own, the second employed the *Repulsive Force* method, and finally, the *Hybrid* method was used for a third simulation. In order to quantify the results and allow comparison to experiments, the time, T , the remaining height of the dam, H , and the surge front, Z , were normalised using the initial dam height, as was done by Martin & Moyce (1952). These parameters are given by

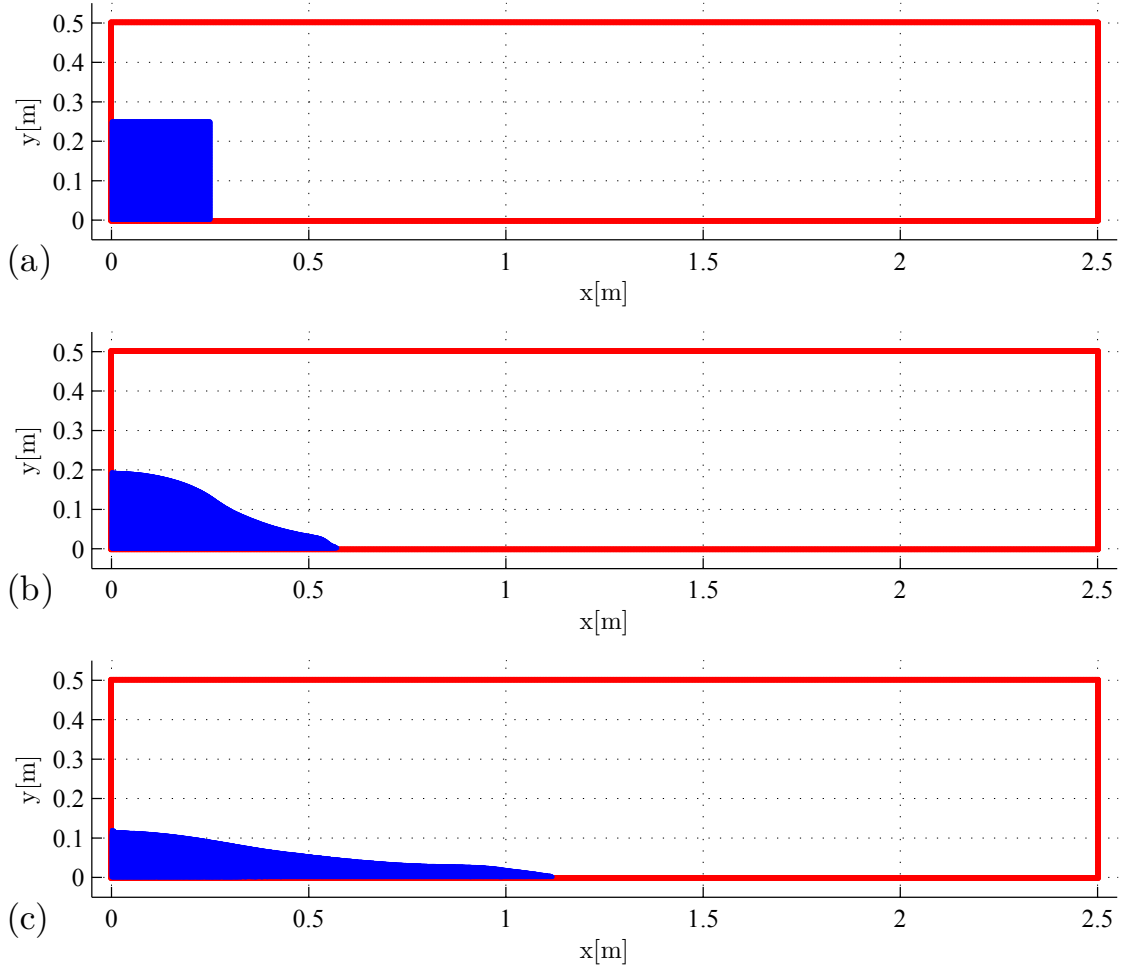


Figure 3.21: Numerical setup and progression of a collapsing dam. (a) Initial dam layout, (b) 0.2s after the dam begins to collapse and (c) 0.4s after the dam begins to collapse. • SPH water particles and • boundary particles.

$$T = t\sqrt{(g/H_0)} \quad (3.81)$$

$$H = h/H_0 \quad (3.82)$$

$$Z = z/H_0 \quad (3.83)$$

where t defines the time, g the acceleration due to gravity, H_0 the original height of the dam, h the remaining height of the dam and z the position of the surge front.

The laboratory experiments of Martin & Moyce (1952) considered the two-dimensional collapse of a block of water 2.25×2.25 inches in the horizontal and 5 inches high. The initial dam water was held in place by a very thin waxed paper diaphragm, sealed by a

steel strip. The dam was released by passing a strong current through the steel strip, freeing the waxed paper and allowing the collapse to commence. Due to this release method, they were unable to precisely record the time at which motion began, the suggested error being around $T = 0.1$. Although this implies that there is considerable uncertainty in the precise timings of the collapse, once motion began, measurement of the rate of collapse was recorded accurately relative to itself. Martin & Moyce (1952) observed that the rate of progression of the surge front, Z , quickly accelerates up to a maximum velocity, then progresses uniformly for some time before eventually tapering off. Their observations indicated that the results were scalable as long as the experiments were not heavily influenced by viscosity or surface tension.

Figure 3.22 shows the reduction in the normalised dam height, H , for each of the numerical simulations, employing the *Stationary Ghosts*, *Repulsive Force* and the *Hybrid* representations respectively, along with the experimental observations reported by Martin & Moyce (1952). It can clearly be seen that there is very little difference in H between any of the boundary representations and they all closely match the experimental results. However, qualitatively, the problem with drying boundaries when using the *Stationary Ghosts* method is present, as was shown in Figure 3.20, and the well-reported problem of water particles peeling away from the boundary near the free-surface is present with the *Repulsive Force* method, as was shown in Figure 3.18. Although neither of these problems affect the height of the dam a small distance away from the wall, they both create visually obvious errors that should be avoided.

Using the *Hybrid* method avoids both of these problems. Since the majority of the adjacent boundary particles are initially prescribed as being wet and the few near the top that are prescribed with the *Repulsive Force* method use a force acting over a reduced distance compared to the usual method, the water particles do not peel away from the wall. As the dam height starts to reduce, using the *Repulsive Force* method for the boundary particles adjacent to the top few water particles ensures that no water particles are left attached to the wall. An image similar to that presented in Figure 3.20, but taken from the simulation using the *Hybrid* method, is presented in Figure 3.23. It is clear in this image that the problems found with using either of the methods on their own have been eliminated. Since the graph in Figure 3.22 shows that the rate at which the dam collapses hasn't been affected by the *Hybrid* method and is still very

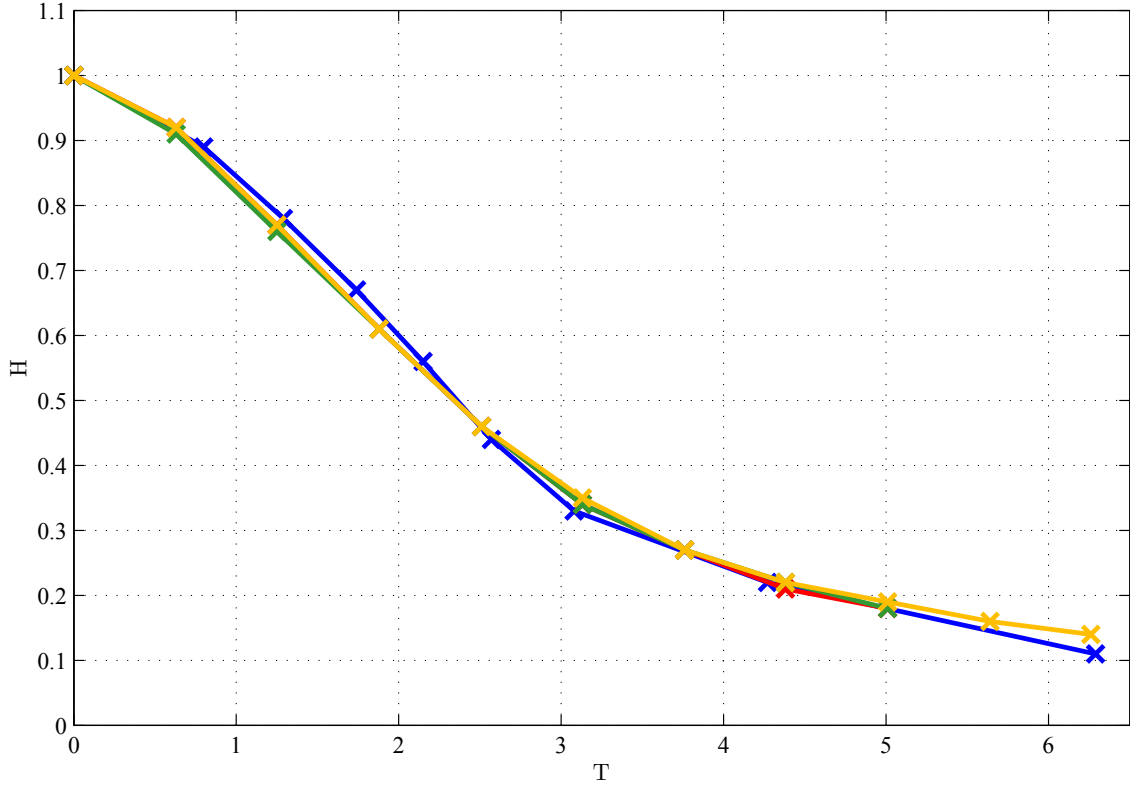


Figure 3.22: Progression of the reduction in height, H , of the remaining dam with time, normalised to the initial dam height. Time series show results of the experiments of Martin & Moyce (1952) and SPH predictions using various boundary representations: — experimental observations, — *Repulsive Force* boundary method, — *Stationary Ghosts* boundary method and — *Hybrid* boundary method.

close to the experimental rate of collapse, it can be said that the *Hybrid* method is highly successful for use on drying boundaries.

A graph of the progression of the surge front, Z , is presented in Figure 3.24. Differences in the rate of progression can be observed between both the varying boundary representations and the experimental results. The rate of movement across the dry boundary is more complex than the drying process since frictional forces at the boundary play a role in how fast the water can move across it. This is particularly true in the latter stages when the surge front thins out. Although no specific frictional force has been applied in this SPH model, all of the boundary representations used in these test cases prescribe no-slip boundary conditions. Therefore, it is expected that they should be able to model the progression of Z correctly. Figure 3.24 shows that both the

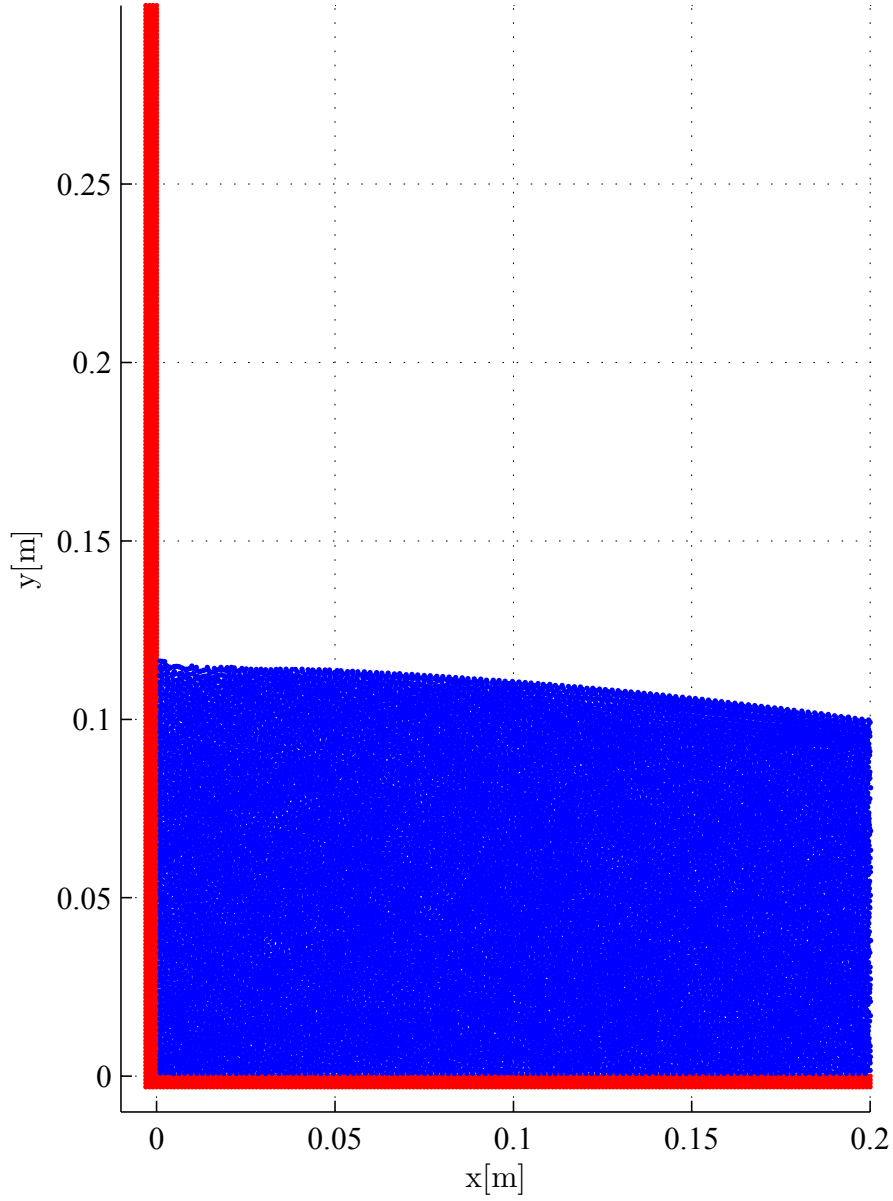


Figure 3.23: SPH particle positions arising some time after a dam break problem. When the water level on the rear containing wall reduces, and the boundary effectively dries out, the *Hybrid* method provides a better qualitative representation of the water adjacent to the wall than the *Stationary Ghosts* boundary method. • SPH water particles and • ghost particles.

Stationary Ghosts method and the *Repulsive Force* method have caused Z to progress faster than in the experiment. Since there are uncertainties in the start time of the experiment, it is not possible to calculate precise errors in the numerical results. However, this does show that the *Stationary Ghosts* method appears to be less responsive to

the no-slip condition than the *Repulsive Force* method. Indeed, Figure 3.19 showed the problem associated with water moving over a previously dry boundary in the *Stationary Ghosts* method. Since the water is held so far away from the boundary, it is perhaps not surprising that the water does not appear to feel the full effect of the imposed no-slip condition. Figure 3.25 shows a similar image when using the *Repulsive Force* method, where it can be seen that the water particles remain close to the boundary as they progress.

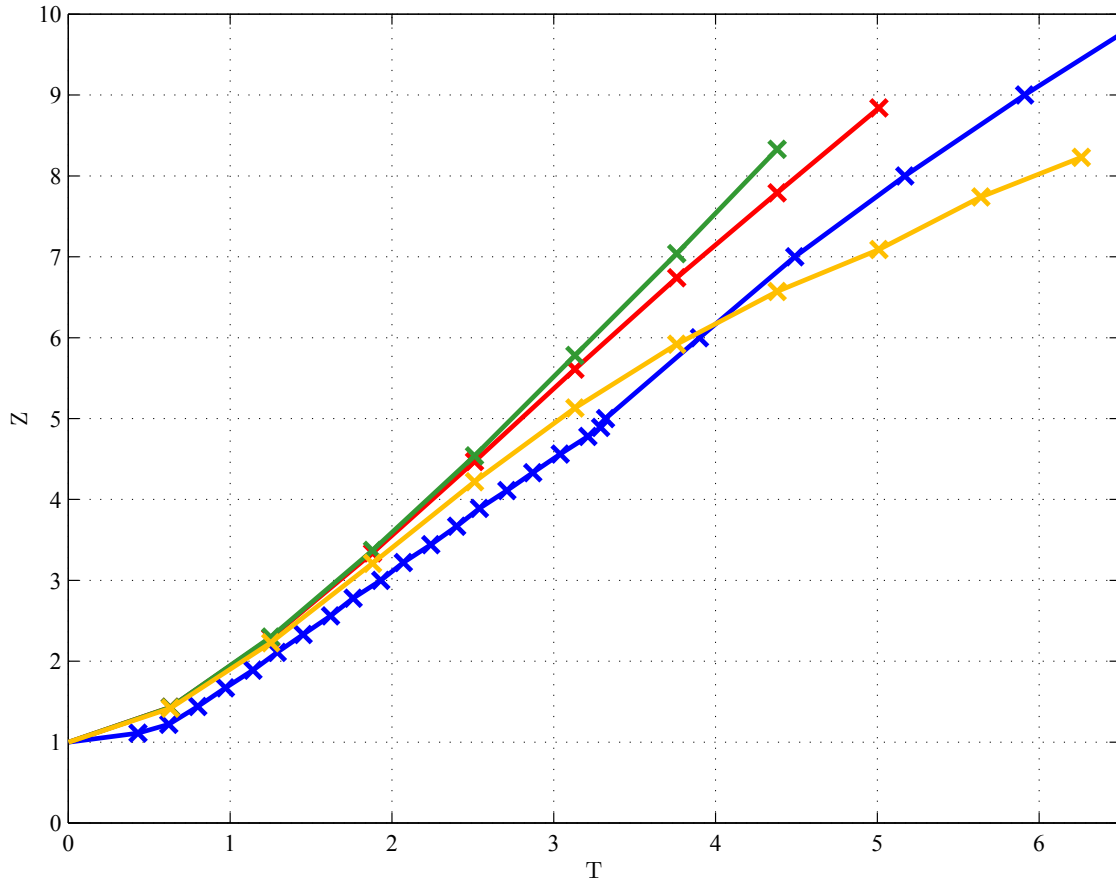


Figure 3.24: Progression of the movement of the surge front, Z , over the dry bed with time, normalised to the initial dam width. Time series show results of the experiments of Martin & Moyce (1952) and SPH predictions using various boundary representations: — experimental observations, — *Repulsive Force* boundary method, — *Stationary Ghosts* boundary method and — *Hybrid* boundary method.

The progression of Z arising from the application of the *Hybrid* method is slower than that predicted by either the *Repulsive Force* or the *Stationary Ghosts* boundary

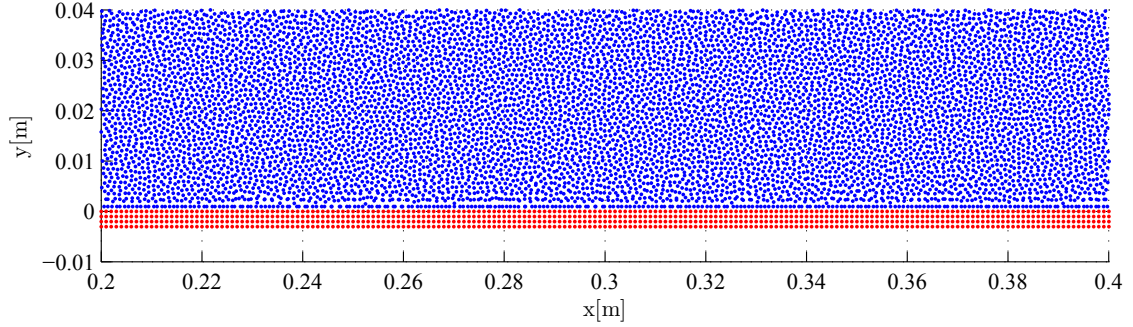


Figure 3.25: SPH particle positions arising in the initial stages of a dam break problem. When water flows across a previously dry boundary the *Repulsive Force* boundary method provides a better qualitative representation of the water immediately above the bed than the *Stationary Ghosts* boundary method. • SPH water particles and • ghost particles.

conditions and consequently lies closer to the experimental data of Martin & Moyce (1952). Again, due to the uncertainty in the experiments, it is impossible to state the absolute error. However, it is considerably slower than even the *Repulsive Force* method; at $T = 4.4\text{s}$, Z is at approximately 84% of that recorded when using the *Repulsive Force* method alone. A qualitative examination of the progression of the surge front across the bed shows that in the initial stages of the dam collapse, the water particles move nicely across the bed, remaining far closer to the boundary than with the *Stationary Ghosts* method alone. This is clearly shown in Figure 3.26. However, as the depth of the water in the surge front decreases, some disruption to the particle positions arises from the *Hybrid* boundary representation. This is believed to be the cause of the slowing observed on the graph of the progression of Z in Figure 3.24. These results suggest that the *Hybrid* approach is successful for modelling the wetting of previously dry boundaries when the depth of the water is relatively high, but may not be as good as the *Repulsive Force* method alone in cases where only a thin strip of water covers the previously dry boundary. As this problem only affects SPH water particles within the boundary particles' influence domain, the problem diminishes with increasing resolution. Nevertheless, it cannot be avoided if the depth of the strip of water thins out to only a few water particles deep. However, in this case the accuracy of any SPH calculation is likely to be very poor.

Further discussion on the merits of the various boundary representations will be

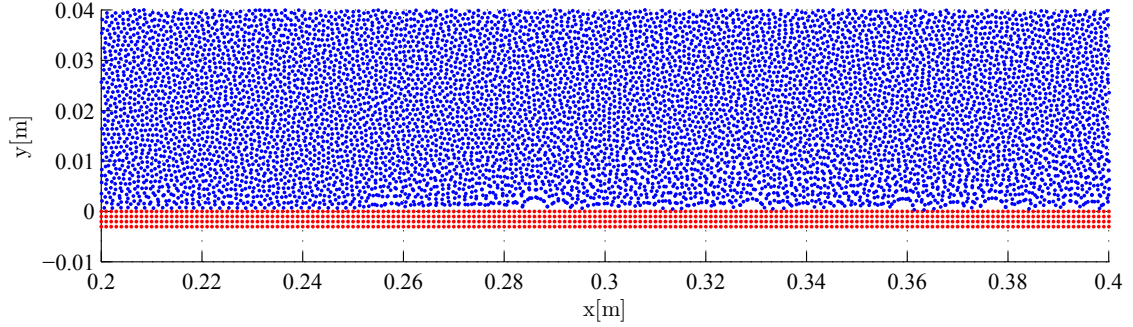


Figure 3.26: SPH particle positions arising in the initial stages of a dam break problem. When water flows across a previously dry boundary, initially the *Hybrid* boundary method provides a better qualitative representation of the water immediately above the bed than the *Stationary Ghosts* boundary method. • SPH water particles and • ghost particles.

given in relation to the specific flows discussed in this thesis in Chapters 4 and 5.

3.10 The turbulent boundary condition

When modelling turbulent flow, the methods described in § 3.5 are insufficient when it comes to the description of turbulence arising in the vicinity of solid boundaries. This is due to the presence of a very thin viscous sub-layer, or wall roughness, having a profound effect on the nature of the flow. Unfortunately, typical particle resolutions do not allow the viscous sublayer to be modelled explicitly in this region. As a result, a turbulent wall condition must be prescribed. According to Violeau & Issa (2007), this can be achieved in SPH by using similar approaches to grid-based methods in which the boundary condition is based on the “law of the wall”. In order to achieve this, the wall particles are now assumed to be located a short distance, δ , away from the physical (or actual) wall. At this location, the law of the wall is applied. For a hydraulically smooth wall, the wall velocity, $\bar{u}_a$, is given by

$$\bar{u}_a = u_a^* \left(\frac{1}{\kappa} \ln \frac{\delta u_a^*}{\nu_a} + 5.2 \right), \quad (3.84)$$

where u_a^* is the local wall shear velocity for particle a , $\kappa = 0.41$ is the von Karman constant and ν_a is the effective viscosity for particle a (set to $\nu = 10^{-6}$ for all water particles). For a rough wall, the wall velocity is given by

$$\bar{u}_a = u_a^* \left(\frac{1}{\kappa} \ln \frac{\delta}{k_s} + 8.5 \right), \quad (3.85)$$

where k_s is the roughness length which depends upon the wall material being modelled. Typically, δ is chosen to be 1/10th of the initial particle spacing, such that the water particles are situated outside of the viscous sub-layer, or roughness length, and the characteristics of the log-layer can be applied. It should be noted that the wall particles are not actually moved from their original locations and that for all other calculations the wall particles behave in the usual way. Furthermore, despite being prescribed a velocity, the wall particles remain stationary. This allows their velocities to be used in calculations to generate the turbulent boundary condition, but prevents structural change to the boundaries.

The local wall shear velocity still needs to be calculated. This can be done by differentiating Equation (3.84) or (3.85) with respect to the direction perpendicular to the wall (remembering that $\delta = \delta(n)$) to give an equation for the velocity gradient of the form

$$\frac{d\bar{u}}{dn} = \frac{u^*}{\kappa\delta}. \quad (3.86)$$

This can be rearranged to give an estimate of u^* . The velocity gradient on the wall can be calculated in SPH form using

$$\left(\frac{d\bar{u}}{dn} \right)_a = \nabla \bar{u}_a \cdot \mathbf{n}_a, \quad (3.87)$$

where $\mathbf{n}_a$ is the boundary normal vector and the components of $\nabla \bar{u}_a$ can be calculated using Equation (3.61).

3.11 Time marching

In SPH, the equations of motion can be integrated using any of the usual time-stepping schemes, provided they remain stable. The size of the time step is limited by a CFL condition. The most general form of this is given by Monaghan & Kos (1999) as

$$\delta t = \beta \min_a \left(\frac{h}{c_a + \sigma_a} \right), \quad (3.88)$$

where the minimum is found by considering all particles (a), β is a constant dependent upon the time stepping scheme, h is the smoothing length and c_a is the speed of sound for particle a . Within Equation (3.88), σ_a is defined by

$$\sigma_a = \max_b \left| \frac{h \mathbf{v}_{ab} \cdot \mathbf{r}_{ab}}{r_{ab}^2} \right|, \quad (3.89)$$

where the maximum is found by considering all particles (b) that contribute to the viscosity of particle a . For an improved Euler predictor-corrector method, Monaghan & Kos (1999) give $\beta = 0.3$, whereas Colagrossi & Landrini (2003) give $\beta = 2.5$ for a fourth-order Runge-Kutta method. The CFL condition (Equation (3.88)) makes the time steps significantly smaller than is necessary with most finite difference methods. However, the compensation for this is that, unlike finite difference methods, there is no need for iterations on each time step as SPH is a fully Lagrangian method.

In addition to the CFL condition given by Equation (3.88), a second limiting condition is often used to calculate the SPH time step size (Monaghan & Kos (1999)). This is given by

$$\delta t = \beta \min_a \left(\sqrt{\frac{h}{\mathbf{F}_a}} \right), \quad (3.90)$$

where h is the smoothing length and $\mathbf{F}_a$ is the force per unit mass of particle a . This ensures that the time step is kept small when large particle accelerations occur within the flow. When both limiting conditions are used, the time step size is determined by the minimum δt calculated from Equations (3.88) and (3.90).

Monaghan & Kos (1999) state that the advantages of using higher order time-marching schemes in SPH (higher than second-order) are more than discounted by the stability properties of the partial differential equations. In contrast, Colagrossi & Landrini (2003) found some improvement in stability with a fourth-order Runge-Kutta scheme, also having the advantage of allowing much larger time steps, thus reducing the computational time. In a follow-up paper, Monaghan (2005) argues that despite the Runge-Kutta scheme being of higher order, it does not conserve angular momentum and is not reversible. Indeed, he argues that for SPH schemes, good conservation is more important than higher order integrators.

Monaghan (2007) proposed an alternative second order scheme in which the density

and accelerations are updated at different stages within the time step. This scheme was shown to be more stable than the standard Verlet scheme and would be reversible in time if the viscosity term was ignored. Details of this time stepping procedure are given below for the case where the accelerations depend on the velocity due to the viscosity term in the momentum equation and a variable time step is used. The subscript, a , implies that the time stepping scheme is applied to each particle, while the superscripts refer to the stage within the time step: 0 refers to the beginning of the time step, 1/2 is halfway through and 1 is the end, which becomes the beginning of the next time step. Taking each step in turn, and adopting the notation that $\mathbf{F}_a$ and D_a are respectively the rate of change of velocity and density, the time marching progresses as follows:

- Calculate $\mathbf{F}_a^0 = \frac{d\mathbf{v}_a}{dt}$
- Calculate δt^0 according to Equations (3.88) to (3.90) (depends on $\mathbf{r}^0$, $\mathbf{v}^0$ and ρ^0)
- Update $\mathbf{r}$, $\mathbf{v}$ and ρ to the mid-step according to

$$\mathbf{r}_a^{1/2} = \mathbf{r}_a^0 + \frac{1}{2}\delta t^0 \mathbf{v}_a^0 \quad (3.91)$$

$$\mathbf{v}_a^{1/2} = \mathbf{v}_a^0 + \frac{1}{2}\delta t^0 \mathbf{F}_a^0 \quad (3.92)$$

$$\rho_a^{1/2} = \rho_a^0 + \frac{1}{2}\delta t^0 D_a^0 \quad (3.93)$$

- Calculate $\mathbf{F}_a^{1/2} = \frac{d\mathbf{v}_a}{dt}$
- Calculate $\delta t^{1/2}$ (depends on $\mathbf{r}^{1/2}$, $\mathbf{v}^{1/2}$ and $\rho^{1/2}$)
- Set $\delta t^1 = 2\delta t^{1/2} - \delta t^0$
- Update $\mathbf{v}$ and $\mathbf{r}$ to the end of the time step according to

$$\mathbf{v}_a^1 = \mathbf{v}_a^{1/2} + \frac{1}{2}(\delta t^0 + \delta t^1)\mathbf{F}_a^{1/2} \quad (3.94)$$

$$\mathbf{r}_a^1 = \mathbf{r}_a^{1/2} + \frac{1}{2}\delta t^1 \mathbf{v}_a^{1/2} \quad (3.95)$$

- Calculate $D_a^1 = \frac{d\rho_a}{dt}$

- Update ρ to the end of the time step according to

$$\rho_a^1 = \rho_a^{1/2} + \frac{1}{2}\delta t^1 D_a^1. \quad (3.96)$$

3.12 Computational considerations

Whenever the summation technique is used in a calculation, the particles taking part in that calculation must have already been identified. For a calculation on particle a , the particles taking part in the calculation are all those lying within the influence domain of particle a . It is, therefore, useful to store a list of particles that form pairs with one another at the beginning of each time step. Particle pairs are found by searching the domain, including both the fluid and boundaries, to find particles lying within each other's influence domains. If the whole domain is searched for each particle, then the time taken to perform this search is proportional to the square of the total number of particles. The search time can be reduced by using link-lists of particle pairs. At each time step, a background grid is used with cells of size κh , where κ is the number of smoothing lengths used by the particular kernel to define the region of compact support. In order to find particle pairs for a particle a , it is only necessary to search within the grid space containing a , and the eight surrounding grid spaces. It is known that any particle lying further away than this would not be close enough to form a pair with a . The grid is shown in Figure (3.27). It must be stressed that the background grid is only used in the pair searching process. Alternative particle pair searching methods have been developed, such as the Inner and Outer Particle Searching method by Zheng *et al.* (2010), which may be more efficient. However, the SPH model developed herein used the link-list method throughout.

As most calculations use the summation technique, the computational time is also affected by the number of particle pairs. The number of particle pairs depends on the smoothing length, the kernel and the dimension of the model. Both the smoothing length and the kernel define the size of the influence domain. Increasing the smoothing length, or using a kernel with a compact support region defined by a higher number of smoothing lengths, increases the number of particle pairs. In two dimensions, the number of particle pairs per particle has a square relationship. However, in three dimensions,

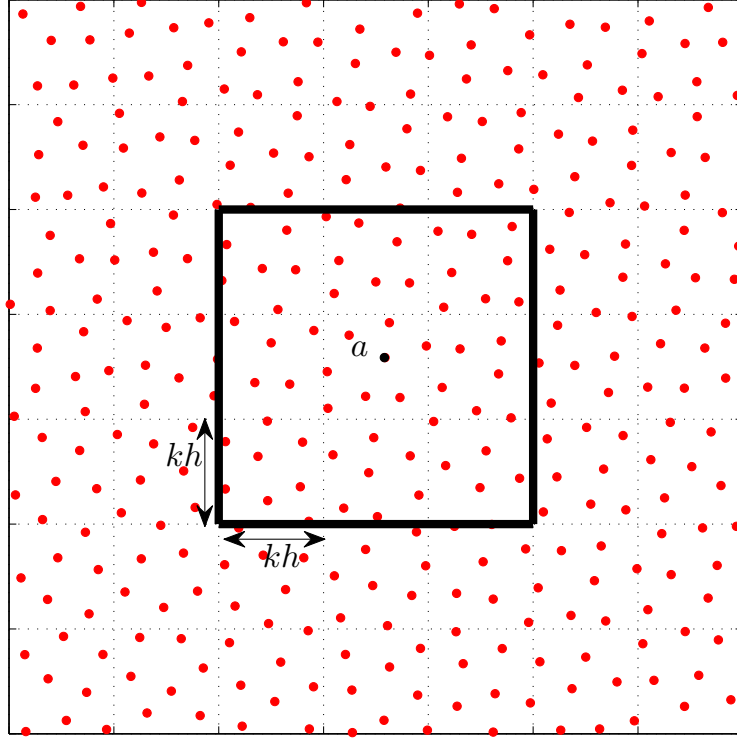


Figure 3.27: Background grid for particle pair searching, grid spaces highlighted for particle a , where k is the number of smoothing lengths, h , used by a particular kernel to define the region of compact support.

the relationship is cubic, thus increasing the total number of pairs. In three-dimensional simulations the computational time is also greatly increased because of the large increase in the number of particles needed to maintain an equivalent resolution.

The development of parallel SPH codes has become more popular in recent years due to the demands of performing both high resolution and three-dimensional simulations. In parallel SPH codes, the overall computational time is reduced by splitting the model domain into a number of sub-domains. The calculations for each sub-domain are then carried out by separate processors and the results recombined on each time-step. More recently a number of authors have also converted their SPH codes to run on Graphics Processing Units (GPUs) in order to gain further efficiency, for example, that developed by Crespo *et al.* (2011).

3.13 Conservation properties of SPH

In the absence of external forces, mass, linear and angular momentum, and energy must all be conserved within a continuum. This section describes how these conservation properties behave in an SPH model.

Modelling a fluid flow with SPH requires the whole fluid to be represented by SPH particles, each particle carrying with them a fixed mass relating to the initial spacing of the particles. As each particle's mass remains the same for the entire simulation, conservation of mass is automatically upheld.

Total linear momentum is conserved if the rate of change of total linear momentum, which is equal to the sum of all the internal forces, is equal to zero. The internal forces consist of the pressure gradient and the viscosity term given in Equation (3.26). Regardless of the chosen form of the viscosity, these terms satisfy the action-reaction law. The force applied to a particle, a , by another particle, b , is equal and opposite to the force applied to particle b by particle a ; the asymmetry arises from the term, $\nabla_a W_{ab} = -\nabla_b W_{ab}$. As this is true for all particle pairs within the domain, it can be seen that the total sum of the internal forces for all interacting pairs will vanish. As a result, linear momentum is conserved (Bonet & Lok (1999)).

Angular momentum has already briefly been discussed in § 3.4 as its conservation properties are mainly related to the chosen representation of the viscosity term. Angular momentum is preserved if the total moment of the internal forces about the origin vanishes. This can only occur if the interaction force is co-linear with the vector $\mathbf{r}_b - \mathbf{r}_a$ that joins two particles (Bonet & Lok (1999)). This is true of the pressure gradient term, where there are no moments between pairs of interacting particles, and so the total moment of internal pressure forces is exactly equal to zero (Khayyer *et al.* (2008)). This is also the case if the artificial viscosity term, given by Equation (3.32), is adopted. However, if the more realistic viscosity term, given by Equation (3.38), is used this is no longer true. In this case the viscous forces do not act along the line $\mathbf{r}_b - \mathbf{r}_a$ and will, therefore, produce a moment. The sum of these moments over the whole domain is not necessarily equal to zero and so angular momentum may not be conserved exactly (Khayyer *et al.* (2008)). In general, angular momentum will only be conserved when the stress tensor is isotropic, which can only happen in the absence of shear stresses (Bonet

& Lok (1999)). However, Violeau & Issa (2007) noted that the exact conservation of angular momentum may not be important for violent flows, particularly where these involve large deformations of the free-surface.

As far as energy conservation is concerned, Violeau & Issa (2007) show that with the SPH method, the energy dissipation rate is related to the viscous forces. This is exactly as expected and is the same as that which occurs with continuous equations.

3.14 Errors and accuracy in SPH

As with any numerical method, small numerical errors arise as the continuous equations of fluid flow are approximated on a computational mesh or, in the case of SPH, on moving interpolation points (or particles). Errors arise both in the integral interpolant, where the kernel function is used to approximate the dirac delta function, and in the summation interpolant, where quantities are approximated by using discrete particles rather than continuous space (Monaghan (2005)). The accuracy, in both cases, depends on the kernel being used. However, with all kernels, the interpolation is of at least second order. Errors in the summation interpolant and its derivative are negligible provided the smoothing length (h) is greater than the particle spacing and the particles are equi-spaced in all space. However, when modelling real flows the particles will not always remain equi-spaced and boundaries will be encountered. It is therefore necessary to consider how the errors change as the fluid motion is modelled.

According to Monaghan (2005), the magnitude of the errors is not easy to estimate as the particles become disordered and the motion evolves. However, he found that the errors are smaller than might be expected if the particles were all displaced randomly. At rest, SPH particles have a tendency to fall into an almost regular cell structure, whereas in a breaking wave, for example, the particles behave as if they are a sheared regular array. Error estimation is very different in SPH compared with traditional mesh-based methods. In particular, the particle disorder depends on the flow dynamics, with particles moving to suit the interpolation formula, rather than the best interpolation formula being sought for the mesh. For this reason, the accuracy of SPH is usually assessed by comparisons with experiments or seeking convergence by increasing particle resolution. Another reason that the accuracy of SPH may be better than would be

expected considering the particle disorder during motion, is that the SPH form of the governing equations are devised to conserve momentum and energy (see § 3.13). As a result, the conservation properties are very important in SPH. For a complete discussion of the errors and the accuracy of SPH, the reader is directed to Monaghan (2005).

3.15 Other modelling techniques

The Riemann solver approach was originally introduced into SPH modelling by Vila (1999). The Riemann problem is solved between each SPH particle pair and eliminates the need for a stabilising viscosity term, whilst conserving both total energy and momentum. Subsequently, Riemann solvers have also been used in SPH models by a number of other authors such as Inutsuka (2002), Roubtsova & Kahawita (2006) and Ferrari *et al.* (2007). Roubtsova & Kahawita (2006) showed that this technique was promising for modelling free-surface flows, particularly if interactions with structures are involved. These calculations were based upon studies of a collapsing dam, a sloshing tank and the Vaiont natural disaster. More recently, the Riemann solver approach of Vila (1999) was used by Rogers *et al.* (2010) to study impact forces on a caisson breakwater. They showed that the Riemann solver approach produced results in better agreement with experiment than their standard SPH model for displacement of the caisson.

Colagrossi & Landrini (2003) introduced interfacial flows to their SPH model by simulating the air as well as the water with SPH particles. This technique involved adapting the standard SPH formulation of the governing equations to be able to cope with the sharp density gradient across the interface between the two fluids. The model was tested on a rising air bubble in water and on backwards plunging breakers entrapping air. Comparisons with experiments were undertaken and found to be in qualitative agreement. This work was extended by Grenier *et al.* (2009) who made further improvements to the SPH model for interfacial flows through the use of a Hamiltonian system of particles.

Other modifications of the basic SPH model include the introduction of time-varying smoothing lengths (Oger *et al.* (2006)), a method for moving rigid bodies (Monaghan *et al.* (2003)) and the introduction of a friction model (Monaghan *et al.* (2003)). There has also been some discussion over the usefulness of modelling surface tension and the

inclusion of a model to correct for tensile instability, the latter resulting in the tendency for particles to clump together.

Extracting specific results from SPH can also be non-trivial, particularly as the free-surface is not defined exactly. Gomez-Gesteira *et al.* (2005) created a numerical wave gauge to measure the free-surface position. However, the method introduces a somewhat arbitrary definition of the free-surface which has a compromising effect on its accuracy. Furthermore, Oger *et al.* (2006) developed a method for measuring the local pressure on a solid boundary.

SPH models have been adapted by a number of authors to model the Shallow Water Equations (SWE). These equations assume a hydrostatic pressure distribution and are typically used with Eulerian mesh-based methods to model coastal flows over relatively large areas. However, these methods often encounter difficulties when regions of the model domain are subject to wetting and drying due to tidal flows or flooding. For this reason, using SPH to solve the SWEs can be advantageous as the problem of wetting and drying is intrinsically resolved. Within SPH models, this approach was first introduced by Rodriguez-Paz & Bonet (2005) and was later employed by De Leffe *et al.* (2008), who solved the SWEs to simulate a dam break and solitary wave propagation, and more recently by Vacondio *et al.* (2010), who simulated flood inundation due to dam breaks, tsunamis and levee breaches. Although this approach shows clear benefits for modelling these types of flows, the hydrostatic pressure assumption renders this method inappropriate for modelling the more violent flows described in this thesis.

Recently, several authors have coupled SPH models with other numerical models in order to draw upon the advantages of both and overcome some of the difficulties of SPH. For example Kassiotis *et al.* (2011) coupled an SPH model with a one-dimensional Boussinesq-type wave model. Boussinesq models are known for their accuracy in modelling wave propagation before breaking, whereas SPH is more beneficial in capturing the breaking itself, but often suffers from dissipation during the propagation phase. In the coupled model, the Boussinesq model is used to drive the SPH component by providing the water height and velocity at the inlet boundary. A second example is provided by Fourey *et al.* (2010) who coupled an SPH model with a FEM model for simulating violent impacts with solid structures. In this case, the SPH model was used to capture the complex free-surface flows, whilst the FEM was used to represent the solid structure.

3.16 Weakly compressible versus incompressible formulation

In SPH the particles movements are driven by their interactions with one another. As they move closer together the pressure increases, driving them further apart, and keeping them moving in the direction prescribed by the equations of motion. The allowed particle proximity is defined by the speed of sound chosen for a particular simulation. This choice is designed to prevent the flow from being overly compressible, but at the same time allowing some compressibility, thereby facilitating a reasonable time step. It therefore follows that although in reality water is an incompressible fluid, the SPH representation of water is usually weakly compressible. Since the particle motion is driven in this way, it seems at odds to describe a variant of the SPH method as being truly incompressible. However, this method is nevertheless chosen by a number of authors after its initial introduction by Cummins & Rudman (1999). The main difference between the weakly compressible SPH (WCSPH) and the incompressible SPH (ISPH) formulations lies in the calculation of the pressure. In the WCSPH methods the system of equations is closed by calculating the pressure through an equation of state, whilst in ISPH methods the pressure is calculated by solving a pressure Poisson equation. The latter approach prevents small, artificial fluctuations in the density and pressure field. However, using a density correction method, such as those discussed in § 3.7, can improve the quality of the pressure field predicted with WCSPH. Although ISPH allows larger time steps than WCSPH, ISPH requires a system of matrices to be solved, often by a chosen iterative process, thus reducing the benefits to the overall simulation time.

WCSPH and ISPH were directly compared by Lee *et al.* (2008) through comparisons based upon lid-driven cavity flow and a breaking dam. Although Lee *et al.* (2008) found that the pressure and velocity fields were more accurate with the ISPH method, no correction technique was applied to the density calculation. Furthermore, it is clear from the images comparing the dam break to experimental data, that the ISPH has not been able to simulate the free-surface profile correctly. In particular, particles appear to spray out in all directions from the overturning jet, perhaps indicating a lack of surface tension. Discrepancies close to the boundaries in the collapsing dam simulations could also be improved in the WCSPH method by selecting a better boundary model for this

type of problem. The two methods were again compared to each other and also to experimental observations by Lee *et al.* (2010), who this time investigated the collapse of a three-dimensional water column and impact with an obstacle. The results showed clearly that the pressure calculations with ISPH are more accurate than with WCSPH. However, as in the previous work, no density filter was applied to the WCSPH model, which has been used by many other authors to improve the pressure field. Again, a lot of unrealistic surface spray is visible in images taken from the ISPH simulations. In another comparison of WCSPH and ISPH models, Hughes & Graham (2010) also found that the ISPH model produced excessive spray.

Shao (2006) also used an ISPH model, including a $k-\epsilon$ turbulence model, to simulate waves overtopping a sea wall. Particle snapshots are given throughout the process. However, a direct comparison with experimental data is not given during the actual breaking process. Furthermore, the aspect ratio of these snapshots makes it very difficult to assess the quality of the breaking wave. Khayyer *et al.* (2008) added a correction term to this ISPH model and compared results using the new model to those without the correction for solitary waves breaking on a plane slope. Both sets of results are qualitatively compared to experimental photographs throughout the breaking process. It is clear from these results that without this correction the ISPH method is completely unable to capture the wave breaking process. In particular, what should be the distinct profile of the overturning jet appears as a mass of particles spraying in all directions, bearing no resemblance to the correct form of the evolving wave profile. The correction to the model undoubtedly improves the results arising from this simulation, but the overturning jet still appears to have a lot of particles spraying outwards. Such effects are not visible in the experimental observations. An ISPH formulation was also used by Ataie-Ashtiani *et al.* (2008), who considered solitary waves and collapsing dams. The pressure field is better represented than with standard SPH, however, the free-surface still appears messy.

Improvements to the basic ISPH model have been made by Xu *et al.* (2009) and Lind *et al.* (2012). In the work by Xu *et al.* (2009) the stability and accuracy of three variations of existing ISPH models, along with a new approach, were tested on enclosed flows such as vortex spin-down. The new method combines elements of a divergence-free approach with a density invariant method, and in addition shifts particles slightly across

streamlines to avoid problems with particle clustering and spacing distortion. They found that this method produces both stable and accurate results without sacrificing the efficiency of other approaches, but ceases to be strictly conservative. Lind *et al.* (2012) builds upon the work of Xu *et al.* (2009) by extending the method specifically for modelling free-surface flows. Indeed, a new algorithm is introduced for shifting particles at the free-surface and the approach is validated by making comparisons of a dam break and progressive regular waves with analytical solutions. Once again, the authors cite the noise-free pressure distribution and lack of dissipation as being the main advantages of the method. However, no violent free-surface flows were simulated with this approach so it was not possible to observe any improvements in the appearance of the free-surface under these conditions.

The main advantages of using ISPH methods relates to their ability to produce a noise-free pressure distribution and suffer less due to dissipation than WCSPH methods. However, it will be shown later in this thesis that these issues can be overcome by carefully selecting various aspects of the WCSPH model. Several problems with modelling the free-surface in violent flows, such as breaking waves, were also identified in the literature. For these reasons ISPH will not be considered further.

4

An SPH study of solitary wave propagation

4.1 Introduction

The purpose of this chapter is to collate the optimum SPH modelling techniques to describe free-surface flows, specifically for simulating solitary waves propagating over a distance of the order of a hundred times the wave height. The overall objective of this thesis is to investigate the modelling of free-surface flows with SPH and so this chapter looks at a controlled case study designed to find the best possible modelling techniques. The propagation of solitary waves has been chosen for a number of reasons. First, it is a relatively simple study to set up numerically, with plenty of theoretical and experimental solutions available for comparison. Second, the simulations work well in two dimensions as solitary waves have, theoretically at least, no three-dimensional structure. Third, the SPH model is not expected to require complex modelling techniques, such as bottom friction or turbulence modelling, as these are not important in such a flow. Finally, the following chapter in this thesis (Chapter 5) goes on to investigate solitary waves breaking on a slope, a process that involves many aspects of numerical modelling for which SPH is thought to be a particularly appropriate method. For this next investigation to be truly quantitative, it is important that the solitary wave is behaving as it should do before the wave begins to overturn. The work presented in the current chapter will confirm whether this is indeed the case.

This chapter contains a review of solitary wave theory, the generation of solitary

waves for numerical and laboratory models and previous work in § 4.2. This is followed by a description of the SPH model and details of the numerical simulations used in this study in § 4.3. Components of the SPH model are tested and selected in § 4.4 and the resulting SPH model is then used in a more extensive study, with thorough quantitative comparisons made to theory in § 4.5. Comparisons of the numerical predictions are also made with laboratory observations in § 4.6 and, finally, the main conclusions are drawn in § 4.8.

4.2 Background

4.2.1 Solitary wave theory

A solitary wave is a wave of permanent form, consisting of only the rising crest above still water level with no adjacent wave troughs. The wave is symmetric about the point of its maximum surface elevation and has infinite wavelength, although an effective wavelength can be defined according to Rayleigh (1876).

Since Russell’s original experiments and classification of the solitary wave (Scott Russell (1838), Scott Russell (1845)), a number of theories have been put forward to describe solitary wave characteristics. These largely fall into three groups. The first start from the Euler equations and use series expansions of the wave amplitude. Such solutions include Boussinesq’s first-order theory (Boussinesq (1872)), Laitone’s second-order theory (Laitone (1960)), Grimshaw’s third-order theory (Grimshaw (1971)) and Fenton’s ninth-order theory (Fenton (1972)). The second group are based on a sinusoidal series and include theories due to Stokes (1847), Rayleigh (1876) and McCowan (1891). The third group correspond to more recent theories based on exact integral equations for the surface profile. These are solved numerically and include work by Milne-Thomson (1964) and Byatt-Smith (1970).

The simplest of these theories, still considered to be accurate enough to describe solitary waves with small height to depth ratios, is that given by Boussinesq (1872). Details of this solution are given below. The theory is based on an irrotational, incompressible ideal fluid, allowing a potential flow representation. The wave surface profile, η , is given by

$$\eta(x, t) = H \operatorname{sech}^2(\kappa X), \quad (4.1)$$

where x is horizontal position, t is time and H is the wave height. Within this solution

$$X = (x - ct), \quad (4.2)$$

$$\kappa = \sqrt{\frac{3}{4} \frac{H}{d^3}}, \quad (4.3)$$

where d is the water depth, and the wave speed, c , is given by

$$c = \sqrt{g(d + H)}. \quad (4.4)$$

The horizontal water particle velocity, u , at the bed ($z = -d$, where z is measured vertically upwards from the undisturbed water surface) is assumed to be the same throughout the depth of the fluid and is given by

$$u = \frac{c\eta}{d + \eta}. \quad (4.5)$$

In formulating this solution, Boussinesq did not give an equation for the vertical water particle velocity.

McCowan's alternative theory (McCowan (1891)), although also of first-order, is considered to give a more accurate description of the solitary wave form as well as including depth dependant equations for both the horizontal and the vertical water particle motion (Sander & Hutter (1991)). McCowan gives the surface profile as

$$\eta(x, t) = \frac{dN \sin(M(1 + \eta/d))}{M[\cos(M(1 + \eta/d)) + \cosh(M\frac{X}{d})]}, \quad (4.6)$$

with the wave speed, c , defined by

$$c = \sqrt{\frac{gh}{M} \tan M}. \quad (4.7)$$

Within this solution, the horizontal fluid particle velocity, u , is given by

$$u = \frac{cN(1 + \cos(M\frac{z}{d}) \cosh(M\frac{X}{d}))}{(\cos(M\frac{z}{d}) + \cosh(M\frac{X}{d}))^2}, \quad (4.8)$$

where z is again the vertical position defined relative to the still water level, and the vertical water particle velocity, v , is given by

$$v = \frac{cN \sin(M \frac{z}{d}) \sinh(M \frac{X}{d})}{(\cos(M \frac{z}{d}) + \cosh(M \frac{X}{d}))^2}. \quad (4.9)$$

The relationship between N and M is given by the following two equations:

$$N = \frac{2}{3} \sin^2 \left[M \left(1 + \frac{2}{3} \frac{H}{d} \right) \right] \quad (4.10)$$

$$\frac{H}{d} = \frac{N}{M} \tan \left[\frac{1}{2} M \left(1 + \frac{H}{d} \right) \right]. \quad (4.11)$$

Laitone's second-order theory adds quadratic terms to Boussinesq's linear theory, thereby increasing the accuracy of the formulation. In this case the essential features of the flow field are defined as follows. The wave profile, $\eta(x, t)$, is given by

$$\eta(x, t) = H \operatorname{sech}^2 \left(\alpha \frac{X}{d} \right) \left[1 - \frac{3}{4} \frac{H}{d} \left(1 - \operatorname{sech}^2 \left(\alpha \frac{X}{d} \right) \right) \right], \quad (4.12)$$

where α is given by

$$\alpha = \sqrt{\frac{3}{4} \frac{H}{d}} \left(1 - \frac{5}{8} \frac{H}{d} \right). \quad (4.13)$$

The wave speed is given by

$$c = \sqrt{gd} \left[1 + \frac{1}{2} \frac{H}{d} - \frac{3}{20} \left(\frac{H}{d} \right)^2 + O \left(\left(\frac{H}{d} \right)^3 \right) \right], \quad (4.14)$$

the horizontal water particle velocity is given by

$$u = \sqrt{gh} \left\{ \frac{H}{d} \left[1 + \frac{H}{d} \left(\frac{1}{4} - \frac{3z^2}{2d^2} \right) \right] \operatorname{sech}^2 \left(\alpha \frac{X}{d} \right) + \left(\frac{H}{d} \right)^2 \left(\frac{9z^2}{4d^2} - 1 \right) \operatorname{sech}^4 \left(\alpha \frac{X}{d} \right) \right\}, \quad (4.15)$$

and the vertical water particle velocity is given by

$$v = -\sqrt{gh} \left[\frac{z}{d} \frac{d\eta}{dX} + O \left(\left(\frac{H}{d} \right)^{5/2} \right) \right]. \quad (4.16)$$

As the work in this chapter is concerned with solitary waves of relatively low height to depth ratios, higher order theories will not be considered further. The improvements

to the wave form and speed gained from these theories only start to become apparent at height to depth ratios greater than approximately 0.5 (Fenton (1972)).

4.2.2 Solitary wave generation

In order to generate solitary waves numerically (or experimentally, as will be the case in Chapter 5), we need to be able to program a numerical, or experimental, wave generator. This can be achieved using the theory of Goring (1979), who calculates the horizontal position, ξ , and velocity, $d\xi/dt$, of a vertical piston-type wave board or wave paddle throughout the time, t , needed to generate a first-order Boussinesq solitary wave. The theory is based upon the requirement that the wave paddle has the same velocity as the water particles associated with the desired wave form. Boussinesq assumes the water particle velocities under the wave are constant with depth. As a result, the paddle motion can be purely horizontal, hence the piston-type wave board. In this case

$$\frac{d\xi}{dt} = u(\xi, t), \quad (4.17)$$

from which it is clear that the velocity, u , is a function of the paddle position, ξ , as well as time, t . This is to allow for the wave propagating away from the paddle during generation and has the effect of producing a trajectory which is distorted from what it would look like if the paddle position was not included.

If the surface profile is written in the form

$$\eta = H \operatorname{sech}^2 \theta, \quad (4.18)$$

where

$$\theta = \kappa(ct - \xi), \quad (4.19)$$

it can be shown that the paddle position, $\xi(t)$, is given by

$$\xi(t) = \frac{H}{\kappa d} \tanh \kappa(ct - \xi). \quad (4.20)$$

This equation must be solved numerically at any given time, t . Accordingly, a Newton-Raphson iteration was used to calculate θ via the following equation:

$$\theta^{(i+1)} = \theta^{(i)} - \frac{\theta^{(i)} - \kappa c t + \frac{H}{d} \tanh \theta^{(i)}}{1 + \frac{H}{d} \operatorname{sech}^2 \theta^{(i)}}. \quad (4.21)$$

Once θ has been found, Equation (4.19) can be rearranged and used to find the paddle position, ξ , at the particular time. θ can then be used in Equations (4.18) and (4.5), respectively, to define the paddle velocity at that time.

From the definition of a solitary wave, Equations (4.1)-(4.4), it can be seen that the origin of both the time, t , and displacement, ξ , occur under the solitary wave crest. Theoretically, solitary waves have an infinite wave length and, consequently, an infinite wave period. However, both numerically and experimentally, this is impractical. In order to overcome this difficulty, an effective wavelength it used (Rayleigh (1876)) whereby the wave period is truncated, using the following equation, to calculate the times, $\pm t_0$, at which the leading and trailing edges of the wave pass through the central position, $\xi(0)$,

$$t_0 = \frac{\tanh^{-1}(0.999)}{\kappa c} = \frac{3.80}{\kappa c}. \quad (4.22)$$

The length of the paddle stroke, S , was calculated by putting $t = \pm\infty$ into Equation (4.20) and subtracting to give

$$S = \frac{2H}{\kappa d} = \sqrt{\left(\frac{16}{3} \frac{H}{d}\right)} d, \quad (4.23)$$

where the definition of κ is taken from Equation (4.3). The duration of the motion, τ , used to generate the wave was calculated from the time taken for the leading and trailing edges to pass through the central position plus the time at each end, for the leading edge to travel from the paddle to the central position and for the completion of the trailing edge of the wave. This is given by

$$\tau = 2t_0 + S/c = \frac{2}{\kappa c} \left(3.80 + \frac{H}{d}\right). \quad (4.24)$$

If a solitary wave were to be produced by a less accurate generation technique, such as dropping a weighted box into water as Scott Russell did in his demonstrations in 1844 (Monaghan & Kos (2000)), there would, invariably, be a trail of slower moving oscillatory waves following the solitary wave. Such waves are gradually left behind as

the solitary wave propagates over a significant distance. Goring (1979) found that even using the method of generation described herein, some small amplitude, high frequency waves were produced in the trail and these could be reduced by increasing τ . Indeed, Goring (1979) suggested that τ should be increased by 10% as he found that, in practice, this reduced the amplitude of any trailing waves by 1% to 2%. It is likely that these are due to the paddle following Boussinesq's first order theory for the wave profile and particle velocity under the wave. It is therefore possible that the trailing waves could be further reduced by programming the paddle motion using a higher order solitary wave theory.

4.2.3 Solitary wave propagation

Theoretically, solitary waves propagate over an infinite distance without change in shape. However, in practice they will lose height and energy due to the effects of viscosity and, in particular, their interaction with the domain boundaries including both the bed and the side walls of the flume. In the present numerical calculations, a two-dimensional SPH model is employed. As a result, there are no sides to the flume, or computational domain, and thus the only losses will be due to viscosity and the bottom boundary, or bed. In part, the aim of the present calculations is to propagate a solitary wave with as little reduction in wave height as possible. In this case any observed dissipation is likely to be strongly connected to the viscosity model employed. As a result, this is an important aspect of the model to be explored in the present chapter.

The Boundary Element Method (BEM) is known to be able to propagate waves over long distances extremely well with minimal losses in wave height. This is because the BEM is a potential flow model and so suffers no losses due to the effects of viscosity. The success of the BEM in the context of solitary wave propagation has been demonstrated by Hague (2006). Using a two-dimensional model, she considered solitary waves with $H/d = 0.1, 0.2$ and 0.3 propagating in a 40m numerical wave flume and compared their form to Boussinesq's first-order theory. Although the comparisons relating to the $H/d = 0.3$ case were marginally worse than the excellent comparisons at lower values of H/d , this is more likely to be due to the first-order theory not being as accurate for higher values of H/d than a problem with the BEM's modelling capability. Despite this slight discrepancy, the comparisons provided by Hague (2006) were all very good.

Due to the nature of SPH, it is necessary to use some form of viscosity model to stabilise the method. As a result SPH cannot strictly model potential flow. However, since real fluids do not behave as ideal fluids, with potential flow properties, this should not present a significant problem for modelling solitary waves. Nevertheless, some loss of wave height is to be expected. Indeed, this would also be true when modelling solitary waves experimentally. Since the SPH model used in the present chapter is two-dimensional, the expected loss of wave height is difficult to pin-point. In particular, comparisons with laboratory observations will be inconsistent since these include the presence of side walls which will cause more losses than those arising in a two-dimensional model. Furthermore, in an experimental study the reduction in wave height also depends on the smoothness of the walls. However, within the SPH calculations, reductions in the wave height due to viscosity in the fluid should be small and it is therefore important to attempt to model solitary wave propagation with as little loss in wave height as possible. At the very least, the two-dimensional model calculations should not involve a greater reduction in the wave height when compared to laboratory observations.

One of the earliest attempts to model solitary waves using SPH was outlined by Monaghan & Kos (1999), and primarily concerns the run-up of solitary waves on a slope. Within this study the authors used an alternative method of solitary wave generation. In this case the simulations were initiated with SPH particles located on a regular Cartesian grid, but constructed into the shape of a solitary wave and given an appropriate particle speed depending on their location under the wave. This approach was not chosen for use in the present study since, at the start-up of any simulation, there is a period of time where the SPH particles move from their Cartesian grid locations to settle into more stable positions relative to one-another. This alters the solitary wave form, making it unlikely to propagate as a true solitary wave. Such effects hinder comparison with available solitary wave solutions and also complicate direct comparisons with laboratory generated data.

Lo & Shao (2002) used an incompressible SPH model to simulate solitary waves propagating over a distance of approximately one hundred times the depth of the water and succeeded in maintaining the original wave height and form over this distance with a wave speed very close to its theoretically predicted value. These results suggest that an incompressible SPH model may provide better results than the more typical compress-

ible SPH models. For example, Vaughan *et al.* (2004) adopted a weakly-compressible SPH model and showed that a propagating solitary wave suffered large decreases in wave height when propagating over relatively modest distances. However, one of the fundamental reasons for choosing to apply SPH to model free-surface flows is their ease at simulating violent flows with large deformities and surface break-up, such as that resulting from breaking waves. With this in mind, Khayyer *et al.* (2008) showed that in order to model the breaking of solitary waves, it was necessary to add corrective terms to the incompressible SPH model of Shao & Lo (2003). However, even after the application of these corrections, the results look no better than similar results achieved using a relatively simple weakly-compressible SPH model. As a result, incompressible SPH methods were rejected for the model being developed herein. Indeed, it is proposed that with careful selection of specific modelling techniques, solitary waves can be successfully propagated over long distances using a weakly-compressible SPH model.

4.3 Numerical simulations

4.3.1 SPH model

Solitary waves were generated in a two-dimensional, numerical flume with a numerical wave paddle at one end, programmed to move as described in § 4.2.2. The flume was 25m long, including the distance through which the wave paddle moved to produce the waves, and ran with an initial water depth of $d = 0.5\text{m}$. As the SPH model evolved throughout the testing process, details of specific aspects of the model are given in the discussion of results. Unless otherwise specified, the following applied in each test case:

- The entire water body was made up of approximately 125,000 SPH particles, initially placed on a Cartesian grid, with an initial particle spacing of $\Delta x = \Delta y = 0.01\text{m}$ or $d/\Delta x = 50$.
- The smoothing length was $1.3\Delta x$.
- The speed of sound was calculated following Equation (3.29), with $v_{max}^2 = g(d+H)$ taken from Boussinesq's first-order solitary wave theory (Boussinesq (1872)).
- A variable time step size was used based on the Courant condition given by Equa-

tion (3.88) and the additional condition given by Equation (3.90), resulting in time steps of the order of 10^{-4} s.

- Solitary waves were generated with $H/d = 0.3$.

After the optimum SPH model was found, further simulations were conducted with $H/d = 0.1, 0.2$ and 0.45 in order to obtain further comparisons with theory and ensure the model was not solely optimised for one particular value of H/d .

4.3.2 SPH wave gauges

In order to produce time histories of the free-surface at specific locations along the numerical wave flume, the concept of SPH wave gauges was introduced. These could be placed at any location in the wave flume. However, for the majority of the simulations carried out for the studies discussed in this chapter, the gauges were placed to the right of the paddle's initial position at $x = 1\text{m}$, 2m and then at 2m intervals up to 20m . In SPH the free-surface can clearly be seen from looking at scatter plots of all the particles, but in order to view time histories at the wave gauges, some form of interpolation must be employed. Therefore, at each wave gauge and at every time-step, the highest particle lying within a distance, δx , on either side of the wave gauge was found. Linear interpolation was then used to find the free-surface height at the location of the wave gauge. The distance, δx , was chosen to be $2.5\Delta x$, where Δx is the initial particle spacing. This was chosen to be large enough that a particle forming part of the surface was highly likely to be found, but at the same time, small enough to ensure that details of the wave field, such as the peak of the solitary wave crest, were not lost. A diagram of an SPH wave gauge is shown in Figure 4.1. The highest particles on either side of the wave gauge lying within the region $\pm\delta x$ are circled.

4.4 Optimising the accuracy of the SPH model

4.4.1 Basic SPH model

Initially, a *basic* SPH model was used to simulate solitary waves with a height to depth ratio of $H/d = 0.3$, with the aim of improving upon this model through a case study

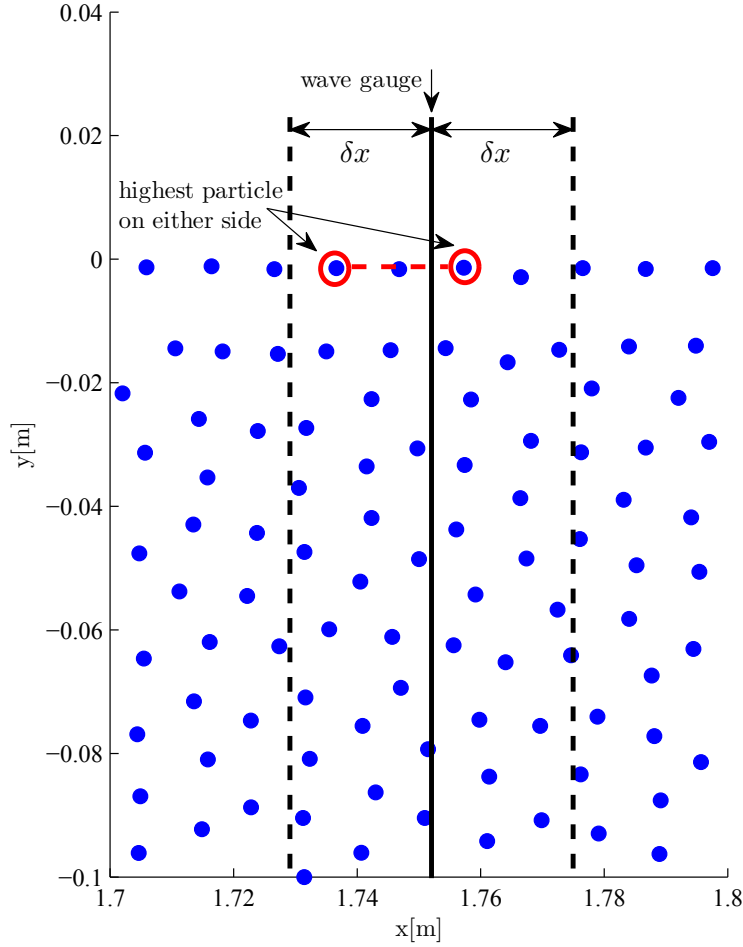


Figure 4.1: Numerical wave gauge, identifying the highest particles, $\bigcirc$, and indicating the linear interpolation, $- - -$.

of model components. The *basic* model was mainly constructed from techniques described by Monaghan (1994). The general SPH equations for the continuity equation (Equation (3.18)), the momentum equation (Equation (3.26)) and the equation of state (Equation (3.27)) were applied. The viscosity was modelled by Monaghan’s original artificial viscosity equation (Equation (3.32)) and neither XSPH nor any form of density smoothing were included in order to keep the model simple. The cubic kernel devised by Monaghan & Lattanzio (1985) and given by Equation (3.5) was applied and the boundaries were modelled using the repulsive force representation, with the equations given by Monaghan *et al.* (2003), given by Equations (3.70) and (3.73)-(3.76). Time marching was carried out using the method described by Monaghan (2007) and given in equations (3.91)-(3.96). It would have perhaps been fairer to include one of the simple

and more frequently used time-marching schemes, such as a predictor-corrector or Verlet scheme, in this basic model. However, at the time of carrying out this case study, it was already clear from preliminary work, that the scheme given by Monaghan (2007) was considerably better than the other more commonly used schemes.

The time history of the relative wave-height resulting from the simulation with the *basic* model is shown in Figure 4.2. Each colour on the graph defines the solitary wave profile predicted at a numerical wave-gauge located at a specific distance from the paddle-end of the wave flume, details of the wave gauge locations being provided with the accompanying legend. Although the paddle has initially produced a wave of the correct height, the dissipation of the wave height is immediately obvious. Indeed, by the time the wave has propagated 18m along the wave flume (a distance equivalent to 120 times the original wave height or 36 times the water depth), its height has been reduced to only 72% of its original height.

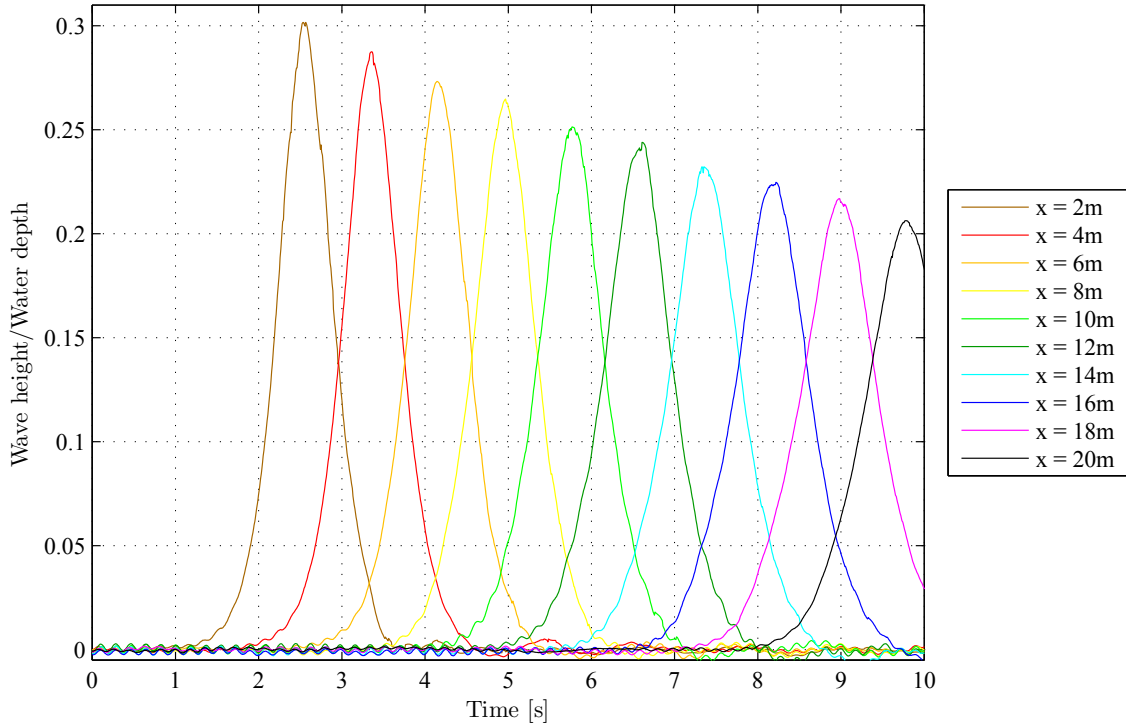


Figure 4.2: Solitary wave propagation, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of numerical wave gauges using the *basic* SPH model. (Note: The legend defines the location of the wave gauges measured from the position of the paddle at the beginning of its motion, $x = 0$.)

The solitary wave produced by this model is clearly not of permanent form, the reduction in wave height being far beyond that which could be expected from frictional losses. Given this result, it is clearly necessary to investigate the best SPH modelling techniques available to describe a solitary wave of permanent form.

4.4.2 Model testing

In order to improve upon the basic model, a case-by-case study was undertaken. For this study, the simulated solitary waves kept the same height to depth ratio of $H/d = 0.3$ as those tested in the *basic* model simulations. Various aspects of the model were isolated and tested individually in order to gauge their particular effect on the predicted data. These included the following:

- Boundaries
- Viscosity
- XSPH
- Density Smoothing
- Kernels
- Time Stepping Schemes

Details of the modelling techniques used in each test case are given in Table 4.1. Taking each column in turn, the definitions and descriptions of the terms employed are as follows:

Column 1: This simply defines the case number.

Column 2: This gives the type of boundaries used, where *Repulsive Force* refers to the repulsive force representation used in the *basic* model and *Stationary Ghosts* refers to the representation employed by Violeau & Issa (2007) and described in § 3.8.

Column 3: This specifies the viscosity representation. *Artificial* refers to the version used in the *basic* model. *Improved Artificial* refers to the improved formulation used by Colagrossi & Landrini (2003) in an attempt to reduce dissipation, full

details of which were given in Equations (3.33)-(3.37). *Realistic* refers to the model described by Morris *et al.* (1997) and given in Equation (3.38).

Column 4: This indicates whether XSPH (Equations (3.62) and (3.63)) was switched on or off.

Column 5: This specifies whether any form of density smoothing was used. *Density Re-initialisation* refers to that given by Colagrossi & Landrini (2003) and defined in Equations (3.64)-(3.68), while the *Shepard Filter* refers to the re-initialisation technique used by Dalrymple & Rogers (2006) and given in Equation (3.69).

Column 6: This indicates which kernel was used: either the cubic kernel used in the *basic* model or the quartic kernel used by Violeau & Issa (2007) and given by Equation (3.7).

Column 7: This defines the time-stepping scheme used. *Monaghan* refers to that used in the *basic* model and *Predictor-Corrector* refers the traditional predictor-corrector scheme.

Case 1 is the *basic* model case, which has already been described above. In *Case 2* the *Repulsive Force* boundary model was swapped for the *Stationary Ghosts* model. Violeau & Issa (2007) favoured this latter method as it ensures the validity of the conservation laws and, unlike the *Repulsive Force* model, it does not require any ad hoc coefficients. The results of *Case 2* are compared with *Case 1* and Boussinesq's first-order theory in Figure 4.3. Here, and in all of the following case comparison results, the numerical results are taken from the wave gauge located at 18m from the paddle position (when it is at the beginning of its motion) and the theoretical curve is aligned in time with the best results in each case. It is clear from these comparisons that, although the *Stationary Ghosts* model has by no means produced perfect results, it provides a huge improvement on the *Repulsive Force* model and therefore has the potential to produce better predictions when used in combination with other improved techniques. As the physics behind this model are also an improvement for the reasons mentioned above, and no adverse effects were encountered, this boundary representation was selected for use in all further simulations.

Case	Boundary Model	Viscosity Model	XSPH	Density Smoothing	Kernel	Time Integration
1	Repulsive Force	Artificial	Off	None	Cubic	Monaghan
2	Stationary Ghosts	Artificial	Off	None	Cubic	Monaghan
3	Stationary Ghosts	Improved Artificial	Off	None	Cubic	Monaghan
4	Stationary Ghosts	Realistic	Off	None	Cubic	Monaghan
5	Stationary Ghosts	Realistic	On	None	Cubic	Monaghan
6	Stationary Ghosts	Realistic	Off	Density Re-initialisation	Cubic	Monaghan
7	Stationary Ghosts	Realistic	Off	Shepard Filter	Cubic	Monaghan
8	Stationary Ghosts	Realistic	Off	None	Quartic	Monaghan
9	Stationary Ghosts	Artificial	Off	None	Cubic	Predictor-Corrector

Table 4.1: Modelling techniques used in a case study for the development of the SPH model.

Case 3 and *Case 4* investigate alternative methods for modelling viscosity. Colagrossi & Landrini (2003) chose to use the *Improved Artificial* viscosity model as they found that although Monaghan’s original model was good for improving stability properties, the *Improved Artificial* model reduced the generation of spurious entropy. Monaghan (2005) concurred that this new model was indeed better at reducing artificial dissipation for flows not involving shocks, noting that the original model was specifically designed to include shocks. The *Realistic* viscosity model comes from Morris *et al.* (1997), who disliked the velocity profiles that were produced when modelling low Reynolds number flows with the *Artificial* viscosity model. As a result, they decided

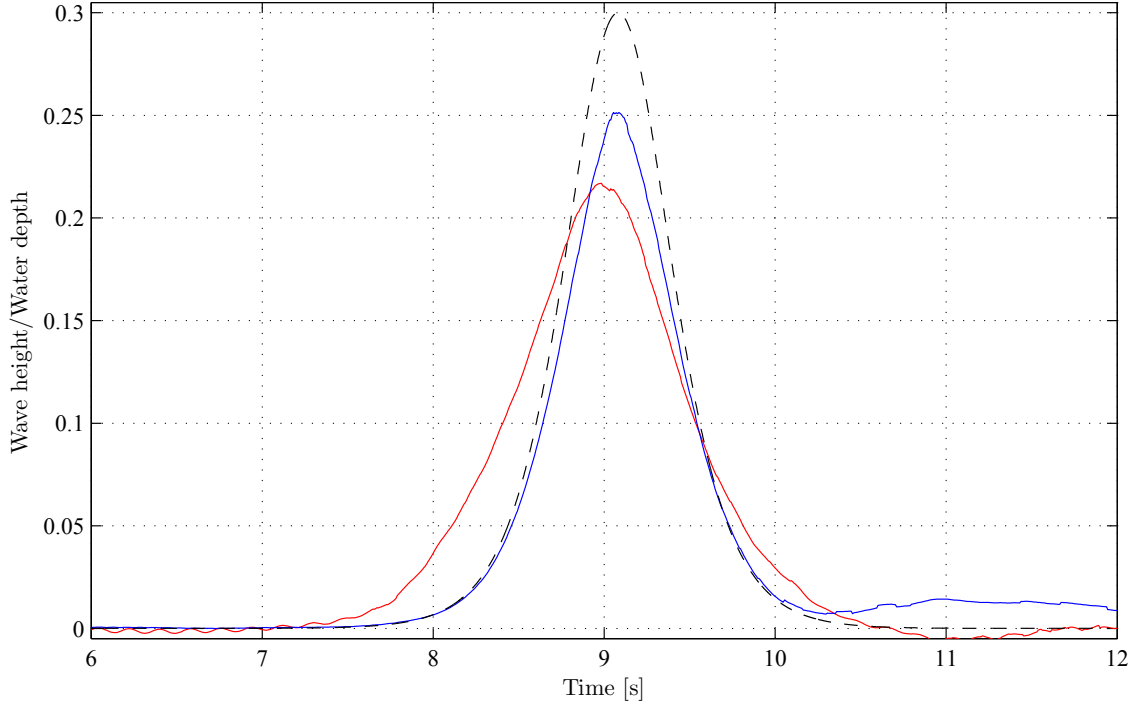


Figure 4.3: Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons between results based on different boundary models: — Case 1 (*Repulsive Force* model), — Case 2 (*Stationary Ghosts* model), - - - - Boussinesq's theory.

to base their viscosity model on a more realistic format. Previously, SPH forms of the realistic viscosity equation had been rejected due to errors arising from the second derivative. However, Morris *et al.* (1997) overcame this problem by introducing a finite difference approximation along with a standard SPH differentiation to calculate the second derivative more accurately. Violeau & Issa (2007) investigated the properties of the *Artificial* viscosity and *Realistic* viscosity flows and reached the conclusion that although only the *Artificial* viscosity (and *Improved Artificial* viscosity) model conserved angular momentum, the *Realistic* viscosity model has other properties which are more important for the violent flows they intended to model.

The results of *Case 3* and *Case 4* are compared with the results of *Case 2* and Boussinesq's first-order theory in Figure 4.4. These comparisons confirm that the *Improved Artificial* viscosity model is clearly an improvement on the *Artificial* viscosity model in terms of reducing the degree of dissipation as the wave propagates. However, the *Realistic* viscosity model gives even better results, reducing the wave height loss

error to below 5%. However, the *Artificial* viscosity and *Improved Artificial* viscosity models were designed to stabilise the SPH model as a whole, as well as representing the viscosity, and this is reflected in the smooth water surfaces provided by these models, particularly around still water level. In contrast, the predicted water surface close to the still water level produced with the *Realistic* viscosity model is bumpy by comparison, particularly at times before the wave has passed the gauge. However, in terms of continuing this model testing process, the *Realistic* viscosity model has been selected for its superior propagation ability and the likely use of the SPH model for simulating violent flows where strict conservation of angular momentum is less important.

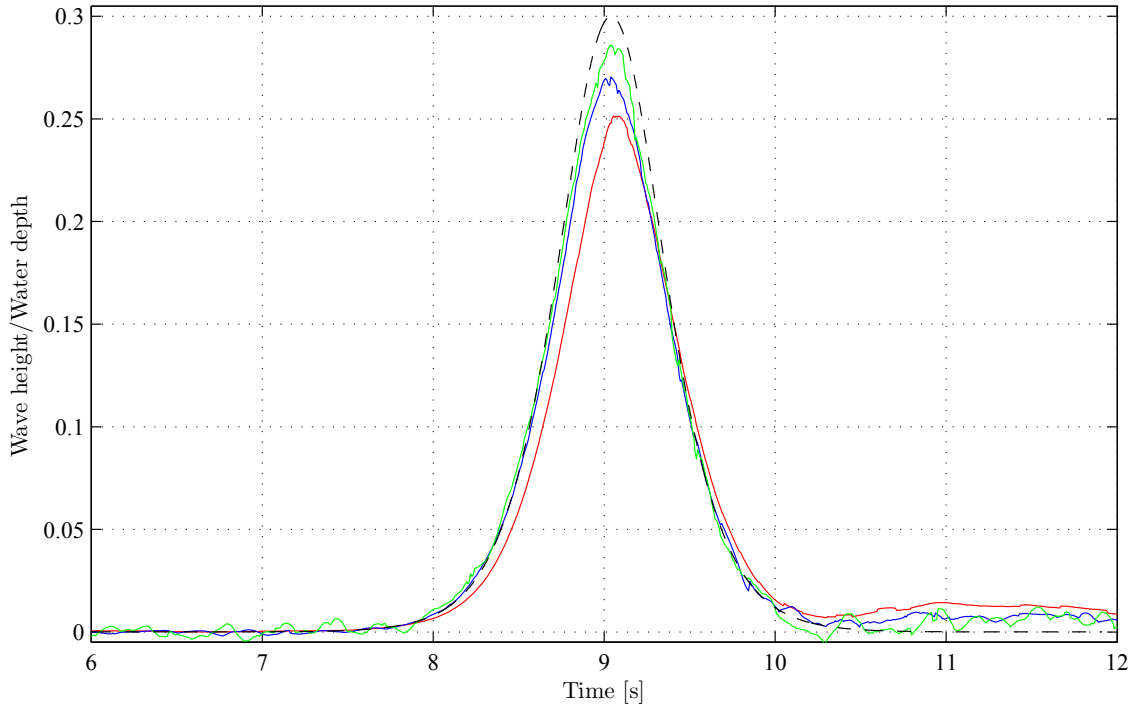


Figure 4.4: Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons between results based on different viscosity models: — Case 2 (*Artificial* model), — Case 3 (*Improved Artificial* model), — Case 4 (*Realistic* model), - - - Boussinesq's theory.

For *Case 5* XSPH was switched on. Monaghan (1994) originally introduced the XSPH correction to keep particles more orderly and prevent particles from penetrating one another in high speed-flows. The results of *Case 5* are shown in Figure 4.5, with direct comparisons to the results of *Case 4* and Boussinesq's first-order theory. These comparisons suggest that there is little difference in the predicted wave height between

the two cases, although *Case 5*'s wave height was marginally higher as it passed the gauge. Furthermore, the bumpiness of the still water level, both before and after the passage of the wave crest, indicates that the particles were not being kept more orderly in an area of the flow where this would be particularly desirable. Also, the mean water level in these regions is slightly elevated above zero, introducing further inaccuracy into the predicted results. As high-speed flows, or flows where particles are likely to collide at high speeds, are not of principal concern in this thesis, XSPH was not used further in the test programme. For what is essentially a very small increase in wave height, it was not considered worthwhile introducing an artificial correction into the flow, with a large consequent increase in the runtime.

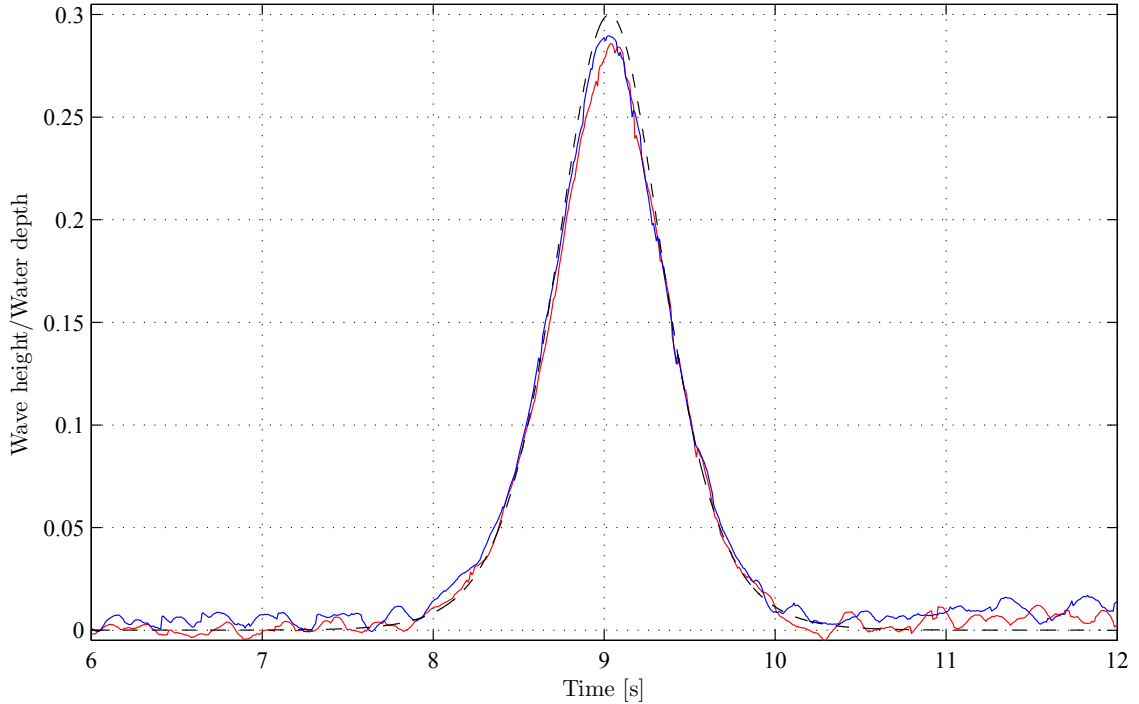


Figure 4.5: Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons between results with and without the inclusion of the XSPH correction: — Case 4 (without XSPH), — Case 5 (with XSPH), - - - - Boussinesq's theory.

Case 6 and *Case 7* investigate the introduction of a density smoothing technique to the model. As the density in the SPH model is calculated through the continuity equation, slight differences between the mass, density and occupied area of the flow as a whole are allowed to build up. If the summation equation (Equation (3.10)) was used

to calculate the density, conservation between the three properties would automatically be held. However, this causes problems at boundaries where each SPH particle is not surrounded by the same number of other particles, when compared to those particles located in the middle of the flow. In order to alleviate these differences, a density smoothing technique can be used periodically to bring the three properties back in line with one another. Colagrossi & Landrini (2003) noted that as well as restoring these properties, their *Density Re-initialisation* technique also improved the pressure distribution. Evidence of this can be seen in Figure 4.6, in which subplot (a) concerns a simulated open-channel flow without using the *Shepard Filter*, whereas subplot (b) has the technique switched on and the hydrostatic pressure distribution can clearly be seen in the flow field.

Dalrymple & Rogers (2006) suggested that the *Shepard Filter* was used to keep the free-surface clean, which was lacking in *Case 4* using the *Realistic* viscosity model. The results of *Case 6* and *Case 7* are compared with the results of *Case 4* and Boussinesq's first-order theory in Figure 4.7. These results confirm that *Cases 6 & 7* produced waves with very similar heights at the location of the wave gauge ($x = 18\text{m}$). Although both are smaller than the wave produced by *Case 4*, they have nevertheless produced a significant smoothing of the still water surface both before and after the wave event is recorded at the wave gauge. *Cases 2 & 3* (Figure 4.4) also produced smooth free-surfaces by using *Artificial* and *Improved Artificial* viscosity respectively. However, the wave height produced by *Cases 6 & 7* are both an improvement on the earlier cases (*Cases 2 & 3*) with the added benefit of producing a regular, hydrostatic pressure field similar to that shown in subplot (b) of Figure 4.6. In comparing the two density smoothing models, there is very little to choose between them. However, given that the *Shepard Filter* technique is less computationally expensive, this was chosen as the preferred approach. Despite the obvious benefits of this approach, neither model was included in the further test cases as the addition of such a technique may have an effect on other aspects of the model when used in combination and this investigation has been carried out specifically to analyse individual components of the SPH model.

Case 8 considered the adoption of a different kernel. Although the cubic kernel produces relatively quick results and is commonly used in SPH modelling, it has a number of drawbacks. Primarily, the second derivative is a piecewise-linear function. This has

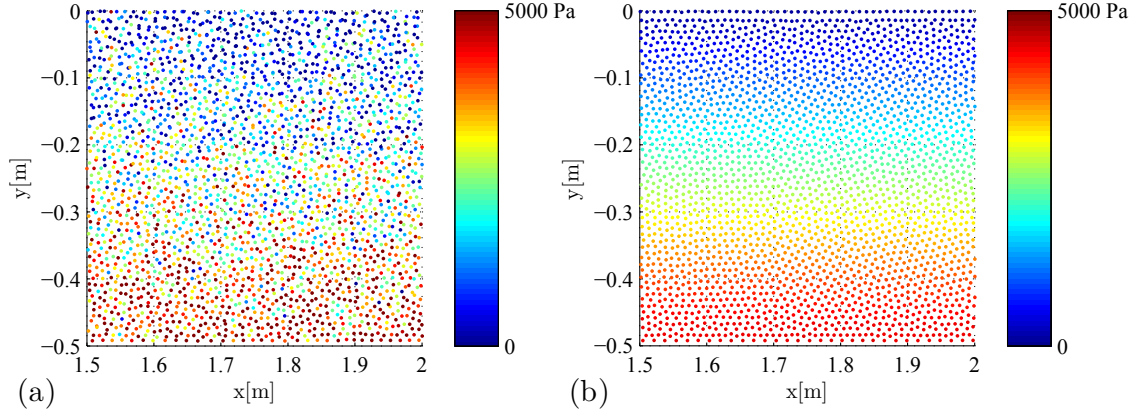


Figure 4.6: Predictions of the pressure field associated with an open channel flow: (a) no density smoothing applied and (b) Shepard filter applied, giving a hydrostatic pressure distribution.

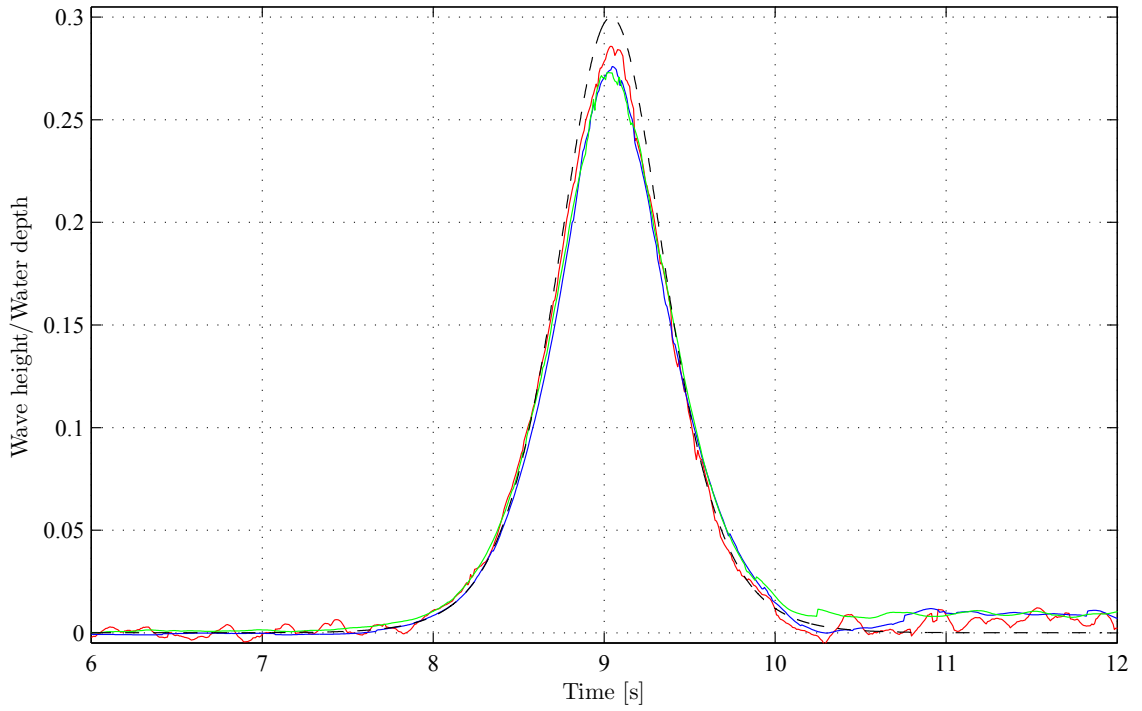


Figure 4.7: Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons between results based on different density smoothing techniques: — Case 4 (no density smoothing), — Case 6 (*Density Re-initialisation*), — Case 7 (*Shepard Filter*), - - - - Boussinesq's theory.

an effect on the stability properties of the position of the particles relative to one another, causing the well-documented problem known as “particle clumping” in which the particles tend to move around in pairs. Evidence of this effect is provided in subplot (a)

of Figure 4.8. As the order of spline-based kernels increases, they more closely represent a Gaussian function and the instabilities are reduced accordingly. However, this is paid for in terms of an increase in computational time due to the associated increase in the smoothing length. In an attempt to limit the increase in the computational effort, the present tests were carried out using a quartic spline kernel. The results of this simulation are shown on Figure 4.9, with comparisons to the results of *Case 4* and Boussinesq's first-order theory. In terms of modelling the solitary wave propagation, using the quartic kernel results in a slight increase in wave height compared to *Case 4*. More importantly, it is also preferable over the cubic kernel as it allows the particles to move relative to one another as individuals rather than in pairs. Although there are many other kernels and alternative methods that can be used in SPH to prevent particle clumping, such as the tensile instability correction by Monaghan (2000), the spline based kernels seem to provide the best results for many different flows. Also, since higher order spline kernels eliminate the clumping effect and produce more accurate results, it is preferable to use the highest order spline kernel, consistent with a reasonable computational effort.

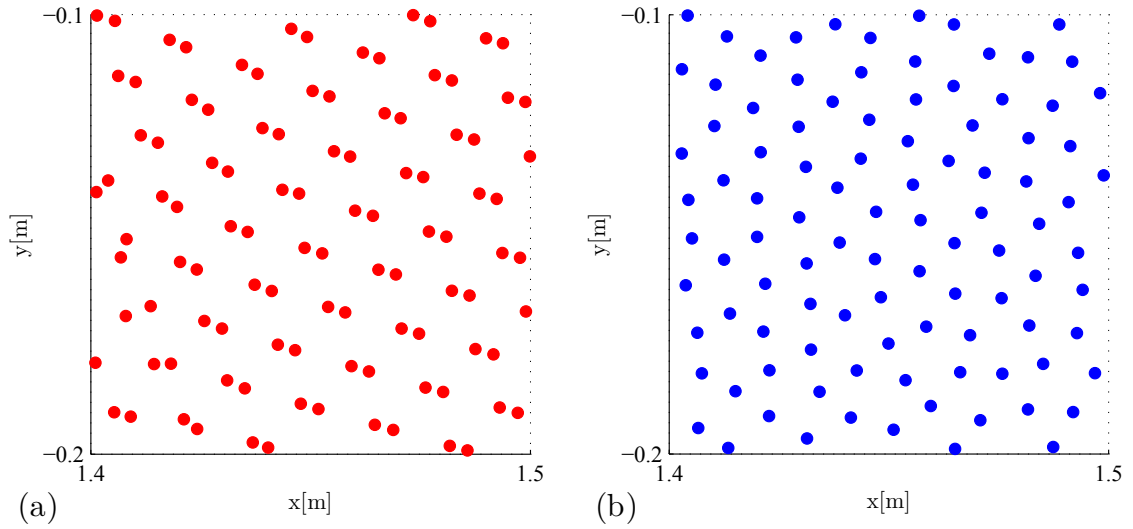


Figure 4.8: Predictions of SPH particle distribution: (a) cubic kernel, causing particle clumping, where particles move around in pairs and (b) quartic kernel, allowing particles to move around as individuals.

Although preliminary work demonstrated that the *Monaghan* time stepping scheme is both stable and relatively accurate, a comparison with a standard predictor-corrector

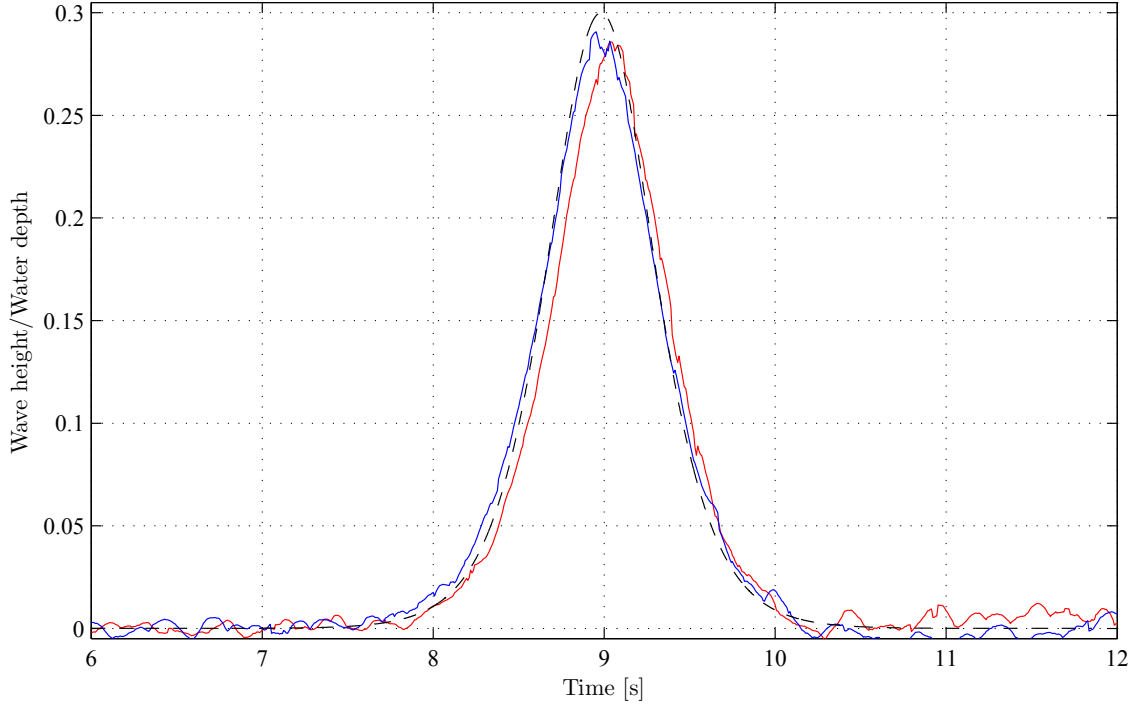


Figure 4.9: Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons between results using different kernels: — Case 4 (cubic kernel), — Case 8 (quartic kernel), - - - Boussinesq's theory.

scheme (*Case 9*) was carried out to explore the importance of the differences. In attempting to make a comparison with *Case 4*, it was found to be impossible to get a predictor-corrector scheme to run with the *Realistic* viscosity model as the scheme was not sufficiently stable to run without the added stability of *Artificial* viscosity. Therefore, the results of *Case 9*, shown in Figure 4.10, are compared with *Case 2* and Boussinesq's first-order theory. These comparisons show that the *Predictor-Corrector* scheme produces an even smaller wave than that produced by *Case 2*. As a result, the *Predictor-Corrector* scheme was rejected and is not considered further in this study. However, it would be possible to use a higher order scheme in SPH, as any stable time-stepping scheme for ordinary differential equations can be used. Unfortunately, these schemes increase the computational effort required. Furthermore, Monaghan & Kos (1999) argue that due to the stability conditions for the SPH partial differential equations, there is little benefit in using them. For these reasons, the application of a higher order scheme has not been considered further.

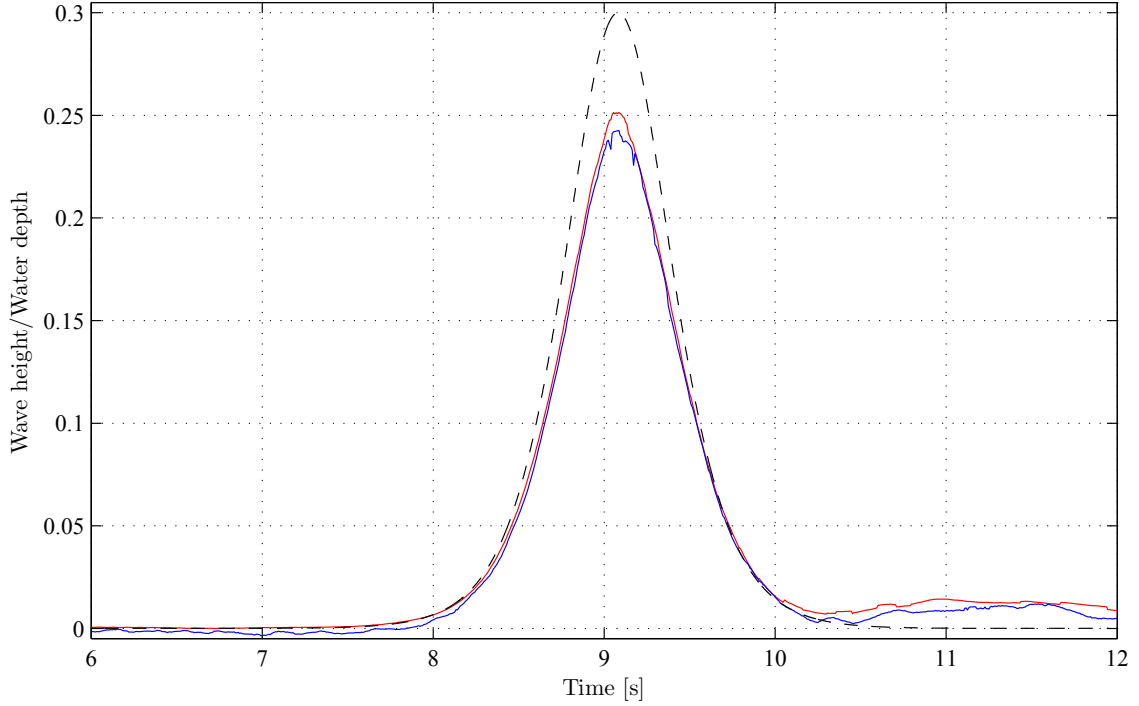


Figure 4.10: Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons between results based on different time stepping schemes: — Case 2 (*Monaghan's* scheme), — Case 9 (*Predictor-Corrector* scheme), - - - - Boussinesq's theory.

Figure 4.11 concerns the spatial variations of the wave height from each of the cases outlined in Table 4.1. It is clear from these graphs that *Cases 4, 5* and *8* are considerably better than the rest, with *Case 8* producing the smallest reduction in the wave height, H , over the length of the computational domain. As XSPH has already been ruled out, these results suggest that using *Stationary Ghosts* to represent the boundaries, *Realistic* viscosity, no density smoothing, a quartic kernel and the *Monaghan* time-stepping scheme will produce the best results. However, it has already been noted that some form of density smoothing was beneficial in producing a smooth free-surface, the importance of which outweighs the small decrease in wave-height caused by its use.

In the process of developing a numerical model, computational considerations, such as the length of time taken to conduct a simulation, must also be taken into account. Table 4.2 gives relative runtimes for each case as a proportion of the time taken for *Case 1* to run. First, it can be seen that *Case 7* was slightly faster than *Case 6* and for this reason the *Shepard Filter* was selected for density smoothing. Other than this,

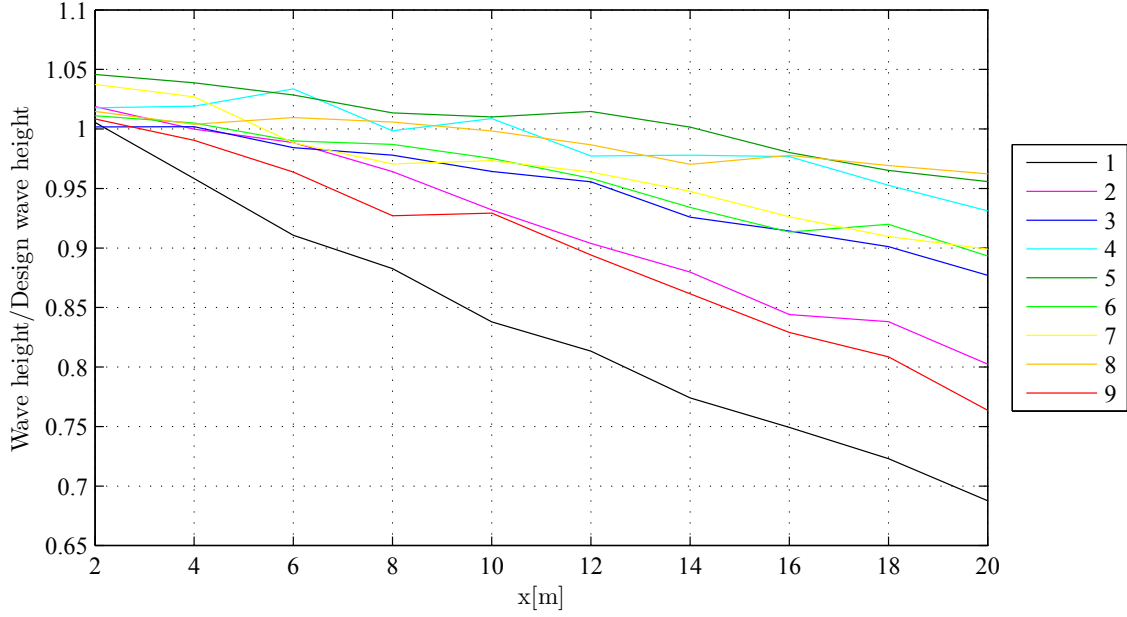


Figure 4.11: Spatial variations in the wave heights over the length of the computational domain. Note: Cases are as indicated on the legend and correspond to the details given in Table 4.1.

there was very little difference between the two density smoothing techniques considered, both in terms of the underlying equations and the results of the simulations. Second, it can be seen that although *Case 8* produced the best results, or the smallest reduction in wave height, it took 1.5 times as long as *Case 1* to run. This could become a limiting factor when performing simulations with large numbers of SPH particles. For this reason, another run with the same attributes as *Case 8* was carried out, but with half the particle to depth ratio, giving a four-fold reduction in the total number of particles. The results of this case, which shall be referred to as *Case 10*, are shown in Figure 4.12. These results confirm that the decrease in resolution has had no effect whatsoever on the predicted wave height at the downstream end of the domain, but has increased the irregularity, or bumpiness, of the free-surface. Since it is intended that the *Shepard Filter* will be used to smooth the free-surface, the additional disturbance at low resolution was thought to be fairly inconsequential. As a result, the quartic kernel was selected as the preferred approach. As a proportion of the time taken to run *Case 1*, the runtime for *Case 10* was only 0.2. This implies that such a large reduction in resolution was unnecessary and that even a small reduction would make the runtime acceptable compared with the other cases.

Case	Time taken to run 0.1s of real time (minutes:seconds)	Time taken as a proportion of <i>Case 1</i> 's runtime
1	28:10	1.00
2	20:06	0.71
3	30:07	1.07
4	25:59	0.92
5	32:56	1.17
6	27:08	0.96
7	25:03	0.89
8	42:17	1.50
9	26:32	0.94
10	5:43	0.20

Table 4.2: runtimes for all cases relative to *Case 1*. Note: Case 10 relates to an additional case with a four-fold reduction in the number of SPH particles, but in all other respects is identical to Case 8.

The results of the comparisons described above suggest that the optimum SPH model for simulating the propagation of a solitary wave and other related free-surface flows should be based on *Case 8*, but also include a *Shepard Filter*. Since the runtimes for these simulations were not considered to be unacceptably long for any of the cases, the particle resolution will be kept at its original value, with $d/\Delta x = 50$. From this point on, this model will be referred to the *optimum* model and is tested in *Case 11*. The results of these calculations are given in Figure 4.13 and compared to Boussinesq's first-order theory, which has been aligned in time with the numerical results. In undertaking these calculations, the *Shepard Filter* used in combination with the quartic kernel did not have the same detrimental effect on the wave height as it did when used in conjunction with the cubic kernel in *Case 7*. As a result, the *optimum* model produces a wave with only a 2.5% error in the wave height at the 18m wave gauge location and a smooth free-surface. The full time histories at all of the wave gauges for *Case 11* are given in Figure 4.14. If this figure is compared to the time histories achieved with the *basic* model, given in Figure 4.2, the improvements seen with the *optimum* model are more clearly defined.

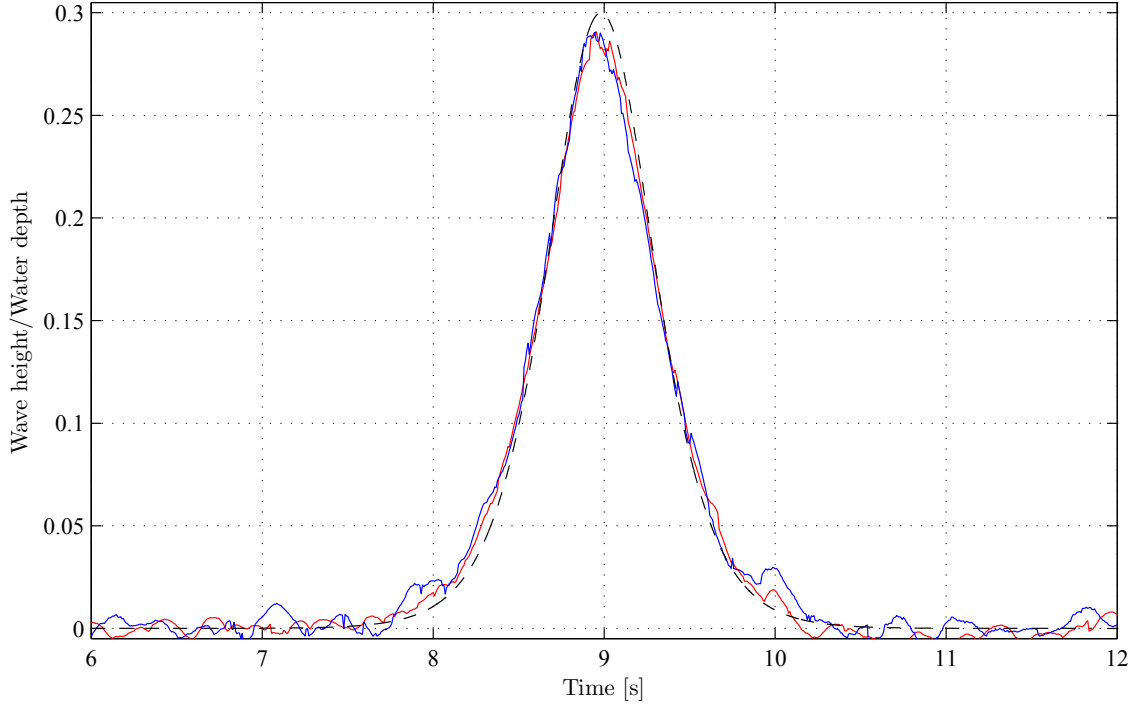


Figure 4.12: Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons between results using different SPH particle resolutions: — Case 8 (standard SPH particle resolution, used for the majority of the test cases in this study), — Case 10 (four-fold reduction in SPH particle resolution), - - - - Boussinesq's theory.

For further information concerning the results of this case study, Table 4.3 provides data relating to each of the 11 cases. Specifically, it defines the wave heights at the 18m gauge, the time taken for the crest of the wave to reach this gauge and the average wave speed between the gauges at 2m and 18m. For each quantity, percentage errors arising from direct comparison with Boussinesq's first-order theory are provided. Although the wave speed is also an important characteristic to model correctly, the precise theoretical wave speed depends on the solitary wave theory adopted (§ 4.2.1). In contrast, the theoretical wave height is unanimously constant. As a result, the wave speed was not considered the most appropriate parameter when seeking to establish the *optimum* model. However, it will be the subject of further discussions in the comparisons which follow.

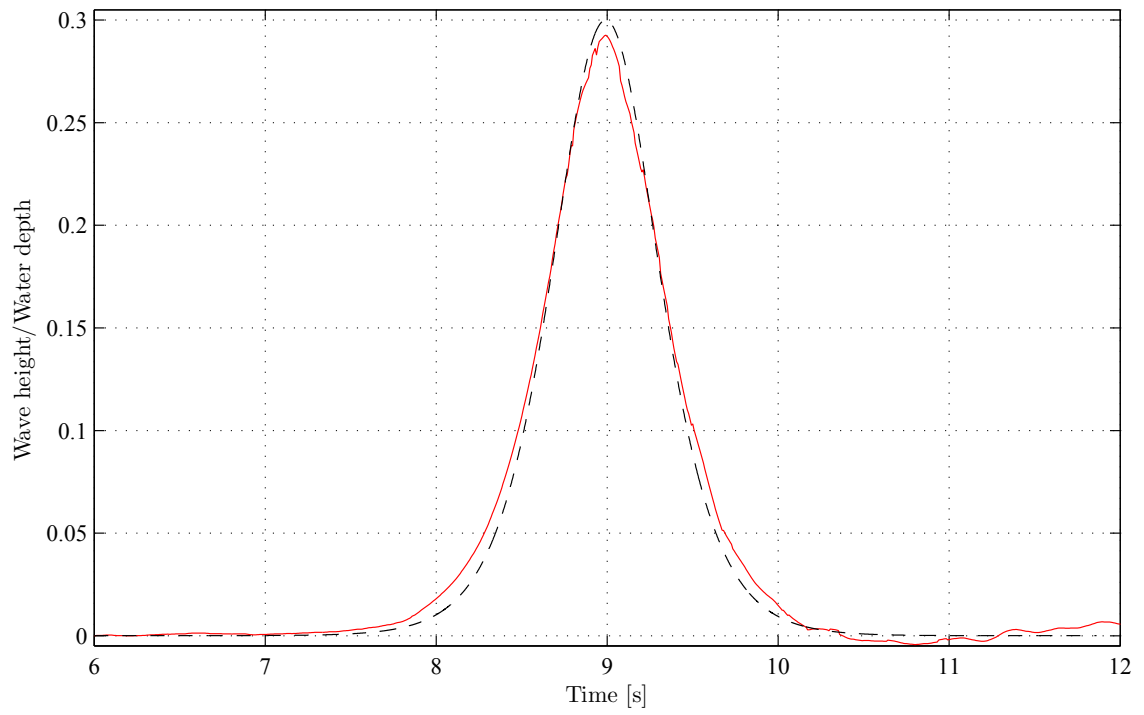


Figure 4.13: Predicted water surface elevations, $\eta(t)$, at $x = 18\text{m}$, comparisons between results produced with the *optimum* model and Boussinesq's first order solitary wave theory: — Case 11 (*optimum* model), - - - - Boussinesq's theory.

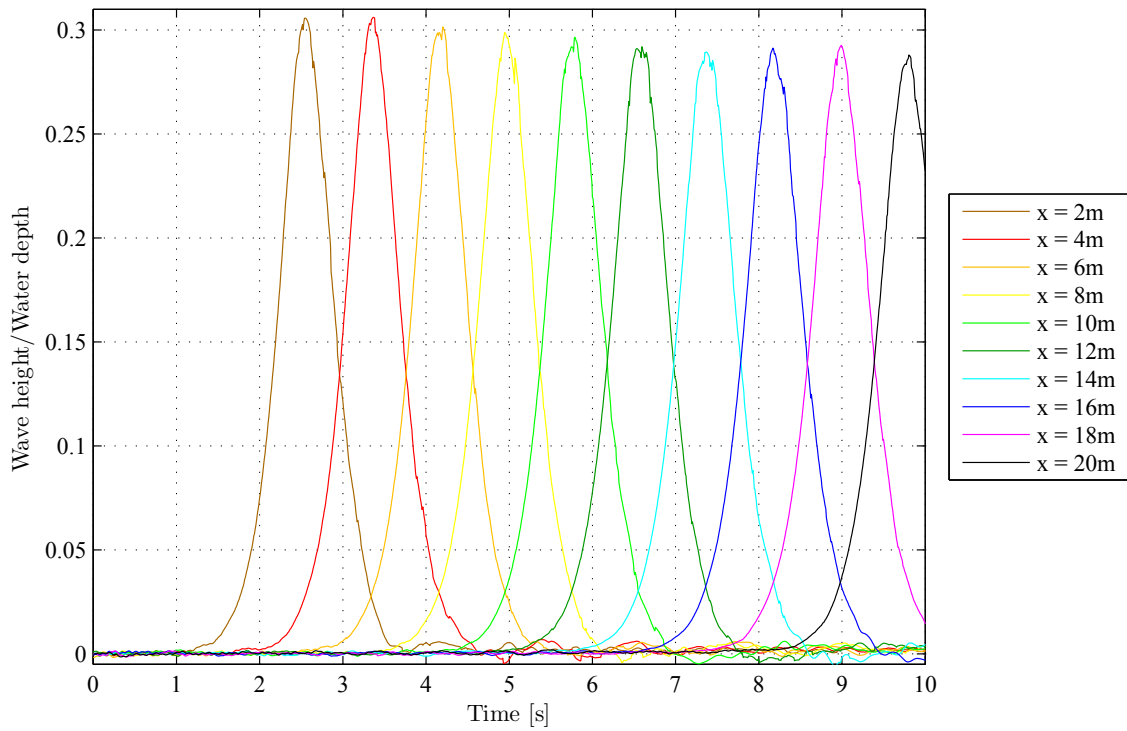


Figure 4.14: Solitary wave propagation, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of numerical wave gauges using the *optimum* SPH model. (Note: The legend defines the location of the wave gauges measured from the position of the paddle at the beginning of its motion, $x = 0$.)

Case	Wave height at $x = 18\text{m}$ (m)	% error	Time taken for wave to reach $x = 18\text{m}$ (s)	% error	Average wave speed between $x = 2\text{m}$ and $x = 18\text{m}$ (m/s)	% error
First order theory	0.150		8.891		2.525	
1	0.108	27.7	8.985	1.1	2.346	7.1
2	0.126	16.2	9.086	2.2	2.312	8.4
3	0.135	9.9	9.039	1.7	2.327	7.8
4	0.143	4.7	9.039	1.7	2.327	7.8
5	0.145	3.5	9.024	1.5	2.333	7.6
6	0.138	8.0	9.047	1.8	2.325	7.9
7	0.137	9.0	9.016	1.4	2.335	7.5
8	0.145	3.1	8.953	0.7	2.357	6.7
9	0.121	19.1	9.086	2.2	2.312	8.4
10	0.145	3.1	8.946	0.6	2.359	6.6
11	0.146	2.5	8.992	1.1	2.343	7.2

Table 4.3: Properties of the propagating wave form, a comparison between the 11 test cases and first order Boussinesq solitary wave theory. Comparisons concern the wave height at $x = 18\text{m}$ (from the paddle position), the time taken for the wave crest to reach $x = 18\text{m}$ and the average wave speed between $x = 2\text{m}$ and $x = 18\text{m}$.

4.5 Comparison with established theories

4.5.1 $H/d = 0.3$

In the previous section, the outcome of the case study resulted in an SPH model which could reasonably simulate solitary waves propagating over a distance of the order of one hundred times their wave height. In this section, the model is further compared with solitary waves based upon the first-order theories of Boussinesq, McCowan and, briefly, to the second order theory of Laitone. Once again, solitary waves with $H/d = 0.3$ are used in order to carry out the comparisons. Up to this point, comparisons have focused mainly on comparing and attempting to improve the spatial variations in the wave height arising from an SPH description of a propagating solitary wave. In order to quantify the accuracy of the SPH model further, an assessment of the wave form, the wave speed and the horizontal velocity of the water particles under the crest of the wave will also be considered.

As Boussinesq's first-order theory is the most commonly cited solitary wave theory, it was used for all the comparisons made in the previous sections. The *optimum* model arising from the comparisons is considered in more detail below. Figure 4.15 compares time histories of the water surface elevation, $\eta(t)$, predicted at four wave gauges spread along the length of the wave flume. At $x = 1\text{m}$ on Figure 4.15(a) the SPH wave profile is almost an exact replica of the theoretical curve. However, as the wave travels along the flume (in subplots (b), (c) and (d)), the small loss in wave height seems to develop alongside a small reduction the SPH predicted wave speed, the numerically simulated wave taking longer to reach each gauge than the theoretically predicted solution. Figure 4.16 concerns the spatial wave profile, $\eta(x)$, at four different times, comparisons between the SPH model predictions and Boussinesq's first-order theory again being provided. These results are consistent with those given on Figure 4.15; the SPH model predicts a small reduction in both the wave height and the phase velocity relative to the first-order Boussinesq theory.

Boussinesq's theory assumes that the horizontal water particle velocity under the wave crest is constant with depth. Comparisons with the SPH predicted particle velocities, given in Figure 4.17, show that the average horizontal particle velocities are approximately the same as the Boussinesq theory predicts. However, the SPH particle

velocities show a marked variation with depth. These results are clearly at odds with the Boussinesq first order theory. From these results it would seem that although the SPH predictions are close to the theoretically predicted solitary wave, there are small differences in both the wave form, as can be seen in Figure 4.13, and the wave speed. However, other solitary wave theories are known to give slightly different wave profiles. Therefore, before the differences in the SPH predictions are noted as errors, it is worth comparing the results to other established theories.

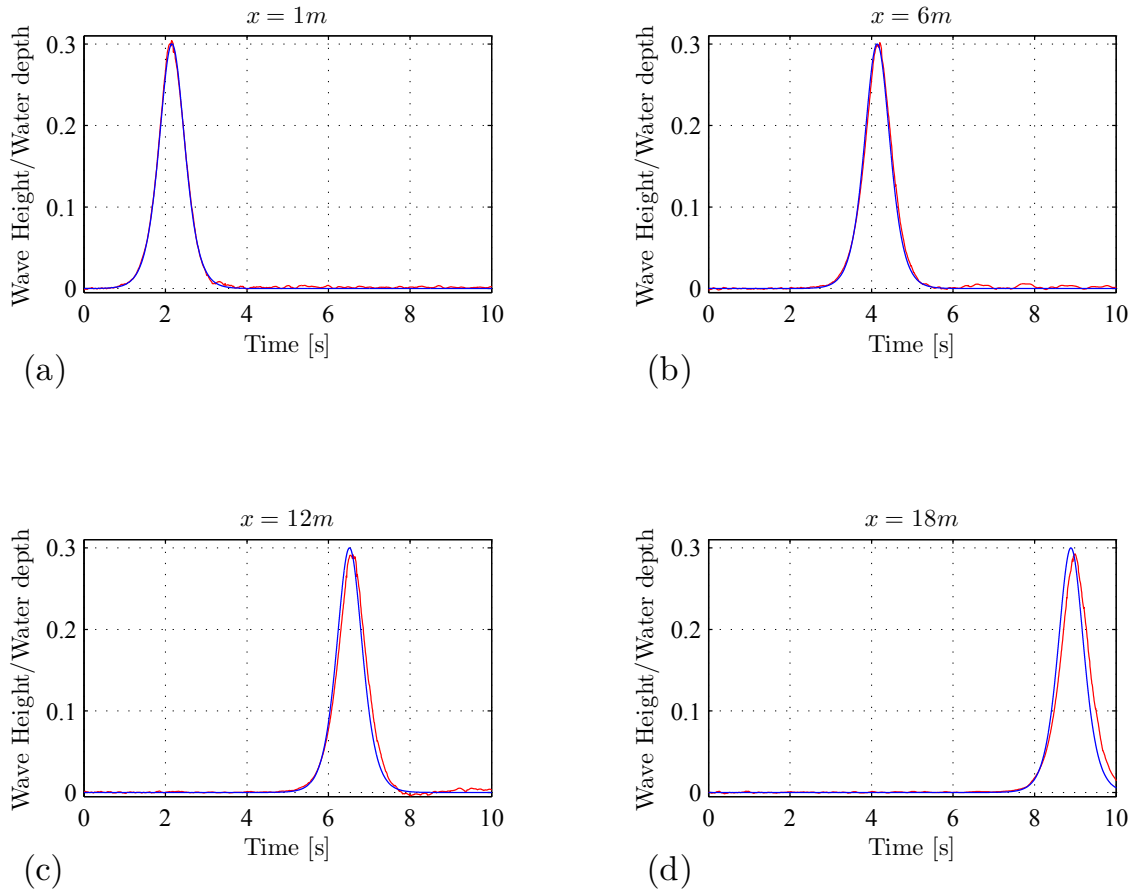


Figure 4.15: Solitary wave propagation with $H/d = 0.3$, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of four numerical wave gauges, comparisons between results produced with the *optimum* SPH model and Boussinesq's first order solitary wave theory: — SPH model, — Boussinesq's theory. Wave gauge locations are: (a) $x = 1m$, (b) $x = 6m$, (c) $x = 12m$ and (d) $x = 18m$.

Initially, the wave speed will be ignored and the results will focus on the SPH pre-

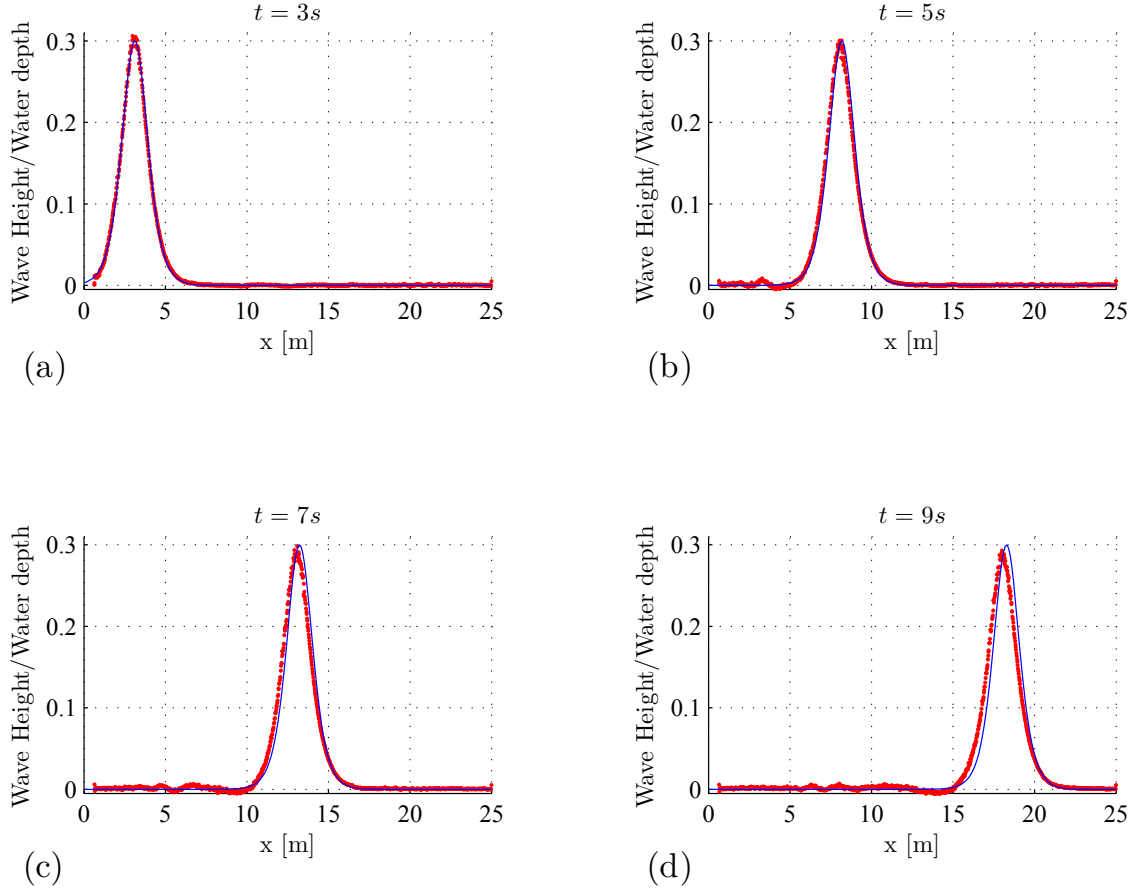


Figure 4.16: Water surface elevation, $\eta(x)$, predicted at four times during solitary wave propagation with $H/d = 0.3$, comparisons between results produced with the *optimum* SPH model and Boussinesq's first order solitary wave theory: • SPH model, — Boussinesq's theory. Times in plots are: (a) $t = 3s$, (b) $t = 5s$, (c) $t = 7s$ and (d) $t = 9s$.

dicted wave form as it passes the wave gauge located at $x = 18m$. Figure 4.18 compares the wave form predicted by the SPH model to the theories proposed by Boussinesq (1872), McCowan (1891) and Laitone (1960). Although there is a slight reduction in the wave height of the SPH solitary wave, the wave profile almost perfectly fits the theoretical wave profile given by McCowan (1891). In generating The SPH solitary wave, the wave paddle was programmed according to Boussinesq's theory, as described in § 4.2.2. However, as the solitary wave propagates away from the paddle, it is expected to tend towards the natural form of a solitary wave. The results shown in Figure 4.18 suggest that the natural form for the SPH solitary wave is very close to that given by McCowan's

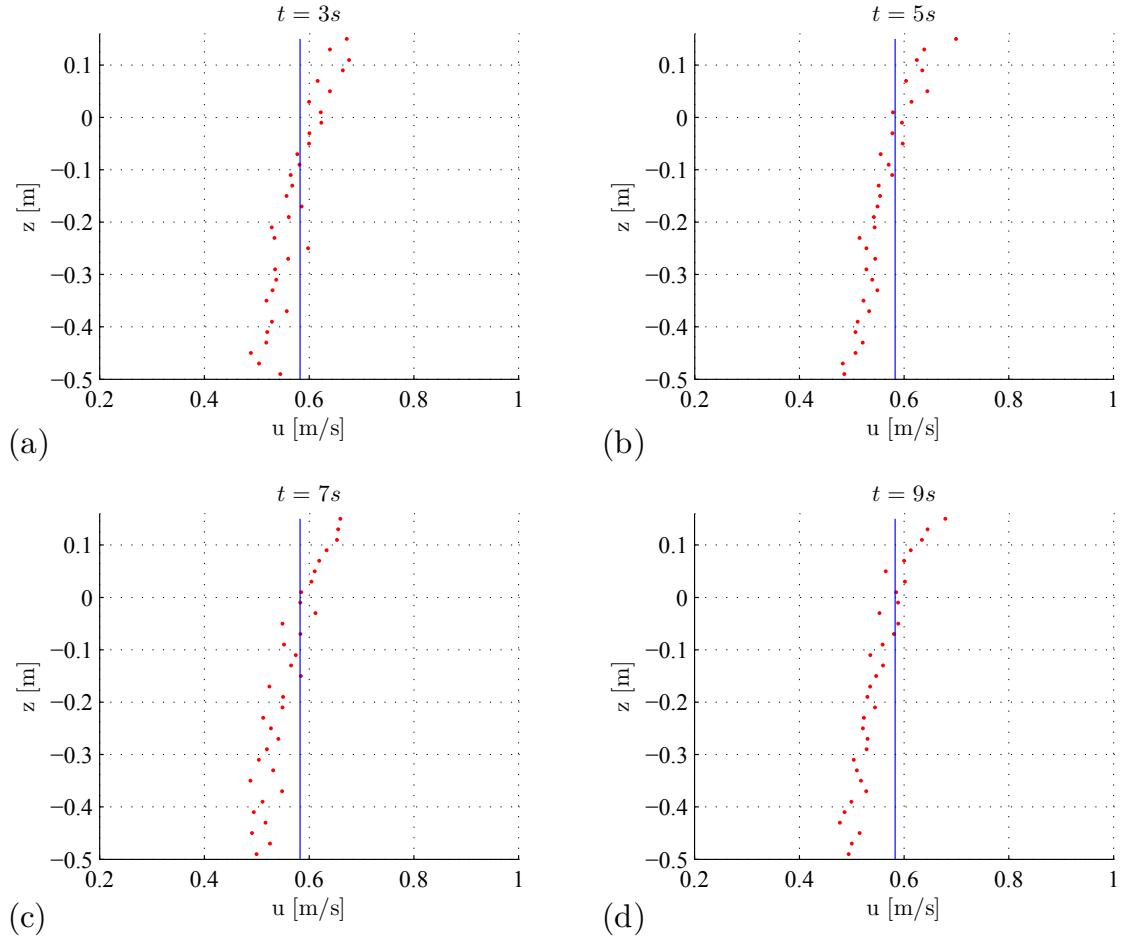


Figure 4.17: Horizontal water particle velocities, $u(z)$, under the wave crest, predicted at four times during solitary wave propagation with $H/d = 0.3$, comparisons between results produced with the *optimum* SPH model and Boussinesq's first order solitary wave theory: • SPH model, — Boussinesq's theory. Times in plots are: (a) $t = 3s$, (b) $t = 5s$, (c) $t = 7s$ and (d) $t = 9s$.

theory.

It might also be expected that as the solitary wave propagates, its form would tend towards that predicted by a higher order theory, such as Laitone, since a higher order theory is more accurate than a first-order theory. However, the comparisons provided on Figure 4.18 show that the SPH predicted wave is considerably smaller, both in terms of wave height and wave length, than the profile predicted by Laitone's theory. There is, perhaps, a fairly obvious reason for this: all of the theoretical waves shown are for the intended wave height to depth ratio of $H/d = 0.3$. As a result, the area under the graph, which is related to the total wave energy, is based upon this initial wave height. Since the

SPH calculations were initiated using Boussinesq's theory, which has a relatively small area under the graph, it is impossible for the SPH wave to tend towards the Laitone profile, as this would effectively involve an increase in the wave energy. It therefore follows that, even if the SPH wave was tending towards a more accurate higher-order form, it would be at a lower wave height, the total wave energy having been prescribed when it was generated at the wave paddle. Indeed, this provides an explanation for some of the reduction in the SPH wave height. Since the McCowan wave profile also has a slightly larger area below it when compared to the Boussinesq wave profile, this may explain why the SPH wave profile provides a better match with the McCowan profile, but with a slightly reduced wave height.

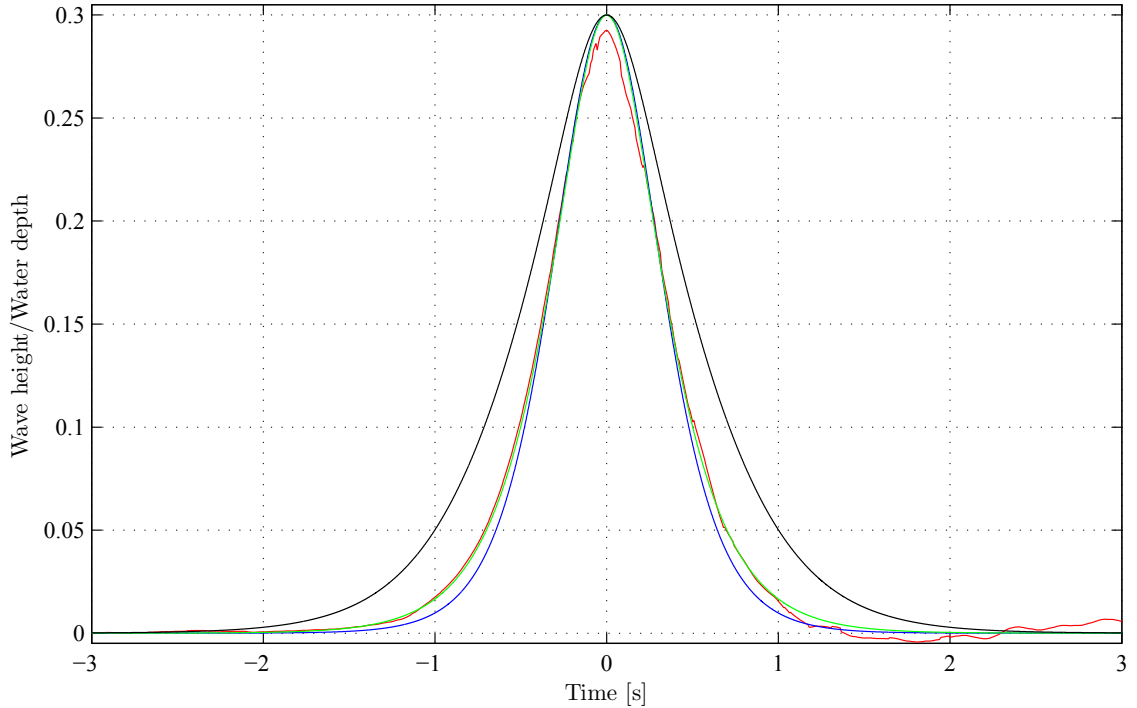


Figure 4.18: Solitary wave propagation with $H/d = 0.3$, a time-history of the water surface elevation, $\eta(t)$, predicted at $x = 18\text{m}$, comparisons between results produced with the *optimum* SPH model, Boussinesq's first order theory, McCowan's first order theory and Laitone's second order theory: — SPH model, — Boussinesq's theory, — McCowan's theory, — Laitone's theory. (Note: the time axis has been shifted to give $t = 0\text{s}$ as the wave crest passes $x = 18\text{m}$.)

In light of these results, Figures 4.19, 4.20 and Figure 4.21 compare the SPH model predictions to the theory of McCowan (1891). Figure 4.19 shows the time history of

the water surface, $\eta(t)$, at four gauges along the flume. The wave profiles match well as they propagate, with no noticeable difference in the wave speed. This is confirmed by Figure 4.20, which shows the spatial wave profiles, $\eta(x)$, at four different times. More significantly, Figure 4.21 compares the horizontal water particle velocities under the wave crest. Unlike the Boussinesq theory, McCowan's theory does not assume that the particle velocity is constant with depth. Comparisons between the latter theory and the SPH predictions are extremely good.

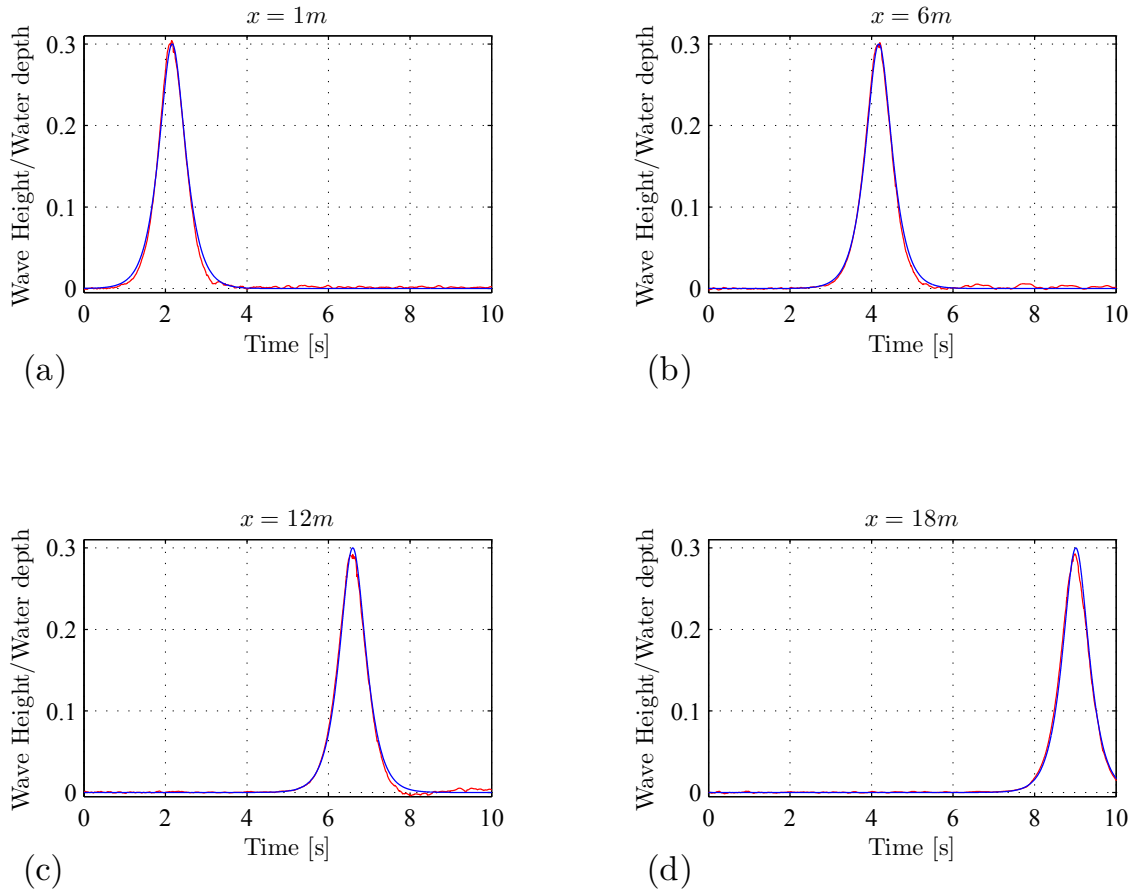


Figure 4.19: Solitary wave propagation with $H/d = 0.3$, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of four numerical wave gauges, comparisons between results produced with the *optimum* SPH model and McCowan's first order solitary wave theory: — SPH model, — McCowan's theory. Wave gauge locations are: (a) $x = 1\text{m}$, (b) $x = 6\text{m}$, (c) $x = 12\text{m}$ and (d) $x = 18\text{m}$.

Despite the success of these latter comparisons, there is one further discrepancy

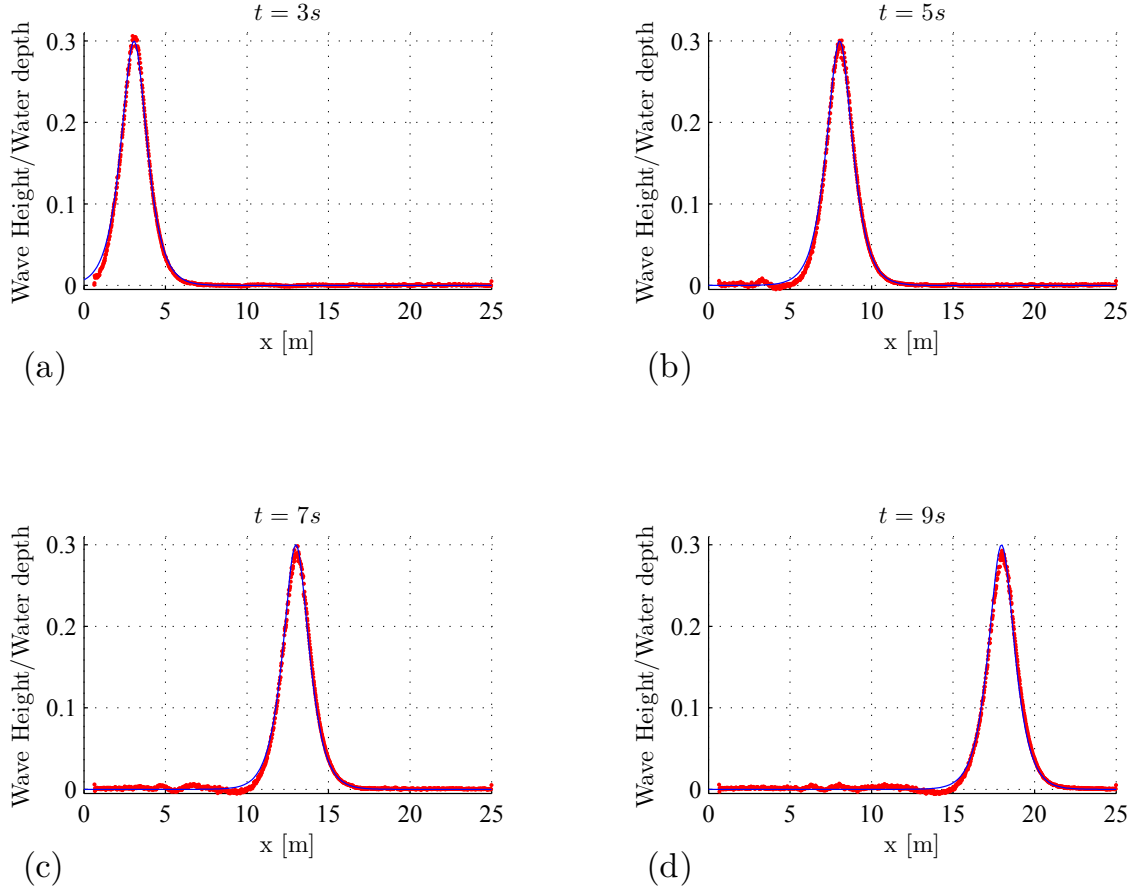


Figure 4.20: Water surface elevation, $\eta(x)$, predicted at four times during solitary wave propagation with $H/d = 0.3$, comparisons between results produced with the *optimum* SPH model and McCowan's first order solitary wave theory: • SPH model, — McCowan's theory. Times in plots are: (a) $t = 3s$, (b) $t = 5s$, (c) $t = 7s$ and (d) $t = 9s$.

between the SPH results and the theoretical wave profiles which is apparent no matter which theory is employed. After the peak of the solitary wave has passed, the SPH predicted wave profile fails to reproduce the flat water surface in the tail following the solitary wave. Indeed, small-amplitude oscillations are clearly observed in both $\eta(t)$ and $\eta(x)$. These disturbances are likely to have been caused by slight inaccuracies in the production of the solitary wave at the paddle, either due to the approximation in the length of time of the paddle stroke (Goring (1979)) as discussed in § 4.2.2 or the translation of the theoretical motion to the discrete form used in the numerical simulation.

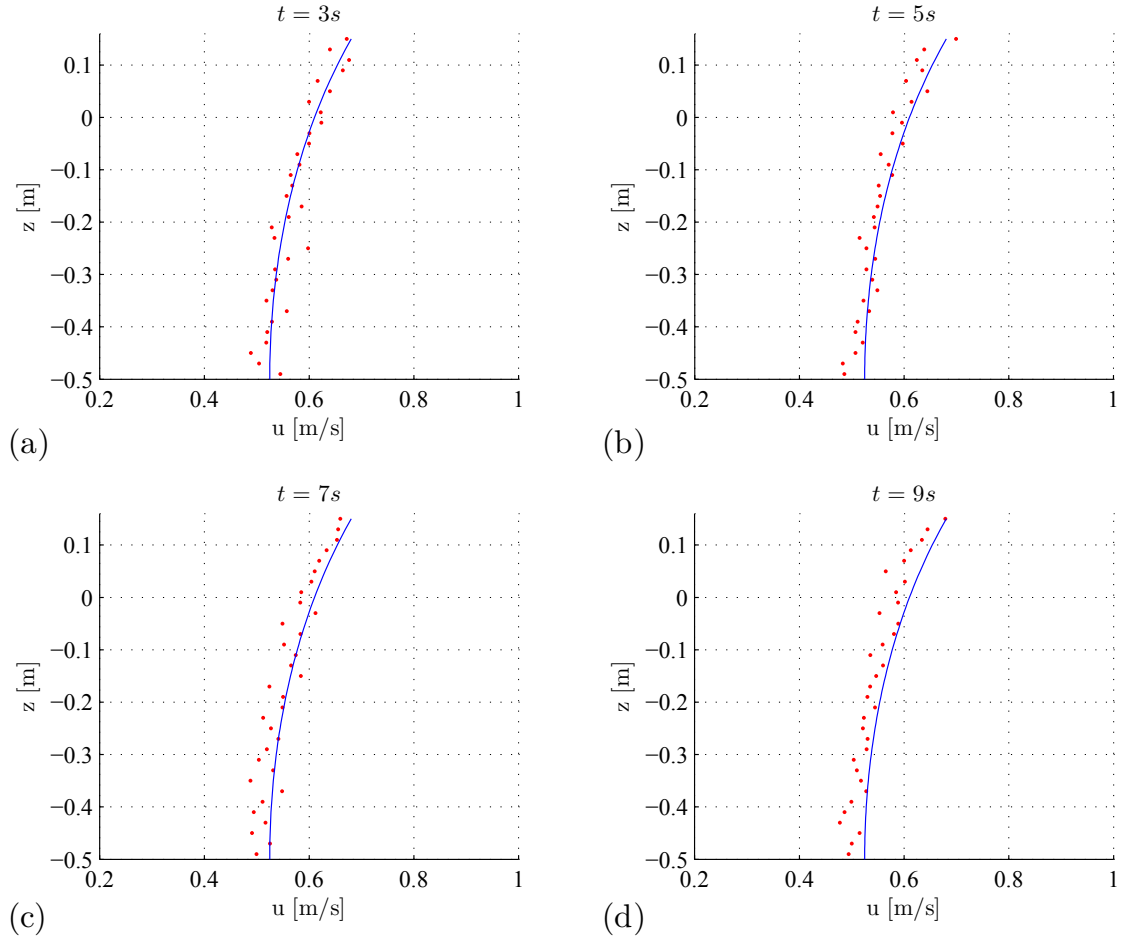


Figure 4.21: Horizontal water particle velocities, $u(z)$, under the wave crest, predicted at four times during solitary wave propagation with $H/d = 0.3$, comparisons between results produced with the *optimum* SPH model and McCowan's first order solitary wave theory: • SPH model, — McCowan's theory. Times in plots are: (a) $t = 3s$, (b) $t = 5s$, (c) $t = 7s$ and (d) $t = 9s$.

4.5.2 Varying H/d

In order to ensure that the SPH model has not been optimised solely for solitary waves with $H/d = 0.3$, further simulations were run with $H/d = 0.1, 0.2$ and 0.45 . In accordance with the results of the $H/d = 0.3$ simulations discussed in § 4.5.1, the present results were also compared to McCowan's solitary wave theory. These are presented in Figures 4.22-4.30 as time-histories of the water surface elevation, $\eta(t)$, spatial wave profiles of the water surface elevation, $\eta(x)$, and vertical profiles of the horizontal water particle velocities, $u(z)$, beneath the crest of the wave.

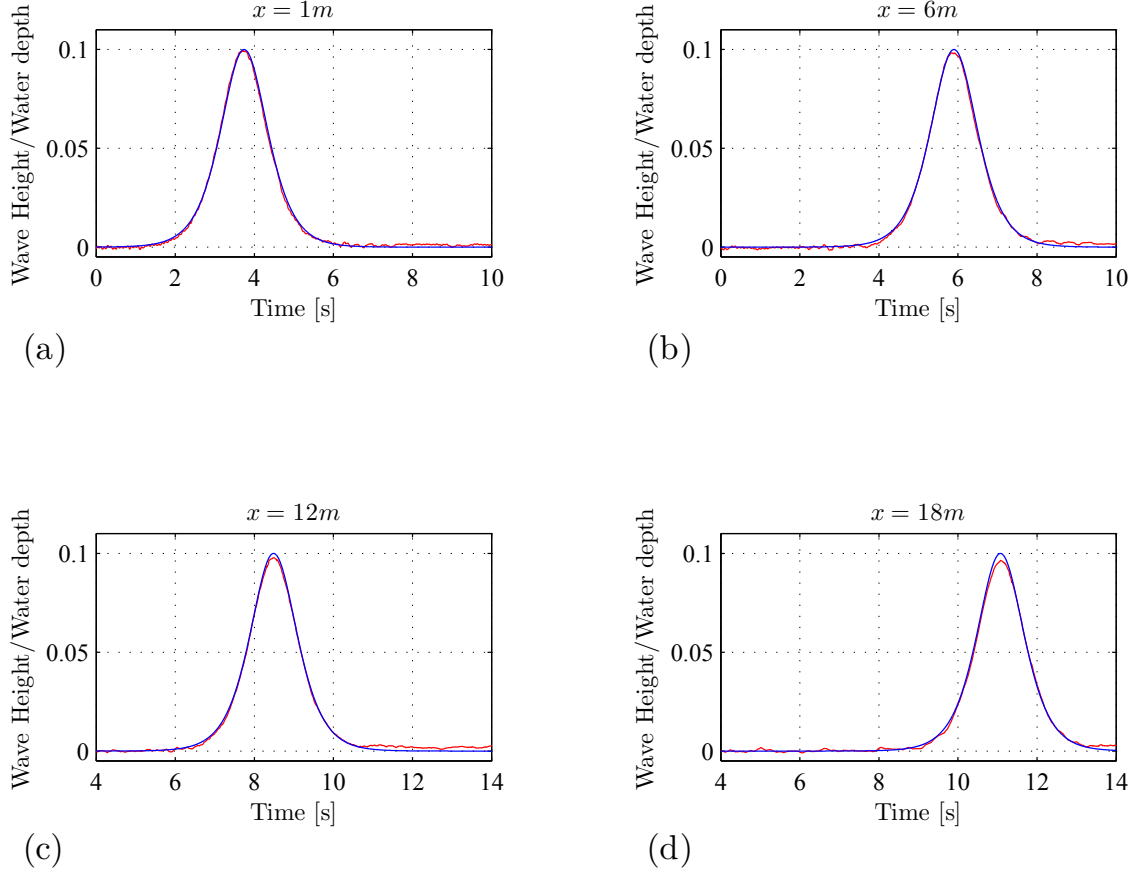


Figure 4.22: Solitary wave propagation with $H/d = 0.1$, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of four numerical wave gauges, comparisons between results produced with the *optimum* SPH model and McCowan's first order solitary wave theory: — SPH model, — McCowan's theory. Wave gauge locations are: (a) $x = 1m$, (b) $x = 6m$, (c) $x = 12m$ and (d) $x = 18m$.

The two smaller height to depth ratios, $H/d = 0.1$ and 0.2 , both provide excellent comparisons with McCowan's theory in terms of the wave form, $\eta(t)$ and $\eta(x)$, and the wave speed, and only show a slight reduction in wave height. Details of this agreement are provided on Figures 4.22, 4.23, 4.25 and 4.26.

In considering Figure 4.18 it was shown that the SPH wave with $H/d = 0.3$ matched McCowan's first order theory far closer than either Boussinesq's first order theory or Laitone's second order theory, despite the fact that the SPH wave was produced using a wave paddle programmed using Boussinesq's theory. It was also noted that the wave would be expected to tend towards the form of a wave described by a higher order theory,

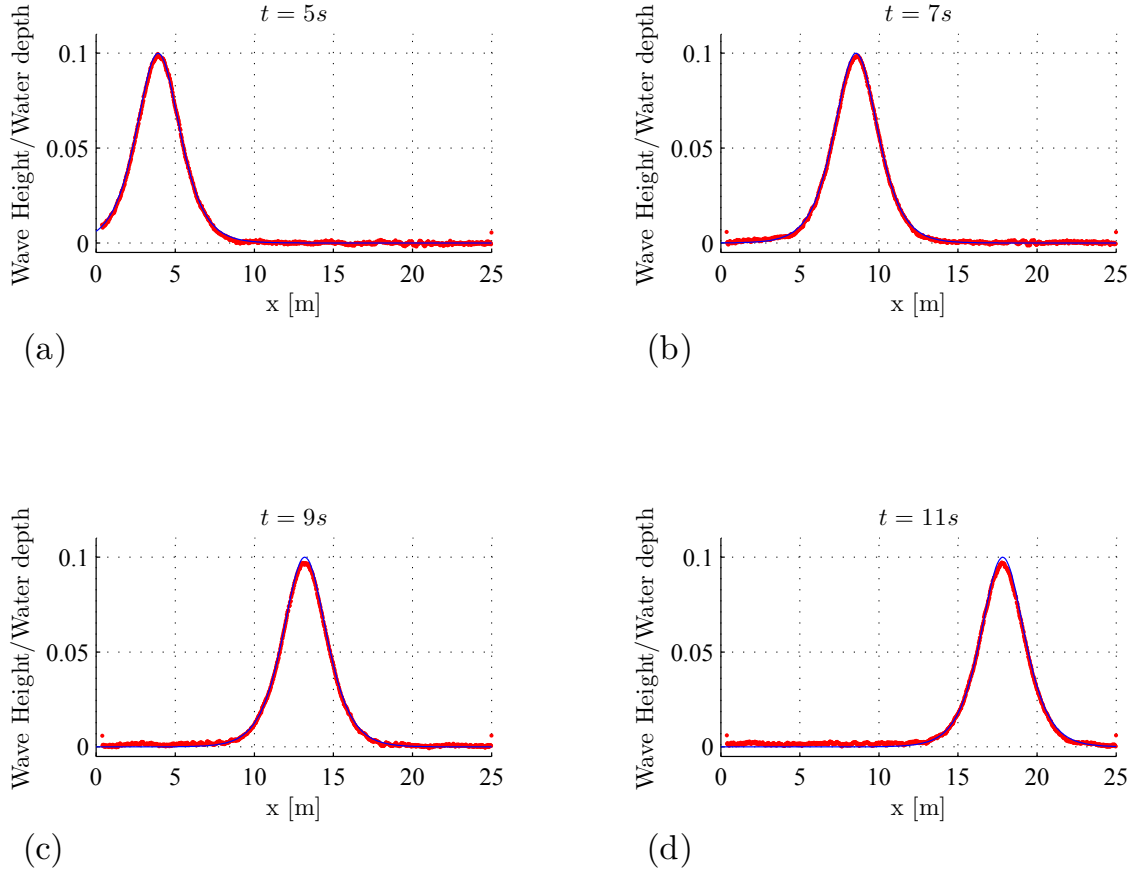


Figure 4.23: Water surface elevation, $\eta(x)$, predicted at four times during solitary wave propagation with $H/d = 0.1$, comparisons between results produced with the *optimum* SPH model and McCowan's first order solitary wave theory: • SPH model, — McCowan's theory. Times in plots are: (a) $t = 5s$, (b) $t = 7s$, (c) $t = 9s$ and (d) $t = 11s$.

but with a slight reduction in wave height; the higher-order waves contain more energy at a given wave height than those predicted by a first order theory. Figure 4.31 compares the SPH wave form for a solitary wave with $H/d = 0.1$ in subplot (a) and $H/d = 0.2$ in subplot (b) to all three theories as the waves pass the wave gauge at $x = 18m$. It can be seen that as the height to depth ratio is reduced, the two first order theories become both closer to one another and closer to Laitone's second order theory. This is due to the reduced nonlinearity of the wave events; the higher-order terms are less significant for smaller height to depth ratios. Consequently, with a reduction in the H/d ratio, the SPH model calculations more closely match all three of the solitary wave theories and show smaller wave height reductions than for the wave case with $H/d = 0.3$.

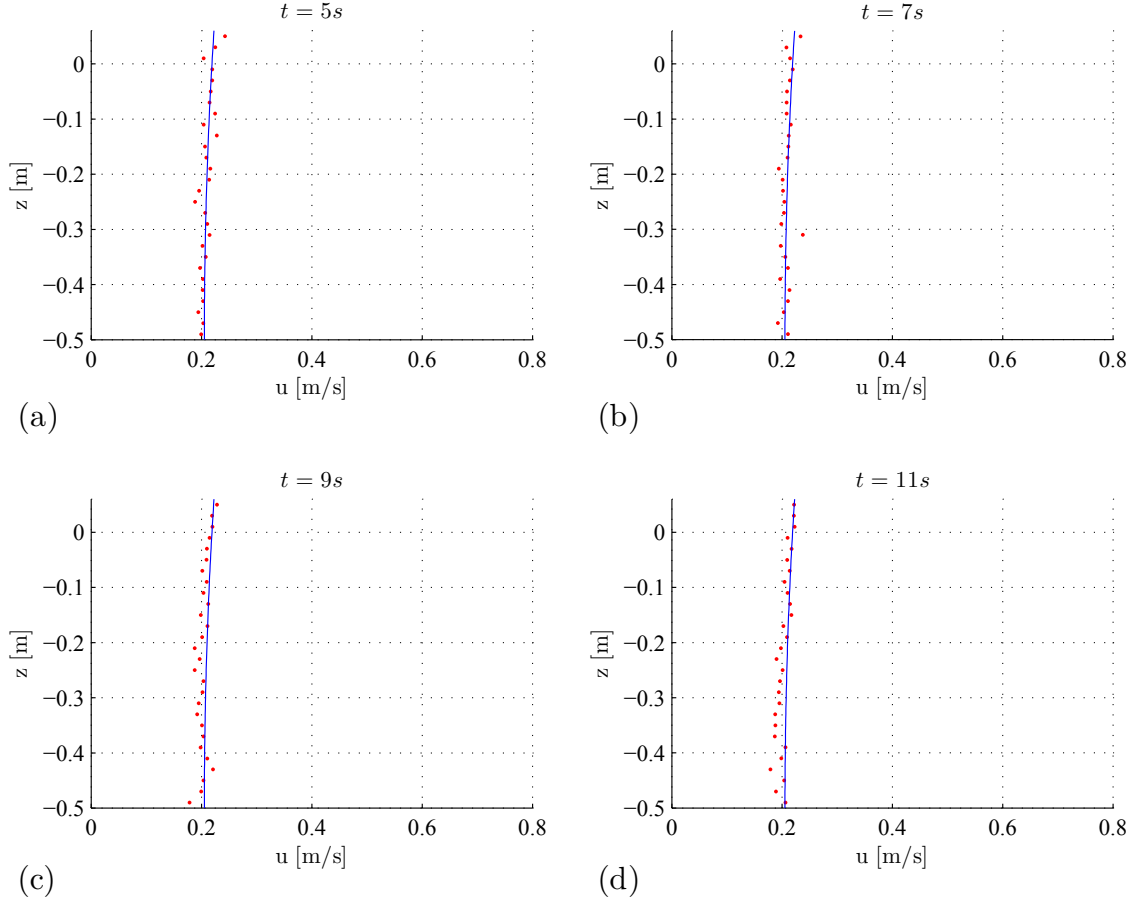


Figure 4.24: Horizontal water particle velocities, $u(z)$, under the wave crest, predicted at four times during solitary wave propagation with $H/d = 0.1$, comparisons between results produced with the *optimum* SPH model and McCowan's first order solitary wave theory: • SPH model, — McCowan's theory. Times in plots are: (a) $t = 5s$, (b) $t = 7s$, (c) $t = 9s$ and (d) $t = 11s$.

The SPH results for $H/d = 0.45$ (Figures 4.28 and 4.29) show a larger reduction in wave height when compared to the earlier comparisons relating to smaller wave height to depth ratios (Figures 4.22, 4.23 and 4.25, 4.26). However, following the arguments outlined above, it is at the larger wave height to depth ratios that non-linear effects become more important. Consequently, the SPH prediction would be expected to differ further from the first-order theories in the $H/d = 0.45$ case. Indeed, based on these arguments, it is likely that the reduced agreement between the SPH results and the first-order theory at $H/d = 0.45$ arises because the theory is inappropriate rather than the SPH results being incorrect. In Figures 4.28 and 4.29, relating to $H/d = 0.45$,

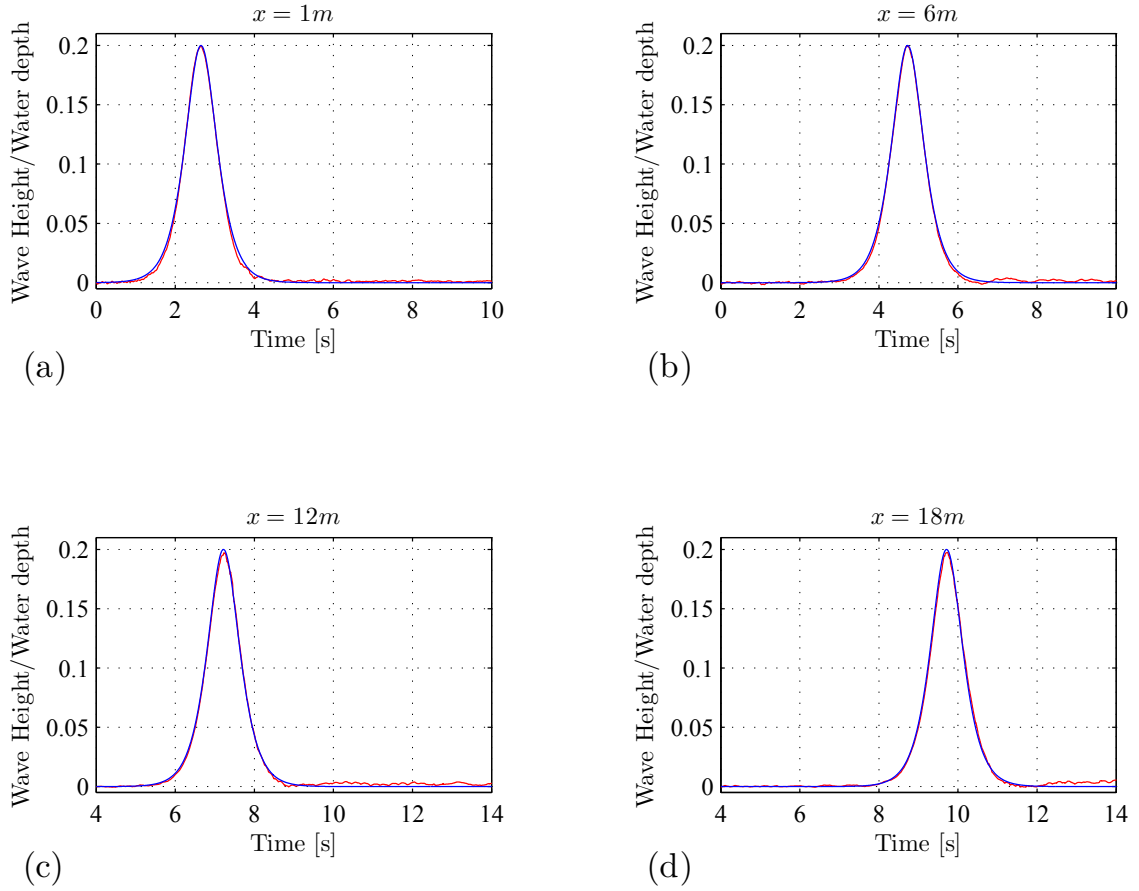


Figure 4.25: Solitary wave propagation with $H/d = 0.2$, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of four numerical wave gauges, comparisons between results produced with the *optimum* SPH model and McCowan's first order solitary wave theory: — SPH model, — McCowan's theory. Wave gauge locations are: (a) $x = 1\text{m}$, (b) $x = 6\text{m}$, (c) $x = 12\text{m}$ and (d) $x = 18\text{m}$.

the wave speed is greater for the SPH predicted wave than the linear theory predicts. Again, this is consistent with the increased importance of nonlinear effects; nonlinear waves tend to travel slightly faster than linear predictions.

Further evidence of the increased importance of nonlinear effects is seen in the vertical structure of the wave induced water particle kinematics, $u(z)$, on Figures 4.24, 4.27 and 4.30; the non-uniform nature of the kinematics becomes more pronounced with increases in the H/d ratio. However, the present comparisons show that this is well modelled in the SPH predictions.

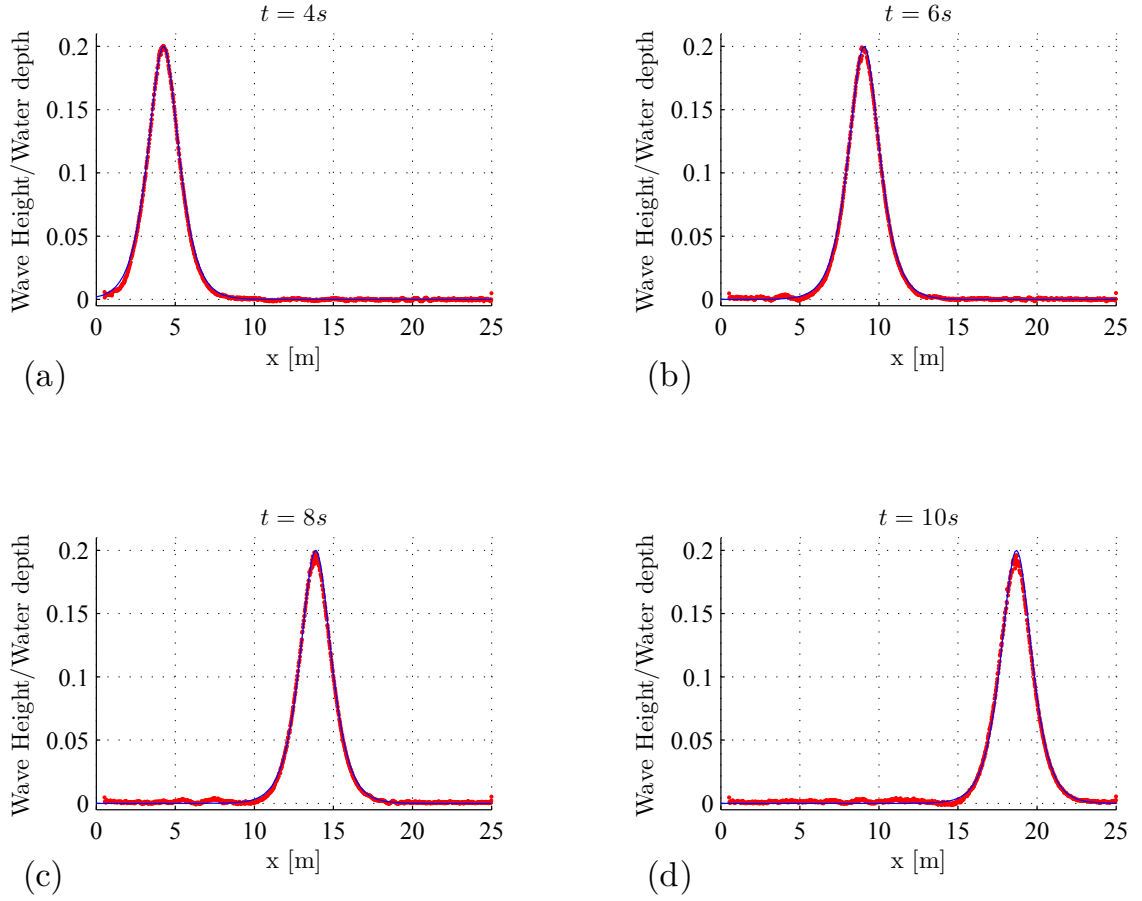


Figure 4.26: Water surface elevation, $\eta(x)$, predicted at four times during solitary wave propagation with $H/d = 0.2$, comparisons between results produced with the *optimum* SPH model and McCowan's first order solitary wave theory: • SPH model, — McCowan's theory. Times in plots are: (a) $t = 4s$, (b) $t = 6s$, (c) $t = 8s$ and (d) $t = 10s$.

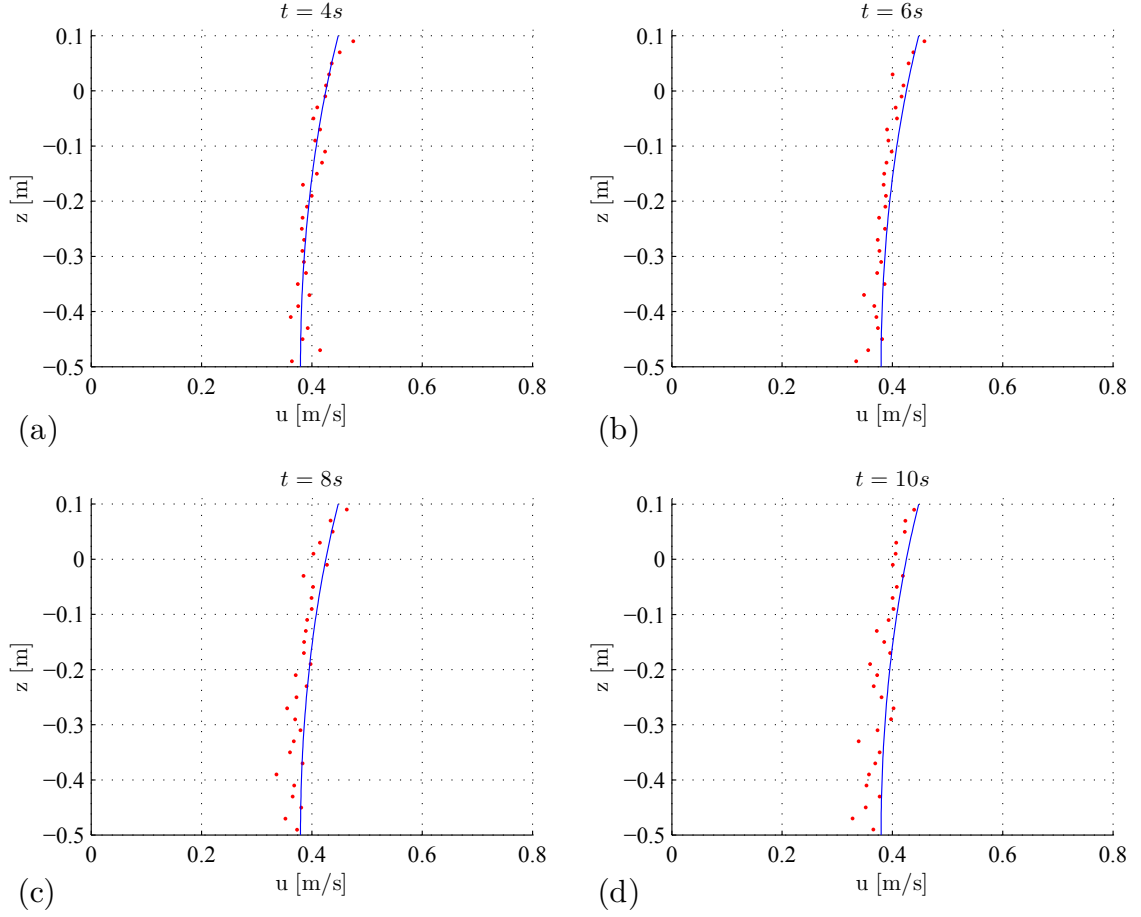


Figure 4.27: Horizontal water particle velocities, $u(z)$, under the wave crest, predicted at four times during solitary wave propagation with $H/d = 0.2$, comparisons between results produced with the *optimum* SPH model and McCowan's first order solitary wave theory: • SPH model, — McCowan's theory. Times in plots are: (a) $t = 4s$, (b) $t = 6s$, (c) $t = 8s$ and (d) $t = 10s$.

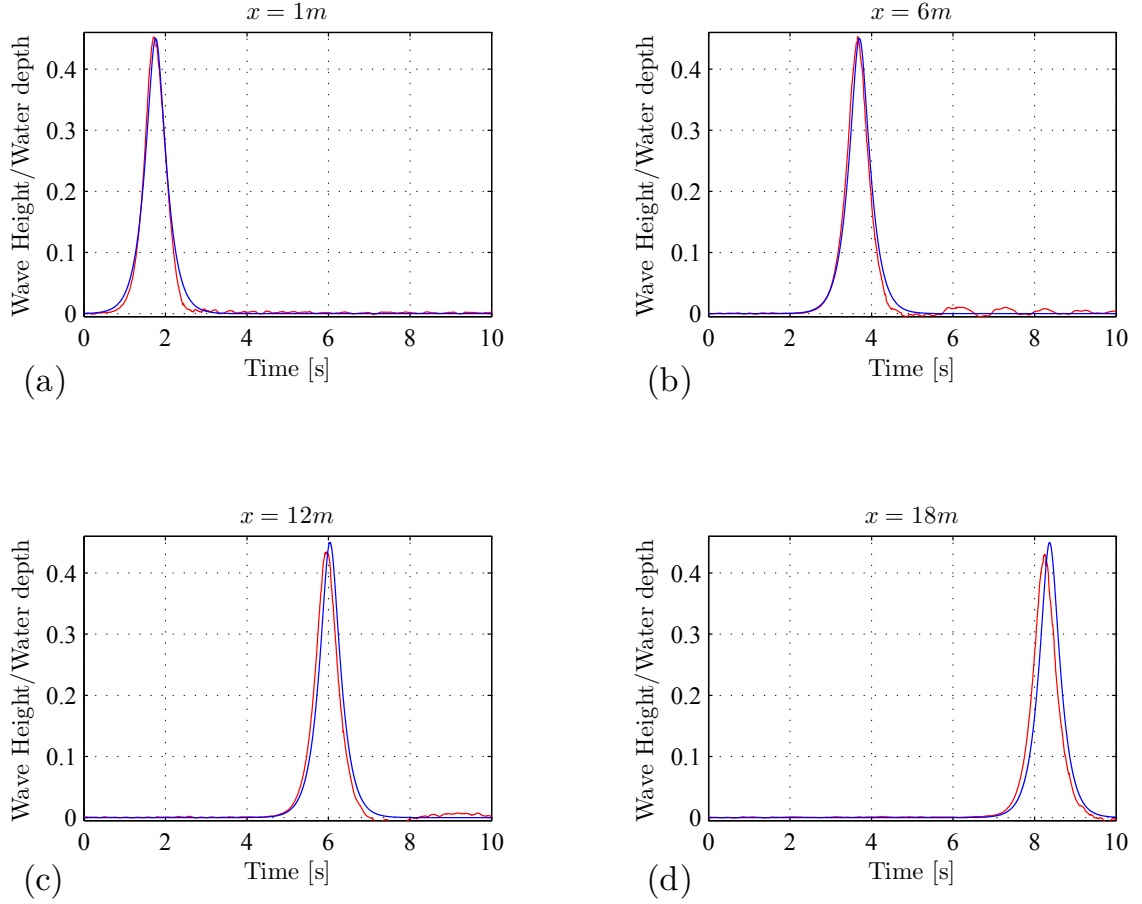


Figure 4.28: Solitary wave propagation with $H/d = 0.45$, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of four numerical wave gauges, comparisons between results produced with the *optimum* SPH model and McCowan's first order solitary wave theory: — SPH model, — McCowan's theory. Wave gauge locations are: (a) $x = 1m$, (b) $x = 6m$, (c) $x = 12m$ and (d) $x = 18m$.

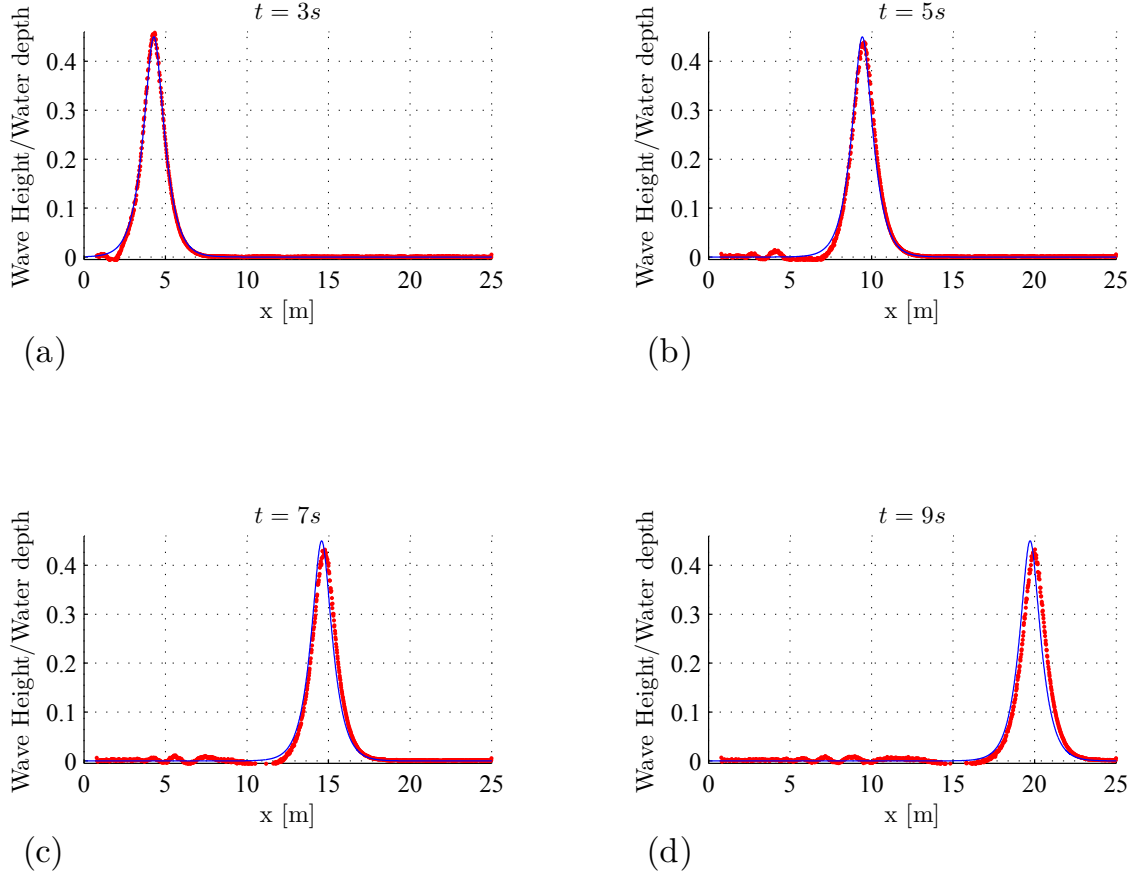


Figure 4.29: Water surface elevation, $\eta(x)$, predicted at four times during solitary wave propagation with $H/d = 0.45$, comparisons between results produced with the *optimum* SPH model and McCowan's first order solitary wave theory: • SPH model, — McCowan's theory. Times in plots are: (a) $t = 3s$, (b) $t = 5s$, (c) $t = 7s$ and (d) $t = 9s$.

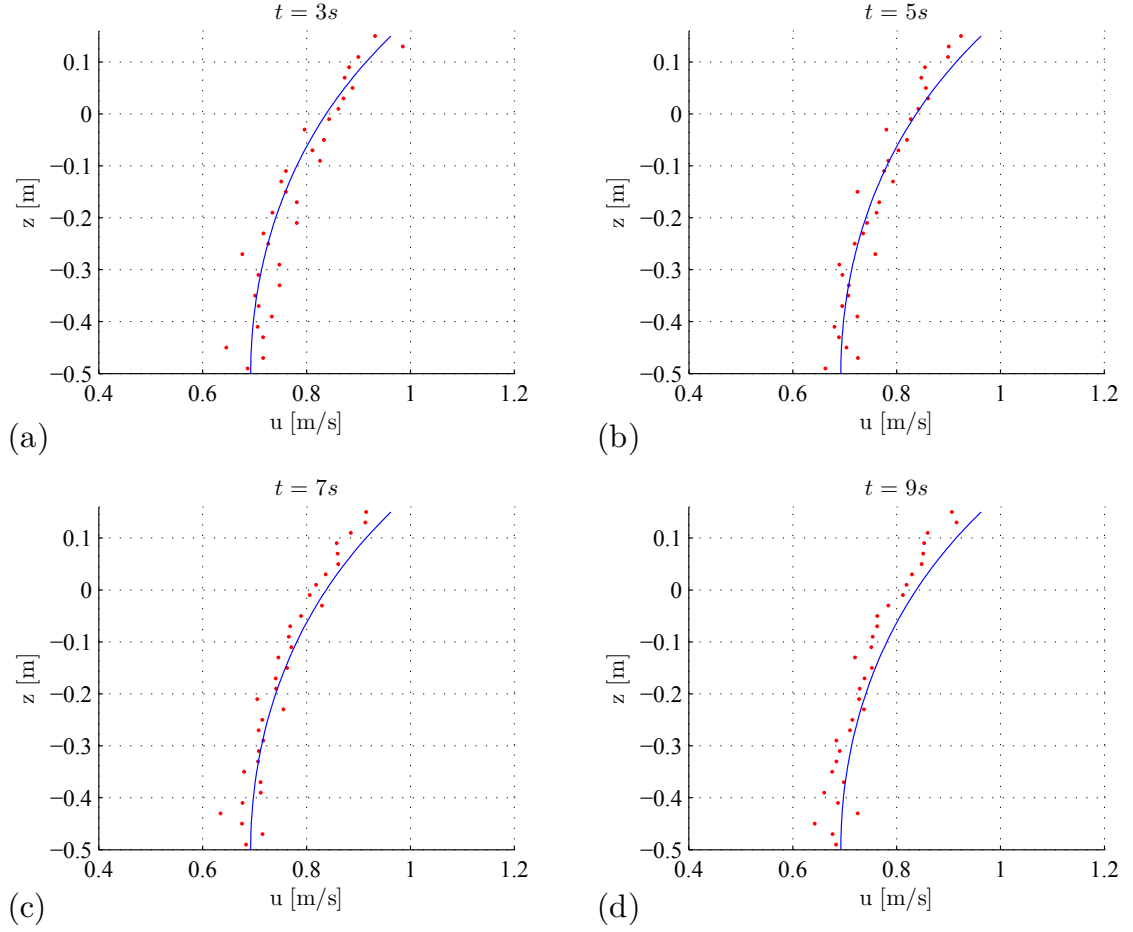


Figure 4.30: Horizontal water particle velocities, $u(z)$, under the wave crest, predicted at four times during solitary wave propagation with $H/d = 0.45$, comparisons between results produced with the *optimum* SPH model and McCowan's first order solitary wave theory: • SPH model, — McCowan's theory. Times in plots are: (a) $t = 3s$, (b) $t = 5s$, (c) $t = 7s$ and (d) $t = 9s$.

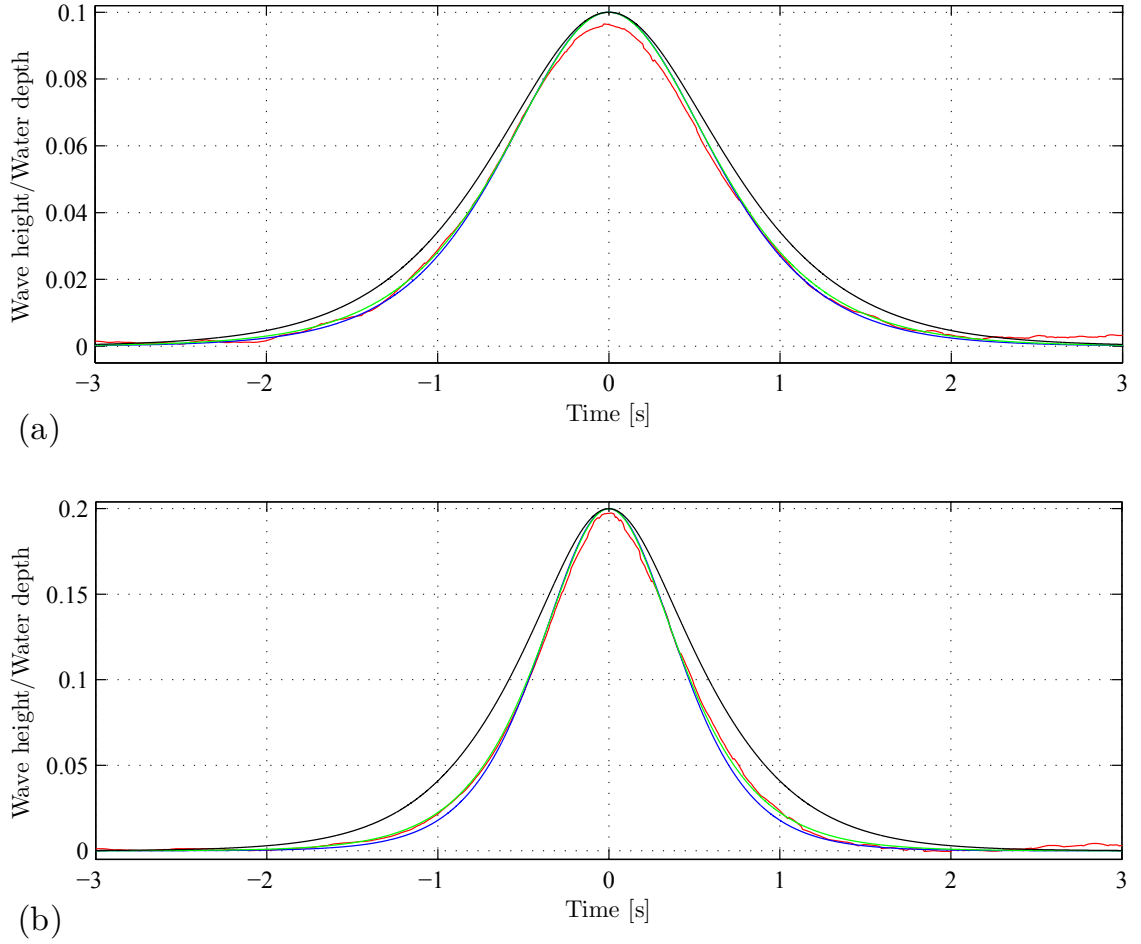


Figure 4.31: Solitary wave propagation, a time-history of the water surface elevation, $\eta(t)$, predicted at $x = 18\text{m}$, comparisons between results produced with the *optimum* SPH model, Boussinesq's first order theory, McCowan's first order theory and Laitone's second order theory: — SPH model, — Boussinesq's theory, — McCowan's theory, — Laitone's theory. (a) $H/d = 0.1$ and (b) $H/d = 0.2$ (Note: in both cases the time axis has been shifted to give $t = 0\text{s}$ as the wave crest passes $x = 18\text{m}$.)

4.6 Comparison with experimental observations

In order to further validate the chosen SPH model, quantitative comparisons were also made with laboratory observations, relating to the case with $H/d = 0.3$. The experimental study was undertaken in the glass-walled coastal flume in the Hydrodynamics Research Laboratory of the Department of Civil and Environmental Engineering at Imperial College London. This flume is 25m long, 0.5m wide and has a maximum depth of 1m. Two surface-piercing, resistance wave gauges were used to record the water surface elevations, $\eta(t)$, at 6m and 11m downstream from the centre of the paddle motion. The paddle was programmed using the same theory as was employed for programming the numerical paddle in the SPH simulations. Further details of the experimental apparatus and the paddle motion are given in Chapter 5 of this thesis.

Figure 4.32 concerns time-histories of the water surface elevation, $\eta(t)$, recorded at the two wave gauges and provides comparisons with the SPH predictions. In considering these comparisons, it is important to note that making good quantitative comparisons of the wave speed and the wave height with the SPH predicted wave is far from trivial, not least because of the nature of the experimental paddle motion. Nevertheless, quantitative comparisons have been achieved and the SPH predictions have been shown to be in good agreement with the laboratory observations. In considering the detail of these comparisons it is clear that there is some reduction in the experimentally recorded wave height between the two wave gauges, just as there is numerically. Indeed, this reduction is larger in the experimentally measured waves. This is to be expected due to the additional frictional losses resulting from the sides of the wave flume, which are not present in the two-dimensional numerical setup. Furthermore, oscillations in the tail of the solitary wave are also apparent in the experimental data, as they are in the SPH predictions. Once again, this almost certainly arises due to inaccuracies in the wave generation procedure.

Figure 4.33 provides a comparison of the SPH and experimental wave profiles at the 11m wave gauge with the theories of Boussinesq and McCowan. In making these comparisons, it should be noted that the experiment was designed to produce a solitary wave with the intended wave height at this specific location, whereas the SPH model produced the intended wave height upon leaving the paddle. As a result, the small

difference between the wave heights recorded and predicted at this gauge location should not be the main focus of a comparison. It can, however, be said that the numerical and experimental wave profiles are very similar, both shown to be in good agreement with the McCowan theory, particularly either side of the wave crest.

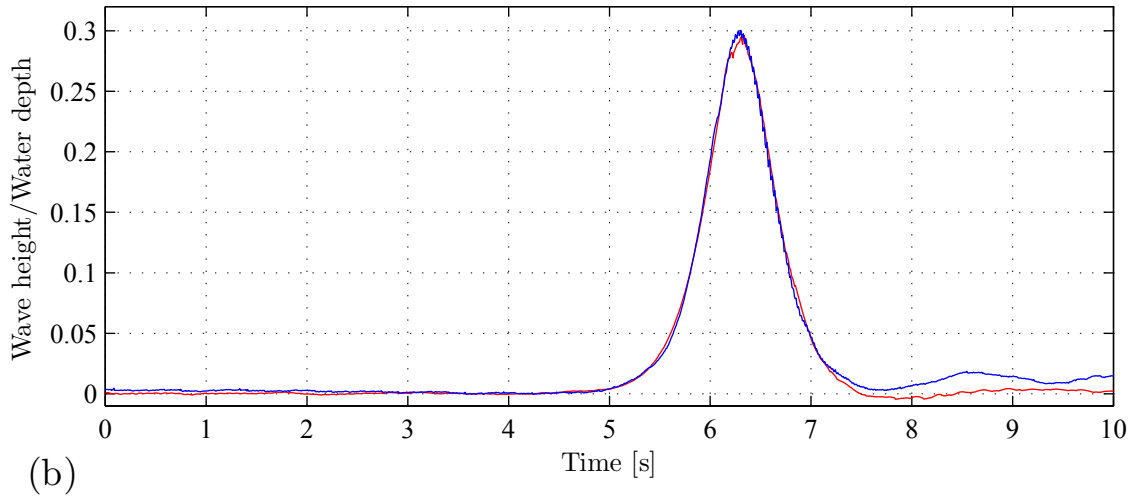
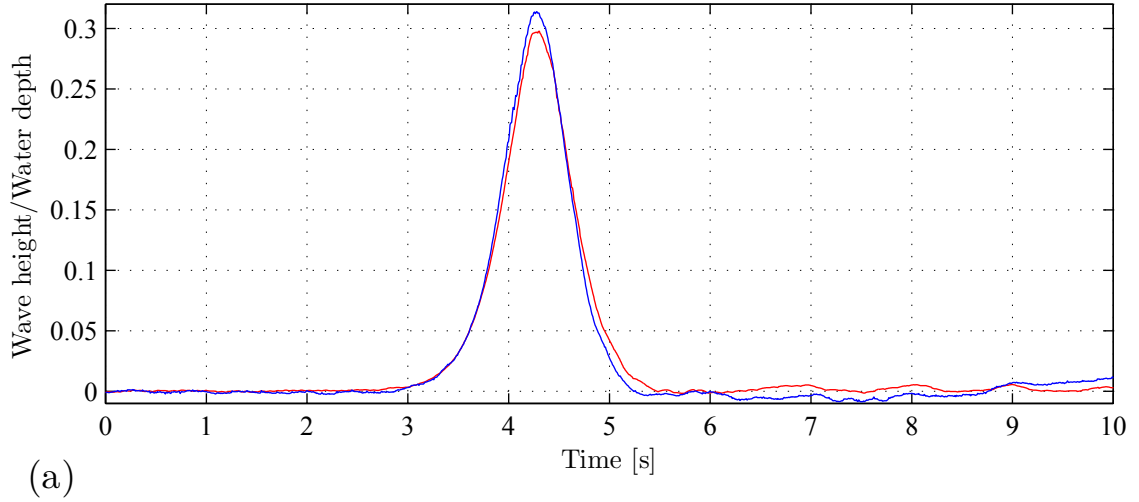


Figure 4.32: Solitary wave propagation with $H/d = 0.3$, time histories of the water surface elevation, $\eta(t)$, predicted at two spatial locations, comparisons between results produced with the *optimum* SPH model and laboratory observations: — SPH model, — laboratory observations. Wave gauge locations are: (a) $x = 6\text{m}$ and (b) $x = 11\text{m}$.

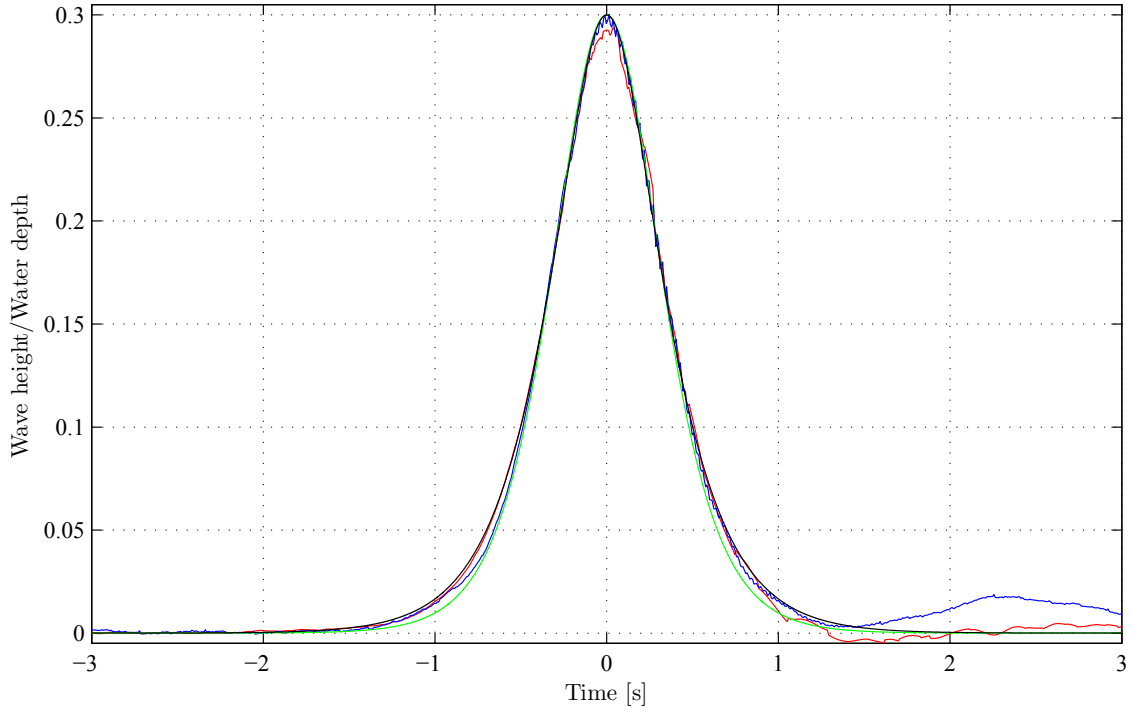


Figure 4.33: Solitary wave propagation with $H/d = 0.3$, time-history of the water surface elevation, $\eta(t)$, predicted at $x = 11\text{m}$, comparisons between results produced with the *optimum* SPH model, laboratory observations, Boussinesq's first order theory and McCowan's first order theory: — SPH model, — laboratory observations, — Boussinesq's theory, — McCowan's theory. (Note: the time axis has been shifted to give $t = 0\text{s}$ as the wave crest passes $x = 11\text{m}$.)

4.7 Effects of bed shear on the dissipation of propagating solitary waves

During the propagation of solitary waves over long distances some reduction in the wave height, H , is to be expected. As the wave height decreases, a slight reduction in the wave speed, c , is also expected since the wave height is used to calculate the theoretical wave speed (Equations (4.4) and (4.7)). These effects occur as the wave's energy dissipates due to shear stresses arising in the boundary layers adjacent to the bed and the side walls of the flume. In the comparisons described in the previous sections of this chapter, reductions in the wave height were observed in both the SPH simulations and the laboratory experiments. However, the results were only compared to potential theories. These do not take into consideration the effects of the boundary layer and therefore predict that solitary waves can propagate over infinite distances without loss of form. Although the expected reduction in wave height has been discussed, it remains to be seen whether this is entirely due to physical effects or whether numerical errors or experimental inaccuracies have also contributed.

Keulagan (1948) developed a theory to describe the attenuation rate of solitary waves by considering the loss of energy in the wave due to the dissipation arising in the laminar boundary layer. Comparisons with the solitary wave experiments of Scott Russell (1838) were then used to verify the accuracy of the theory. In these experiments, solitary waves were setup in a 20ft channel where they were allowed to propagate backwards and forwards, reflecting off the end walls and covering a total distance of up to 1200ft (366m). The development of the theory resulted in the inverse fourth-power decay law given by

$$\left(\frac{H_1}{d}\right)^{-1/4} - \left(\frac{H_0}{d}\right)^{-1/4} = K \frac{s}{d} \quad (4.25)$$

where H_0 is the original wave height, H_1 is the wave height after the wave has travelled a distance s along the channel, and K is given by

$$K = \frac{1}{12} \left(1 + \frac{2d}{B}\right) \sqrt{\frac{\nu}{g^{1/2}d^{3/2}}} \quad (4.26)$$

where B is the width of the channel, d the water depth and ν the kinematic viscosity.

Both the SPH simulations and the experimental results presented earlier in this chapter can be quantitatively compared to this theory, thereby allowing a more meaningful comparison of the reduction in the wave height. In making comparisons with the two-dimensional SPH simulations, the channel width, B , is taken to be infinite, reducing Equation (4.26) to

$$K = \frac{1}{12} \sqrt{\frac{\nu}{g^{1/2} d^{3/2}}}. \quad (4.27)$$

In order to investigate the role of the bed shear stresses two sets of calculations have been undertaken. First, comparisons have been made between both the SPH simulations and the laboratory observations and the decay law proposed by Keulagan (1948). Second, additional SPH calculations have been made with a free-slip condition imposed at the bed. The results of these calculations are discussed as follows.

Initially, the SPH results from the simulation with the *optimum* model with $H/d = 0.3$ and $d = 0.5\text{m}$ are compared to the decay theory by considering the attenuation rate of the wave height and wave speed at numerical wave gauges located at 2m intervals along the length of the flume. The theoretical attenuation rate of a similar solitary wave has been calculated using Equation (4.25). The theoretical wave speed has also been calculated using McCowan's theory (Equation (4.7)), taking into consideration the attenuated wave height. The comparison is presented in Figure 4.34 by overlaying the theoretical crest height and the time of arrival at each of the numerical wave gauges onto the graph shown in Figure 4.14. The theoretical calculation begins by assuming the initial wave height occurs at the $x = 2\text{m}$ gauge. Between the 2m gauge and the 20m gauge, the SPH wave height reduces by 5.8%, compared to a theoretically predicted reduction of 0.8%. Based upon the comparison it is clear that the solitary wave height simulated with the *optimum* SPH model exhibits substantially larger attenuation.

The wave height attenuation was also considered between the two experimental wave gauges located at $x = 6\text{m}$ and 11m , the wave height at the $x = 6\text{m}$ gauge defining the initial wave height. Again, the *optimum* model was used to provide the SPH predictions. The SPH and experimental results are compared to the decay theory separately due to the different values of K in Equations (4.26) and (4.27). The results arising from these calculations are shown in Table 4.4. The attenuation of the SPH predicted wave height

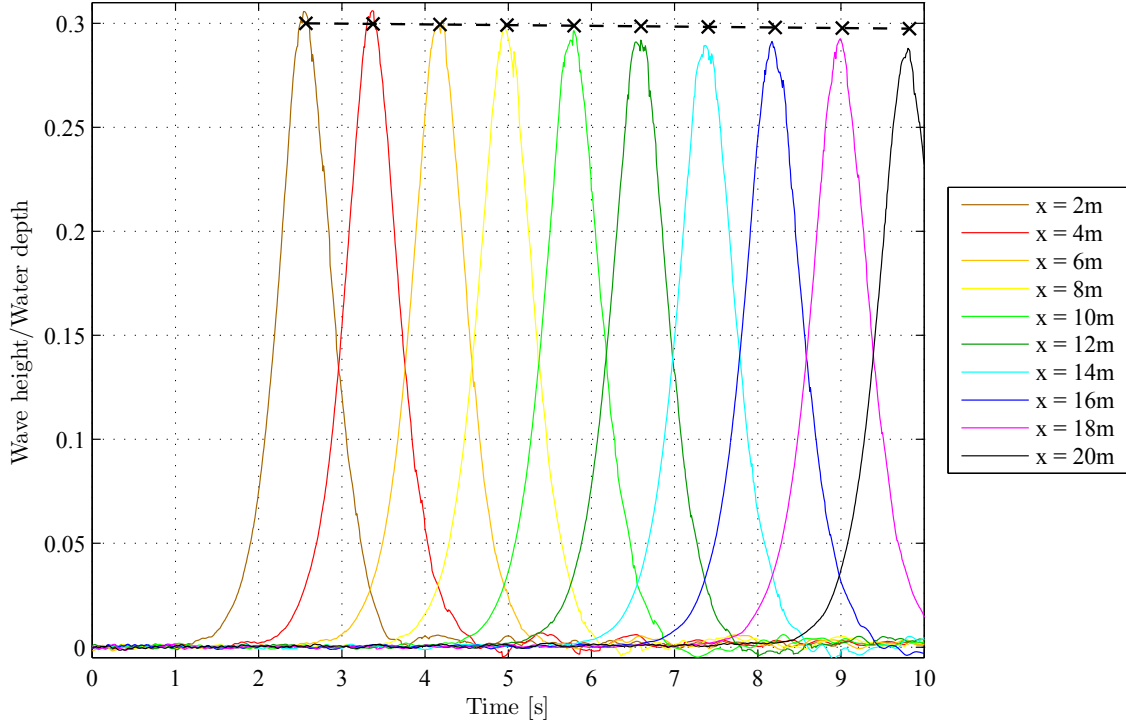


Figure 4.34: Solitary wave propagation, time histories of the water surface elevation, $\eta(t)$, predicted at a sequence of numerical wave gauges using the *optimum* SPH model with comparison to the theoretical attenuation. The legend defines the location of the wave gauges measured from the position of the paddle at the beginning of its motion, $x = 0$: — indicates the theoretically predicted wave height and $\times$ the predicted arrival time at successive gauges.

was twice that of the theoretical decay rate. However, the experimental observations describe a decay that is 5.7 times the corresponding theoretical decay.

In addition, the phase speed of the wave has been calculated by considering the time taken for the crest to pass between the two gauges. These results are also shown in Table 4.4 and are compared to McCowan’s theory, taking into consideration the wave attenuation in both the SPH simulations and the experimental observations. These comparisons show very little difference between either the SPH predictions and theory or the experimental observations and theory, with errors of just 0.45% and 0.15% respectively. These results suggest that the wave attenuation has little influence on the predicted wave speed, the results lying very close to the potential solution in all cases.

Since the wave attenuation is caused by the production of shear in the boundary layers at the bed, an SPH simulation with free-slip boundaries would be expected to more

	SPH simulation		Experimental observations	
	Calculation	Decay law Keulagan (1948)	Recorded data	Decay law Keulagan (1948)
H at 6m (mm)	148.81	148.81	146.96	146.96
H at 11m (mm)	148.10	148.46	141.16	145.94
% reduction	0.48	0.24	3.95	0.69
Phase velocity (m/s)	2.49	2.48	2.47	2.48

Table 4.4: A comparison of the wave attenuation rate and phase speed between two gauge locations comparing both SPH and experiment to theory.

closely represent the potential theories given in § 4.2.1. Unfortunately, the *optimum* SPH model identified in § 4.4.2 employs the *Stationary Ghosts* boundary formulation which cannot be used to describe a free-slip boundary condition. As a result, it was necessary to revert back to the *Repulsive Force* boundary formulation to provide comparisons between a no-slip and a free-slip boundary condition. In making these comparisons all other SPH parameters were kept as in the *optimum* model for these simulations. With the numerical paddle programmed to produce a wave with height $H = 0.150\text{m}$, the corresponding wave heights at $x = 20\text{m}$ were 0.131m for the no-slip boundary condition and 0.127m for the free-slip condition. These results, opposite to what might be expected, merely confirm that the variation in the wave height attenuation due to the physics of the bottom boundary layer (no-slip or free-slip) is small and, in the present case, completely masked by variations in the accuracy of the *Repulsive Force* implementation of the two boundary conditions.

Having undertaken these calculations, the phase velocity was also calculated based upon the travel time between $x = 2\text{m}$ and $x = 20\text{m}$. In all cases the difference in phase speed between each of the simulations and their respective theoretical calculations was less than 0.5%. This is not surprising given that the inclusion of the attenuation makes little difference to the theoretical predictions, and earlier calculations with the no-slip boundary condition were shown to be in very close agreement with the potential theory.

Overall, the inverse fourth-power decay law due to Keulagan (1948) suggests that

some attenuation of the wave height is to be expected due to shear generated in the laminar boundary layers. This will occur in both the SPH calculations and the laboratory observations. However, since the latter effects appear much larger than the theoretical predictions, particularly in the case of the laboratory observations, it is concluded that the main source of the wave height attenuation arises from the accuracy of the solitary wave generation, rather than effects arising in the boundary layer. Even with small inaccuracies in the generation of the solitary wave, it will begin to degenerate or break-up as it propagates. This has been observed both numerically and experimentally, and has resulted in a good description of the phase velocity, but higher than expected wave height attenuations.

4.8 Conclusions

A two-dimensional SPH model has been developed and optimised for simulating non-violent, free-surface flows. This has been achieved using a case study concerning the propagation of a solitary wave, with $H/d = 0.3$, over a distance comparable to one hundred times the wave height. Individual aspects of the model have been investigated in isolation and an *optimum* model has been compiled through selection of the most appropriate techniques working in combination with one another. The *optimum* model was found to satisfactorily propagate a solitary wave over the required distance with only minimal changes in the wave height.

The *optimum* model was then compared further with theory, by considering the precise form of the wave profile ($\eta(t)$ and $\eta(x)$), the wave speed and the horizontal water particle velocities, $u(z)$, under the crest of the wave. Initially, Boussinesq's first-order solitary wave theory was used for comparison, but the SPH predicted waves were found to have a wider wave profile at the leading and trailing ends of the wave. The SPH predicted wave also travelled slightly slower than the Boussinesq theory predicts and did not conform to the assumption that the horizontal water particle velocity was depth uniform.

Other solitary wave theories were discussed in § 4.2.1. As a result of the differences between the SPH wave and those predicted by Boussinesq's theory, comparisons of the wave profile were made with McCowan's first-order theory and Laitone's second-order

theory. The initial comparison with McCowan's theory was excellent and, as a result, further comparisons were made with the wave profile at different times and locations and with the horizontal water particle velocities under the wave crest. In all cases the SPH results were found to compare extremely well with McCowan's theory, the only exception being a slight reduction in the SPH predicted wave height. In contrast, the SPH wave profile did not compare well with Laitone's second order theory. The most likely explanation for this is that the SPH wave was produced by the motion of a numerical paddle based upon first order Boussinesq theory. This gives rise to a fundamental miss-match in the energy flux and for this reason, Laitone's theory was not considered further.

Further SPH simulations were carried out with $H/d = 0.1, 0.2$ and 0.45 , which were also compared to McCowan's theory. In the $H/d = 0.1$ and 0.2 cases the comparisons were again excellent and very similar to the original case, with $H/d = 0.3$. Small differences appeared in the comparison with the $H/d = 0.45$ case. However, this was to be expected as nonlinear effects are more apparent in solitary waves with larger height to depth ratios. Therefore, it is likely that the wave with $H/d = 0.45$ would compare better to a higher-order theory, but with some account taken of the loss in wave height. In order for the SPH solitary wave to settle, or evolve, into the form of a higher-order solitary wave, some loss in wave height is inevitable, not least because a higher-order solitary wave has a greater energy flux than its first-order representation. This also provides the most likely explanation for the increased loss in wave height in the $H/d = 0.45$ case.

The SPH predictions were also shown to be in good agreement with experimental observations. In making these comparisons it was noted that the experimental wave showed a greater reduction in wave height than the SPH simulation as it propagated along the flume. This was to be expected since the increased loss is likely to have occurred due to friction on the side walls.

The reduction in wave height and wave speed in both the SPH simulation with the *optimum* model and the laboratory experiment were further analysed by comparing the attenuation of a wave with $H/d = 0.3$ to the theory of Keulagan (1948). It was found that the wave height reduced by more than the theory predicts in both the SPH simulation and experiment. Nevertheless, the laboratory experiment produces a solitary wave with the desired form at the 11m wave gauge by design and the reduction in wave

height in the SPH simulation is minimal over approximately the first 10m of propagation.

The results presented in this chapter have demonstrated that a suitable SPH model has been developed to simulate a propagating solitary wave. On this basis, investigations into solitary wave breaking can be carried out with confidence that the solitary wave is modelled correctly immediately before overturning begins. Chapter 5 will continue the work on modelling solitary waves, by investigating their behaviour as they run-up, overturn and break on a uniform slope.

5

An SPH study of solitary waves breaking on a plane beach: quantitative comparisons to laboratory observations

5.1 Introduction

The motivation for this chapter lies in the desire to be able to accurately model long waves coming ashore. An investigation by Shuto (1985) concerning the 1983 Nihankai-Chibu tsunami on the north coast of Japan illustrated that the tsunami profile is very similar to that of a solitary wave. Since the Boxing Day tsunami in 2004, which had disastrous consequences in many areas in South-East Asia, there has been considerable worldwide interest in predicting when tsunamis will occur and the impact they will have as they come onshore. Accordingly, the development of reliable numerical models, able to simulate the full tsunami event, is undoubtedly becoming a useful tool in such predictions. However, it is essential that numerical models used to simulate such events have been fully validated and can be relied upon to make accurate predictions.

This chapter focuses primarily on the overturning and breaking of solitary waves on a plane slope. Previously, a lot of work has been carried out, theoretically, experimentally and numerically, into the propagation of solitary waves, which is discussed in § 5.2. Many of the numerical models used previously are based on potential flow, so are unable to

include viscous and turbulent effects. With many of the earlier methods based on the Boundary Integral method, they are also unable to model large deformations of the free surface and its eventual break up, which potentially occurs both before and after the jet touches down on the preceding wave slope. Indeed, it is important to understand the full breaking procedure, not least because it is here that energy is dissipated, affecting the maximum run-up on the slope. In a real tsunami, this would affect the impact on coastal areas where many people live and work. In such an event it may also be important to be able to predict the precise locations of both the breaking wave and the subsequent hydraulic jump which occurs as the supercritical flow rushes back down the slope. Although experiments can often provide the most accurate predictions of the wave breaking procedure, they are often time-consuming and expensive to conduct. It can also be tricky to continuously vary certain parameters. Therefore, it is desirable to develop a numerical model capable of simulating the full process of solitary waves breaking on a slope, including overturning, jet impact with the slope, the resulting splash up, the maximum run-up height and the hydraulic jump occurring during wash down. For this reason the SPH method has been selected.

In the previous chapter, the development of an SPH model was discussed and its ability to propagate solitary waves along the horizontal bed of a flume was investigated. In the present study, the SPH model has been used to generate solitary waves of two height to depth ratios, H/d , both of which result in wave breaking on a plane beach. The SPH calculations provide information on the ability of the model to capture the overturning and breaking process on the slope, observe the maximum run-up height and capture the hydraulic jump as the water washes back down the slope. These results are then validated against new experimental measurements using a visualisation technique, which allows quantitative comparisons to be made.

This chapter begins with a review of previous work investigating solitary waves breaking on beaches in § 5.2. This is followed in § 5.3 by a description of the SPH model, including some further developments and details of the numerical simulations. The experimental procedure used to validate the numerical simulations is described in § 5.4. Results of the investigations are presented in § 5.5 and the main conclusions are drawn in § 5.6.

5.2 Background

The full process of solitary wave breaking on a plane slope is investigated in the present chapter using SPH modelling, with comparisons to new laboratory observations. In the literature, wave breaking has often been considered using numerical models which are unable to explicitly describe the wave as it breaks, but can continue to estimate the run-up on the slope afterwards. Some of the earliest studies were of this nature; such studies also provided breaking criteria to determine whether a solitary wave will break as it travels up a slope. This is discussed in § 5.2.1. Following on from this, § 5.2.2 looks into previous comparisons of numerical simulations with laboratory observations, using boundary integral type methods. These solutions allow the breaking process to be simulated up to some point during overturning, but cannot continue beyond the point at which the overturning jet impinges on the preceding wave slope. Other, more recent additions to the literature, have focused on the full breaking procedure and post-breaking effects, such as jet impact and subsequent splash-up. This is discussed in § 5.2.3. Finally, earlier work carried out using the SPH method is discussed in § 5.2.4.

5.2.1 Solitary wave breaking criteria and run-up on slopes

As a solitary wave begins to propagate up a slope, its wave height will increase. Depending on the height to depth ratio of the wave and the steepness of the slope, the wave will either simply run-up the slope or will overturn and break first. There have been many studies into this process, both numerically and experimentally. Through these investigations, attempts have been made to analyse the run-up and classify waves according to whether they will break or not and, if so, the nature of the breaking that occurs.

Some of the earlier research into solitary wave run-up on slopes included experiments by Hall & Watts (1953), from which they deduced a function to determine wave run-up, but did not distinguish between non-breaking and breaking waves. Synolakis (1987) carried out an extensive study into solitary wave run-up on plane beaches. This included the development of linear and nonlinear theories based on the shallow water equations. Laboratory experiments, involving breaking and non-breaking solitary wave run-up on a slope of 2.88° , were undertaken to support the theory. The results were used to determine

an empirical relationship describing the run-up associated with breaking solitary waves on this particular slope.

In addition, Synolakis (1987) also established a breaking criteria to determine whether a solitary wave will break as it runs up or washes back down a slope. An expression describing the normalised maximum run-up for non-breaking solitary waves on any slope angle was also determined. This last result was found to be in reasonable agreement with the observations of Hall & Watts (1953) and the numerical predictions of Pederson & Gjevik (1983) and Kim *et al.* (1983). Li & Raichlen (2001) improved upon this expression for non-breaking wave run-up and Li & Raichlen (2003) derived an energy balance model for breaking solitary wave run-up. This latter approach assumes any energy dissipated at the free-surface and on the bottom slope, as well as the energy of the reflected wave, are negligible. With the kinetic energy set to zero, the maximum run-up relative to the constant water depth immediately in front of the slope, R/d , is given by

$$\frac{R}{d} = \left(\frac{1 - E_B/E_I}{1.5\alpha} \right) \left(\frac{H}{d} \right), \quad (5.1)$$

where

$$\frac{E_B}{E_I} = C[A \ln(\cot \beta) + B] \quad (5.2)$$

and

$$A = -0.470 \left(\frac{H}{d} \right) + 0.534, \quad (5.3)$$

$$B = 2.165 \left(\frac{H}{d} \right) - 1.154, \quad (5.4)$$

$$C = 0.190 \left[\ln \left(\frac{H}{d} \right) \right] + 0.969. \quad (5.5)$$

Within these equations, β is the slope angle, E_I is the incident wave energy, E_B is the energy dissipated during breaking and α is a shape function dependent on the form of the run-up tongue. In considering this latter value, Li & Raichlen (2003) later assigned a value of $\alpha = 0.18$. The empirical equation, (5.2), is fitted to the numerical data for

the range $0.1 < (H/d) < 0.4$ and slopes ranging between 1:5.67 and 1:50. Although the fit is reasonable it includes some large errors, of around 10%, for higher values of H/d .

In the past, there have been three common numerical approaches used to simulate solitary wave run up on slopes. The first of these uses the non-linear shallow water equations. This was adopted by Li & Raichlen (2003) in the development of their energy balance model described above. Although comparisons with experimental observations suggest that this method is able to describe the maximum wave run-up after breaking reasonably accurately, it is unable to model the breaking process explicitly. Indeed, it instead treats the breaking process as a bore with a shock, adopting a method more commonly used in gas dynamics. As a result, the details of the breaking procedure are lost. Furthermore, although the maximum run-up seems to be reasonably well predicted, the wash-down is less well predicted. Given these shortcomings, the reliability of this method is often questioned.

A second group of numerical simulations are based on solving the Boussinesq equations. Whilst these are capable of modelling wave breaking through the inclusion of *ad hoc* terms, they require calibration with empirical results. Unfortunately, these are not always available, particularly if attempting to model real-life events such as tsunamis. In common with the methods based on the nonlinear shallow water equations, these simulations cannot model the water surface profile of the breaking wave. Once again, they rely on a bore to represent the post-breaking processes. Examples of this type of method are given by Zelt (1991), who used a Lagrangian finite-element Boussinesq wave model, and Stansby (2003), who used a semi-implicit finite-volume method to study solitary wave run-up on various slopes. More recently, Borthwick *et al.* (2006) carried out validation tests using a Godunov-type one-dimensional Boussinesq model coupled with a non-linear shallow water equations model for predicting post-breaking behaviour. These studies considered solitary waves breaking on a plane beach and subsequent run-up, with a view to developing an engineering design tool for modelling tsunamis coming onshore. Excellent agreement with the experiments of Synolakis (1987) was found for predicting run-up on a 1:20 plane beach. Comparisons to experiments carried out in the UK Coastal Research Facility were also presented, in the form of surface elevation time-series and wave profiles collected at a series of wave gauges located along the cross-shore centre-line of the flume. Good agreement was found as the solitary wave approached

the toe of the slope. However, some differences were evident in the wave height as it steepened and moved over the plane beach. In addition, turbulence in the water surface behind the wave was observed experimentally, but absent in the numerical predictions due to the nature of the model. Despite these differences, the model serves its purpose as a successfully validated analysis tool for predicting tsunami run-up for engineering applications.

The third approach is to use the fully nonlinear boundary integral equation, usually implemented within a boundary element method (BEM). However, this is a potential flow model and is unable to model surface break-up. This prevents the model from simulating the surface profile beyond the development of a plunging breaker. As a result, it is unable to simulate the run-up resulting from solitary waves which have broken on the slope. However, using a method of this type, Grilli *et al.* (1997) investigated the breaking criterion for solitary waves on a slope and determined that solitary waves will break when the height to depth ratio exceeds the following condition.

$$\frac{H}{d} > 16.9(\tan \beta)^2, \quad (5.6)$$

where β is again the slope angle. Using this numerical model, Grilli *et al.* (1997) also determined a non-dimensional slope parameter and gave ranges of this value for each type of wave breaking that occur. These are given as $S_0 < 0.025$ for spilling breakers, $0.025 < S_0 < 0.30$ for plunging breakers and $0.30 < S_0 < 0.37$ for surging breakers, where S_0 is given by

$$S_0 = \frac{1.521 \tan \beta}{\sqrt{(H/d)}}. \quad (5.7)$$

In identifying these criteria, Grilli *et al.* (1997) adopted the arbitrary definition that a spilling breaker occurs when the jet size at touchdown is less than half the wave height. Furthermore, to achieve these results extensive filtering and re-gridding was required and this inevitably casts some doubt regarding the exact shape of the breaking wave.

5.2.2 Overturning solitary waves

Comparisons between numerical simulations and laboratory observations of overturning solitary waves are rare in the literature. Examples include studies by Yatsuda *et al.*

(1997), Li & Raichlen (1998) and Grilli *et al.* (2004). Yatsuda *et al.* (1997) used a boundary integral equation method, based on the Cauchy integral formula, and made comparisons with laboratory observations of solitary waves propagating over a submerged breakwater. They noted that the water surface profiles of plunging solitary waves on submerged breakwaters are similar to those observed on beach slopes. The limited comparisons to laboratory observations that they provide look reasonable, but the detail is indistinguishable at the scale shown in the paper. However, the authors were satisfied with these comparisons and went on to study and classify breaking waves using the numerical model. In doing so they investigated the surface profile and kinematics during overturning in considerable detail, reaching several interesting conclusions relating to the underlying physics of breaking waves.

Li & Raichlen (1998) carried out laboratory experiments using a paddle programmed according to Goring (1979) and a high-speed video camera to record results. Image processing was carried out to eliminate distortion and skewness caused by viewing the wave through the glass sidewalls. They compared their results to the numerical results of Grilli *et al.* (1997), who used a fully nonlinear potential flow model based on a BEM solution to simulate solitary waves with $H/d = 0.3$ and $H/d = 0.45$. Li & Raichlen (1998) found that the results compared well with two notable exceptions. First, the numerically predicted wave speed was faster after breaking than that observed in the experiments. Second, the numerical wave jet was significantly thicker. However, rather than using a paddle to generate the solitary waves, Grilli *et al.* (1997) initiated his numerical simulations by defining the free-surface profile of a solitary wave according to the theory of Tanaka (1986). This meant that the solitary wave was not subject to any dissipation as was the case experimentally. In order to rectify this discrepancy, Grilli *et al.* (1998) re-ran his numerical simulations initiating the wave motion with a paddle moving in the same way as the experimental paddle used by Li & Raichlen (1998). Following this adjustment, it was found that the numerical wave speed was in much closer agreement to the experimental data. However, the results were not directly compared, so quantitative comparisons were not possible. Furthermore, the new simulations had little effect on the thickness of the overturning wave jet.

Grilli *et al.* (2004) carried out further numerical simulations using his BEM to model two solitary waves breaking on a compound slope, the main section of the slope having

an angle of 3.81° . The smaller of the two waves, with $H/d = 0.135$, compared reasonably well with the experiment in terms of the plunging breaker profile. However, the larger wave, with $H/d = 0.211$, is less well described. In the numerical simulations a plunging breaker was produced, whilst the experimentally observed wave jet shows significant instabilities and appears more like a spilling breaker. Unfortunately, a BEM solution is unable to model the occurrence of wave spilling. In considering this point, Christou (2008) speculated that this discrepancy could be attributed to the re-gridding process used by Grilli *et al.* (2004) in an attempt to prolong the simulation beyond what is normally achievable with the BEM. This process effectively smooths the free-surface, particularly in the region of the breaker jet, resulting in predictions which are not physically realistic.

In an attempt to improve upon the solitary wave results of Grilli *et al.* (2004), Christou *et al.* (2007) used a new, multiple-flux BEM, developed by Hague & Swan (2007) to simulate solitary waves overturning on a 5° slope and made thorough quantitative comparisons with new laboratory observations. This model does not use any smoothing, filtering, re-gridding or redistribution of the surface nodes. As such it prevents plunging breakers forming when spilling should take place and discourages unphysical thickening in the overturning wave jet. As a consequence, this model becomes unstable sooner than the BEM of Grilli *et al.* (2004) and so it cannot simulate the overturning process through to the wave jet touching down on the slope. However, if results beyond the point where the Hague & Swan (2007) model breaks down are unphysical in predictions made using the Grilli *et al.* (2004) model, then this may not be considered a disadvantage of the new model.

Christou *et al.* (2007) considered two solitary waves, the first with $H/d = 0.3$ and the second with $H/d = 0.45$, both propagating in a water depth of $d = 0.5\text{m}$. The BEM of Hague & Swan (2007) was used and compared to experiments carried out in the glass-walled coastal flume located in the Hydrodynamics Research Laboratory of the Department of Civil and Environmental Engineering at Imperial College London. This apparatus and the visualisation technique used to process the photographic images are described in § 5.4 of the present chapter, since the experiments are very similar to those conducted in the current study. Once the predicted effects of viscous decay in the experimental wave flume had been allowed for in the programming of the paddle

used to produce the waves, the numerical and experimental waves were compared at the breaking point. This point is defined as the point at which the front face of the wave becomes vertical and is adopted because it is clearly defined in both time and space. As the solitary waves approached and moved through the breaking point, the quantitative comparisons with experiment were excellent; the maximum relative errors were 3% for the $H/d = 0.3$ case and 2% for the $H/d = 0.45$ case. As the waves evolved into the true plunging breaker shape, the relative errors increased to 4.5% and 3% respectively. Nevertheless, this still represents very good comparisons with detailed laboratory observations. However, shortly after this time the numerical model broke down due to the sharp edge forming in the wave jet. As a result, the plunging breakers were unable to continue towards touchdown on the slope. Interestingly, it is at this point that the SPH model is anticipated to out-perform any BEM based models; SPH has no difficulty in dealing with large geometric deformations of the free-surface.

Christou *et al.* (2007) also presented the numerical model's predictions of the underlying water particle velocities. These confirm the criteria that for a wave to break, the water particles must be travelling faster than the wave speed, c . Indeed, comparisons between the two confirm that they are of equal magnitude around the breaking point. It was also shown that before the wave started to overturn, the particle velocities were almost uniform with depth, whilst in front of the breaking wave the water barely moved. Time histories for the maximum surface height and the relative height of the wave to the local water depth were also provided in comparison with the experimental results. These give further insight into the mechanisms of solitary waves and provide confirmation of the numerical model's accuracy up to the point where it breaks down.

Overall, Christou *et al.* (2007) provided some excellent comparisons with the laboratory observations using the BEM of Hague & Swan (2007). However, given the inherent limitations of any BEM solution, the model is unable to describe the whole breaking process, particularly the subsequent run-up and wash-down. These processes are important if numerical models are to be able to simulate real-life tsunamis. For this reason, the SPH model developed in this thesis has been used to extend the numerical modelling capability for this type of flow, where the water surface becomes severely deformed and breaks up.

5.2.3 Post-breaking behaviour

Thorough and complete studies of the breaking of solitary waves are rare in the literature. Once details of the overturning wave form have been addressed, authors generally jump forward to the study of wave run-up and wash-down on the slope. This omits a complete study of the wave jet impacting on the slope and the highly turbulent processes which subsequently occur. However, if a numerical model is to be able to correctly predict events following wave breaking, such as run-up on a beach, it is important that the model predicts the correct flow regimes throughout. If this does not happen, then this can lead to numerical results which may appear to provide good comparisons with laboratory observations for the particular case involved, but they cannot be extended to model other situations. As a result, such models may be unreliable when predicting what would happen in a real event, which is the ultimate goal when developing numerical models.

One study concerning the post-breaking behaviour of solitary waves has been carried out by Guignard *et al.* (2001). This study employed a modified version of the Volume of Fluid (VOF) method to model solitary waves breaking on a mild slope and compared the results to experiments. The method captures the movement of both the water and the air, whilst the interface between the two was modelled by the authors own method. In the VOF method the interface is calculated in each cell through the use of a function whose value is the fraction of the cell occupied by the denser fluid. In the original VOF methods, the interface was assumed to be parallel to the grid lines. This was modified in a subsequent version of the method, known as Piecewise Linear Interface Calculation (PLIC), to allow the fraction of the cell to be a segment of any orientation. In Guignard *et al.* (2001), the method was improved further by adding Lagrangian advection of the segments, the method being called the Segment Lagrangian-Volume of Fluid method (SL-VOF). The method was validated using some simple free-surface flow tests and also compared with the BEM of Grilli *et al.* (1997) applied to a solitary wave propagating over a horizontal bed of constant depth. The results seem to compare well with only small errors, although some loss of energy and fluid volume under the solitary wave were observed. Indeed, both the wave amplitude and the fluid velocity are reduced in comparison to the BEM by the time the wave reaches the breaking point.

The SL-VOF method was then used to model a plunging breaking wave and again comparisons were made with laboratory observations. Although this method showed some potential in the previous tests, very few good quality results were achieved after the waves had broken. During overturning the SL-VOF wave appears to have broken faster than the experimental observations suggest. Furthermore, although the location of the numerical and experimental wave breaking is claimed to be the same, the evidence to support this is rather dubious. Most importantly, although the height to depth ratio has been kept the same between the numerical and experimental tests, the actual wave heights and water depths are not identical. In effect the results have been non-dimensionalised and it is comparisons between this data set that are provided. Beyond the point of the wave impacting on the slope, only numerical results are given and these are not compared to the experiments. This renders the results somewhat meaningless as there is no indication as to whether the results compared well with the laboratory observations or not.

A detailed analysis of post-breaking solitary wave behaviour is, however, given by Li & Raichlen (2003). Experimental images are shown capturing the full breaking process, from overturning, through jet impact, splash up and the development of the so-called run-up tongue. Two distinct processes were described in the post-breaking stage, depending on the slope and the height to depth ratio of the incident solitary wave. For smaller waves, it was observed that the jet resulting from the overturning wave impacts on the dry slope beyond the original shoreline. In contrast, larger waves break before they reach the shoreline and so the jet impacts onto the front face of the incident solitary wave, resulting in a second jet being generated and sent almost vertically upwards. This eventually collapses and contributes to the subsequent run-up tongue. During this process a number of vortices are created, both in the water and air phases. The relationship between the slope and the post-breaking behaviour is not as clear, as experimental results were only given for three slopes. Li & Raichlen (2003) also observed that the run-up following the breaking of solitary waves was unaffected by this post-breaking behaviour. Furthermore, the maximum run-up was found to change continuously with the incident wave height.

5.2.4 SPH solitary wave studies

When it comes to modelling solitary waves with SPH, most studies concentrate on the method's ability to capture the full-breaking process, showing the most beneficial aspects of SPH. One of the first SPH studies into free-surface flows, Monaghan & Kos (1999), investigated solitary waves propagating in a simple tank and then running up a slope (the Cretan beach) and breaking as they were reflected off a vertical wall placed at the end of a flat section at the top of the slope. Although their comparisons with experiments lack detail, they show that the SPH method is broadly capable of capturing the main features of solitary wave run-up and breaking waves. Since the publication of this paper, many authors have gone on to improve upon the SPH model and apply the method to many similar free-surface flows with large deformations and break up of the free-surface. Dalrymple *et al.* (2007) and Dalrymple *et al.* (2009) used an SPH model to make qualitative comparisons of solitary wave run-up with the experimental results of Synolakis (1987), again showing that SPH was successful in capturing the main features of the experiments.

Lo & Shao (2002) used an incompressible SPH (ISPH) method to simulate solitary waves propagating over a horizontal bed, running up a vertical wall and breaking on a mild slope. The propagation results were excellent, showing very little reduction in wave height, even over distances of around a hundred times the water depth. The maximum run-up on the vertical wall was shown to be in good agreement with experiment and details of the pressure and turbulence were observed in the SPH simulation. The turbulent kinetic energy was found to be small in a simulation where the wave did not break, and a run without a turbulence model confirmed that turbulence can be neglected for this type of simulation. The results of the solitary wave breaking on a mild slope were qualitatively compared with the experimental results of Synolakis (1987). Unfortunately, the comparisons do not actually show the formation of a plunging breaker as the wave overturns, but jump straight from close to the breaking point to a post-breaking stage, where run-up has already begun. Although the results were said to be in good agreement with the experiments, the plots make this very difficult to see, particularly as only a few SPH particles are shown close to the free-surface and the vertical scale is highly distorted. If the ISPH method is to be seriously considered, it must be able to

demonstrate its ability to simulate the full process of breaking waves.

ISPH was also used by Khayyer *et al.* (2008), who considered the post-breaking behaviour of solitary waves on slopes. In this paper, a corrective term was introduced to the ISPH model to ensure conservation of angular momentum (CISPH). The new model was used to study five cases of solitary waves breaking on slopes: one involving a surging breaker, one a spilling breaker and three plunging breakers. One of the plunging breaker cases was qualitatively compared with the laboratory experiments of Li (2000) and Li & Raichlen (2003). Although the results showed that the method is capable of producing a plunging breaking wave, with the same large-scale features as the experiment, the numerical simulations are by no means a good quality resemblance of the experiments; SPH particles ‘fall’ out of the overturning wave, the air-gap under the overturning wave is too small and no indication is given as to whether the SPH wave breaks at the same time and location as the experimental wave. Results produced using an uncorrected ISPH model are also shown and, in this case, it can be seen that the model is completely incapable of predicting either a plunging wave or the subsequent post-breaking behaviour.

Returning to the results produced using the CISPH model, in terms of the post-breaking behaviour, the secondary splash-up jet is produced, but it is at a far shallower angle than in the experiment. The authors attribute these differences to the emergence of numerical errors resulting from a completeness problem with SPH. Specifically, they relate this issue to the inability of the kernel interpolants to exactly reproduce the physical field, particularly the highly non-linear velocity field. Two of the plunging breaker cases were also compared quantitatively to both experiments and other numerical methods throughout the breaking process. A comparison of wave heights during breaking and post-breaking reveals that this method can produce reasonable results post-breaking, but before the jet impact, the wave height predictions are too low. It is, therefore, uncertain as to whether or not the model is reliable post-breaking, since this is dependent upon pre-breaking predictions, which were not successfully modelled. Conclusions regarding the accuracy and appropriateness of this model are therefore very difficult to assess.

Issa & Violeau (2009) conducted a study into the performance of a range of turbulence models during the breaking of a solitary wave on a slope. The setup is similar to

that considered in this chapter, with the results compared to the experiments of Li & Raichlen (2003) by drawing a line over the apparent water surface visible on the nearside of the flume. Although no visualisation technique was used to quantify the measurements, the accuracy was sufficient to examine differences between the results of the various turbulence models. A solitary wave was generated with $H/d = 0.4$ and allowed to break on a 1:15 slope. Three turbulence models were tested: a constant eddy viscosity model, the mixing length model and the k -equation model. All three are different to that used for the work carried out in this thesis, although the underlying two-dimensional SPH model is similar. As the solitary wave begins to overturn and touchdown on the still water surface in front, the accuracy of the surface profile is similar with all three turbulence models and in good agreement with the experiments, suggesting that the turbulence model is not important up to this stage. However, once the water begins to splash back up, the results start to deviate. Initially only the k -equation turbulence model produces satisfactory results, but later, all three deviate from the experimental profile, with the mixing length and k -equation results being of similar accuracy. The authors speculate that even a k - ϵ turbulence model would not be sufficient for capturing the high rate of strain and curvature involved in the splash-up process and suggest a more sophisticated model, such as the Explicit Algebraic Reynolds Stress Model, may be necessary to improve the SPH results. They also suggest that modelling the air phase would help, and in common with the results presented in this chapter, it would seem that three-dimensional effects are important in the latter stages of breaking.

5.3 Numerical simulations

5.3.1 Introduction

The *optimum* model (developed in the previous chapter) provides the basis for the model used in the present chapter, where it is used to simulate a solitary wave breaking on a slope. However, it was found that some additions to the model were necessary to improve the quality of the simulations in these more complex and violent flows. As anticipated, it proved beneficial to include a turbulence model with an appropriate turbulent boundary condition. In addition, a problem was observed with the *Stationary Ghosts* modelling technique when either fluid particles moved over a previously dry

boundary or fluid particles moved away from a boundary which would subsequently dry out. A solution to this, the *Hybrid* boundary representation, was achieved by combining different boundary modelling techniques and is described in § 3.9. The turbulence model and the turbulent boundary condition are discussed in the following sections.

5.3.2 Turbulence modelling in SPH

As the solitary waves begins to overturn and break on the slope, the flow ceases to be laminar and turbulent processes start to become important. It is therefore beneficial to introduce a turbulent model and here a large eddy simulation (LES) approach has been used with a sub-particle scale model, similar to sub-grid scale models used in grid-based methods, following the work of Dalrymple & Rogers (2006). The governing equations are Favre-averaged and it is assumed that the equations are sufficient to resolve the large eddies. In contrast, modelling vortices smaller than the particle resolution requires a closure model. Within the present study, the standard Smagorinsky model is used to determine the eddy viscosity. Full details of this model are described in § 3.5. Since turbulence is inherently three-dimensional in nature, accordingly, turbulence models are only truly appropriate in three dimensions. However, the turbulence model used for the two-dimensional simulations described herein should be sufficient for capturing a reasonable representation of the turbulent processes involved.

SPS models are based on mixing length models, where the mixing length corresponds to the estimated size of vortices which are smaller than the particle spacing. This model has been chosen for use here in part because it has been used successfully by so many others in SPH modelling. As well as the work of Dalrymple & Rogers (2006), it has been used in SPH modelling by Rogers & Dalrymple (2004) and Lo & Shao (2002) and with the incompressible MPS method by Gotoh *et al.* (2001). It has also been chosen because it is relatively easy to program and does not significantly increase the simulation runtime due to the fact that it is fundamentally an algebraic model. Other authors, notably Violeau & Issa (2007), prefer to solve either the k - L_m one-equation model for turbulent kinetic energy or even the k - ϵ two-equation model, which also solves for turbulent dissipation. These models are based on fewer assumptions than mixing length models and should therefore be more accurate. However, they also require additional equations, which must be solved and the variables time-marched as the solution progresses. This substantially

increases the computational time and effort. As a result, although they hold significant potential for future use, the SPS model is preferred at present.

5.3.3 The turbulent boundary condition

On solid wall boundaries, the turbulence model described above is insufficient and so an additional boundary condition must be added to the turbulence model. In SPH, like grid-based methods, this is usually based on the “law of the wall”, which is given for hydraulically smooth walls by Equation (3.84) and for rough walls by Equation (3.85). In grid based methods, this condition is usually applied at the first point away from the wall (Ferziger & Peric (2002)), which is assumed to be in the logarithmic layer. However, given that SPH is based on moving particles, rather than a grid, this is not possible. As a result, the method proposed by Violeau & Issa (2007) has been applied, whereby the turbulent boundary condition is applied on the boundary particles themselves. In adopting this approach, the boundary particles are now assumed to be a distance δ away from the geometrical boundary; δ is chosen such that it is small compared to the particle spacing, but should be larger than the viscous sub-layer thickness or roughness length if the wall is considered to be rough. In the simulations that follow, δ was chosen to be equal to one tenth of the particle spacing. The wall shear velocity, u^* , is calculated for each particle, at each time step, using δ and the velocity gradient on the wall, by rearranging Equation (3.86).

Since the present simulations are to be validated against laboratory observations, the turbulent boundary condition should be setup to simulate the roughness on the bed of the experimental flume. As these walls are made from glass and perspex, it would seem appropriate to class them either as hydraulically smooth or having a very low roughness coefficient depending on the Reynolds number of the flow. However, it was found that applying a realistic condition caused the turbulent effects to be too strong for these particular simulations. This was probably due to the viscous sub-layer, which cannot be determined *a priori*, being thicker than δ in parts of the flow, particularly before the wave breaks. As the purpose of the turbulent boundary condition is to model realistic effects in a numerical model, it is essentially a numerical condition rather than a strict physical parameter. The SPH model itself could also have had an affect on the required boundary condition due to the SPH particles, the viscosity model or even

the usual laminar boundary model. Accordingly, lower values of U^+ were tested, where $U^+ = \bar{u}/u^*$, and it was found that U^+ lying in the range $1 < U^+ < 2$ gave the best results. Within this range, specific values of U^+ made little difference to the calculated results. Further details concerning the turbulent boundary condition and the required equations are given in § 3.10.

5.3.4 SPH model

As in the previous chapter, the numerical simulations were undertaken in a two-dimensional, numerical wave flume. As before, the solitary waves were generated using a numerical wave paddle located at one end of the flume, which was programmed to move as described in § 4.2.2. The only difference in the paddle motion is that for the simulations carried out in this chapter the centre point of the paddle motion was located at $x = 0\text{m}$ to facilitate comparisons with laboratory experiments.

In all of the simulations undertaken herein, the numerical flume consisted of a uniform section with a horizontal bed that was 5.15m long. The wave paddle was located at one end and a plane slope, or beach, located at the other, the slope making an angle of 5° to the horizontal. A sketch of the computational domain is given in Figure (5.1). The length of the beach was designed to be long enough to ensure that the maximum run-up could be measured without the flow encountering any additional solid boundaries. The length of the uniform section was chosen to be exactly 6m shorter than that of the experimental flume. The reason it was not the same length lies simply with the need to limit the computational runtime. Shortening the flume allowed the simulations to retain the desired resolution whilst reducing the total number of SPH particles, thus limiting the computational effort.

In order to achieve valid comparisons with the laboratory observations, it was necessary to ensure that the solitary waves had the correct height to depth ratio as they approached the toe of the slope. The laboratory experiments are described in the following section of this chapter, where it will be seen that a wave gauge was placed 0.15m upstream from the toe of the slope. This was replicated in the numerical simulations, the wave height being checked at this point using a numerical wave gauge (see § 4.3.2). In the previous chapter it was noted that even with the *optimum* model, there was a small loss in wave height as the waves propagate over the length of the flume. However,

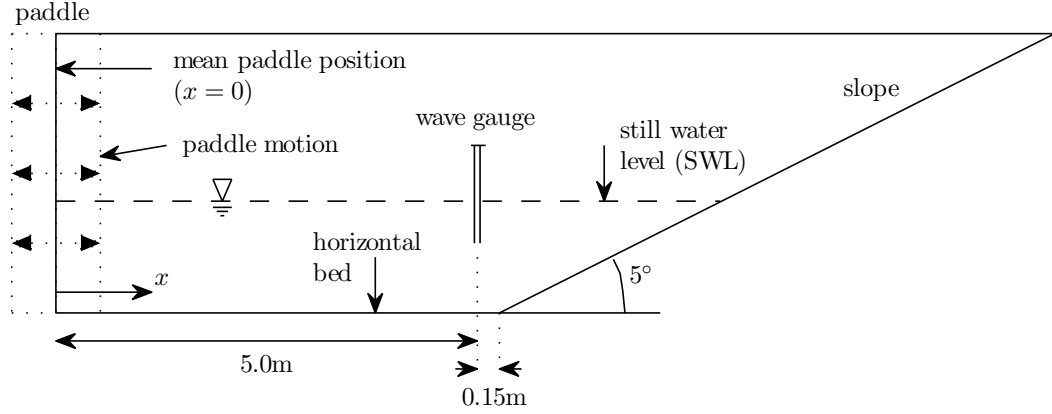


Figure 5.1: Diagram of the numerical wave flume, showing the location and angle of the slope, the wave generating paddle and the location of a numerical wave gauge (not to scale). (Note: the experimental flume had exactly the same setup, except that the horizontal bed section was 11m long rather than 5m.)

as the solitary waves only need to propagate over a distance of 5m in these simulations, it was found that the SPH predicted solitary wave did not lose any height over this distance. As a result, the wave paddle was programmed with the desired height to depth ratio and no further adjustments were necessary.

As in the previous chapter, SPH particles were initially placed above the flat section of the flume on a Cartesian grid. For particles initially located above the slope, the Cartesian grid was rotated such that the particles were stacked perpendicular to the slope. Some particles lying above the toe of the slope were removed to ensure that no two particles were placed too close together. The other option for placing SPH particles above the slope would have been to keep the standard Cartesian grid and ensure no particles at the bottom were placed too close to the boundary in the slope. However, it was found that when using the *Stationary Ghosts* boundary model, the problem described in § 3.9, meant that the particles were unable to move into the gaps and pockets of incorrect pressure that were formed at regular intervals along the slope. As a result, rotating the Cartesian grid for SPH particle placement above the slope was found to be the preferred solution. An initialisation run, with no paddle motion, was also carried out for every simulation, to allow the particles to settle into position, particularly above the toe of the slope where the two grids meet. The initial particle

formation is shown on Figure (5.2).

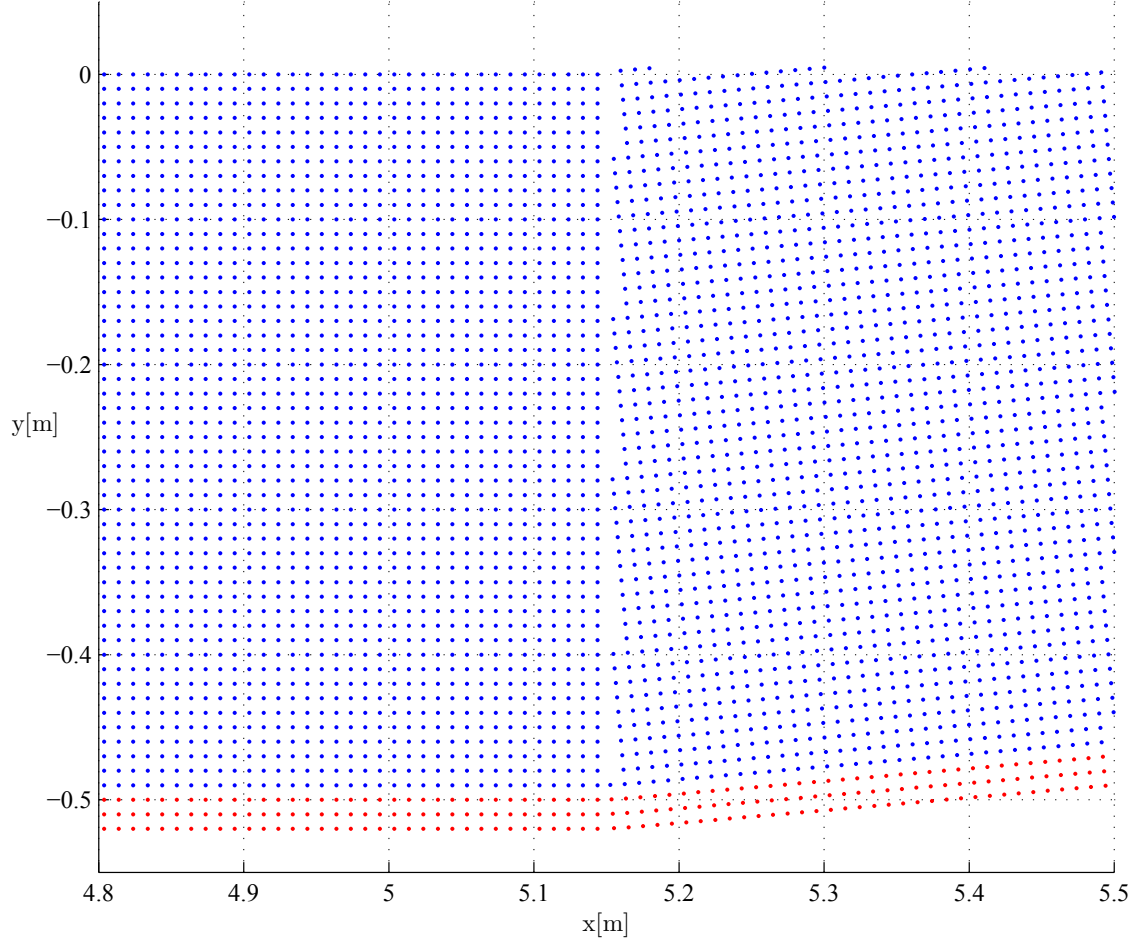


Figure 5.2: Initial layout of SPH particles above the toe of the bed slope, the Cartesian grid being parallel to the flat bed on the left of the toe of the slope and parallel to the bed slope on the right of the toe of the slope. • SPH water particles and • ghost particles. (Note: some particles above the toe have been removed to prevent particles being too close to one another at the start of the simulation.)

In-keeping with the previous chapter, all the present simulations were undertaken with a smoothing length of $1.3\Delta x$ and the speed of sound was calculated following Equation (3.29), with $v_{max}^2 = g(d + H)$ taken from Boussinesq's first-order solitary wave theory. A variable time step size, Δt , was used based on the Courant condition given by Equation (3.88) and the additional condition given by Equation (3.90), resulting in time steps of the order of $\Delta t = 10^{-4}$ s.

The first simulation in this chapter was carried out with the *optimum* model from

the previous chapter. This run used a solitary wave of $H/d = 0.3$ in a water depth of $d = 0.5\text{m}$. The initial particle number to depth ratio in the uniform section of the flume was kept at 50, as it was in the previous chapter, giving a total number of SPH particles of around 43,000 and an initial particle spacing of $\Delta x = 0.01\text{m}$.

In a second simulation, the new modelling techniques for dry boundaries, turbulence modelling and a turbulent boundary condition have been included. The resolution was kept the same and, once again, this was run for a solitary wave of $H/d = 0.3$ in a water depth of $d = 0.5\text{m}$.

Finally, the resolution was increased to give an initial particle number to depth ratio in the uniform section of the flume of 100. Two cases were run in order to make a range of quantitative comparisons with the experiments. The first case, *Case 1*, is a solitary wave of $H/d = 0.3$, run in a water depth of $d = 0.5\text{m}$, giving an initial particle spacing of $\Delta x = 0.005\text{m}$ and approximately 173,000 SPH particles. The second case, *Case 2*, is a solitary wave of $H/d = 0.45$, run in a water depth of $d = 0.4\text{m}$, giving an initial particle spacing of $\Delta x = 0.004\text{m}$ and approximately 202,000 fluid particles. The reduction in water depth to $d = 0.4\text{m}$ for this solitary wave was due to an improvement in the experimental paddle motion when producing this wave. The increase in resolution causes a reduction in the length of the time step, Δt , such that it is of the order of $\Delta t = 5 \times 10^{-5}\text{s}$ for these simulations.

5.4 Experimental investigations

5.4.1 Laboratory apparatus

The solitary wave experiments were conducted using the glass-walled coastal flume in the Hydrodynamics Research Laboratory of the Department of Civil and Environmental Engineering at Imperial College London. The flume is 25m long, 0.5m wide and has a maximum depth of 1m. For the solitary wave experiments the water depth was set at $d = 0.5\text{m}$ for *Case 1* and $d = 0.4\text{m}$ for *Case 2*. In accordance with the numerical simulations, the relative wave heights were $H/d = 0.3$ for *Case 1* and $H/d = 0.45$ for *Case 2*. A beach with a slope of 5° was constructed within the flume with the toe of the slope located 11.15m from the mean position of the wave maker. The beach was constructed from 10mm thick perspex sheets rigidly supported by a metal frame and

sealed to the side walls of the flume. At the other end of the flume, a piston wave maker was programmed to generate the solitary waves. Its movement is described in the following section. The experimental setup is identical to the numerical setup shown in Figure 5.1, the only difference being that the horizontal bed section between the wave maker and the toe of the slope was 6m longer than the numerical domain.

The water surface elevation, $\eta(t)$, was recorded using surface-piercing, resistance wave gauges, each gauge providing a time-history of the surface elevation at a single, fixed location in space. In both tests, two wave gauges were used to check that the correct wave heights were achieved and to ensure repeatability in the experiments. These were placed at $x = 6\text{m}$ and $x = 11\text{m}$, with x being measured from the centre of the paddle motion. A sample rate of 128Hz was used to record the water surface elevation, $\eta(t)$, at each gauge position. Previous studies undertaken within the hydrodynamics laboratory have shown that using these wave gauges, $\eta(t)$ can be measured with an accuracy of $\pm 0.5\text{mm}$.

5.4.2 Wave generation

The wave maker was programmed using the theory outlined in § 4.2.2 and is therefore consistent with the numerical simulations. However, the software controlling the wave maker is based upon the spectral, or frequency, content of the desired wave-field. In practice, the wave paddle continuously generates a large number of closely spaced front numbers, each representing a known frequency component. As a result, the signal sent to the wave paddle needs to be periodic over some large fundamental time interval. In a theoretical sense, the period associated with a solitary wave is infinitely long. In order to replicate this as closely as possible, the fundamental period, or repeat period, adopted in the experimental generation was chosen to be as long as is practically acceptable, a value of 128s eventually being defined. Within this time interval, the desired paddle motion is described as follows.

First, the paddle movement is centred around the midway point in the motion used to generate the required solitary wave. At $t = 0$ the piston moves backwards very slowly from this central point, so as to cause as little disturbance as possible to the water in the wave flume. Once the piston has reached the starting position for the motion needed to generate the solitary wave, the piston swings forwards, replicating the motion defined in

§ 4.2.2. As it passes the centre point, it is exactly half way through the desired motion and has generated one half of the desired solitary wave. When the piston reaches the end of the motion defined in § 4.2.2, it will have come to rest. Beyond this point, it moves very slowly back to the central point, thus completing the generation cycle. In moving back to the central point, the paddle again moves very slowly so as to cause minimal disturbance to the tail end of the solitary wave.

Having described the desired motion of the wave paddle, a Fourier transform of the displacement time-history will define the corresponding spectral content, and this can be assembled in the standard form for implementation in the generation software. Finally, it is important to note that although the wave maker is equipped with active force-feedback control, the nature of the present generation does not require such a facility and may be adversely affected by it. As a result, the force-feedback was switched off and the paddle operated with straightforward positional control, which is ideally suited to the generation of solitary waves. The success of the solitary wave generation has already been demonstrated in § 4.6; Figure 4.33 indicates very good agreement between the theory due to McCowan and the laboratory observations.

5.4.3 Quantitative flow visualisation

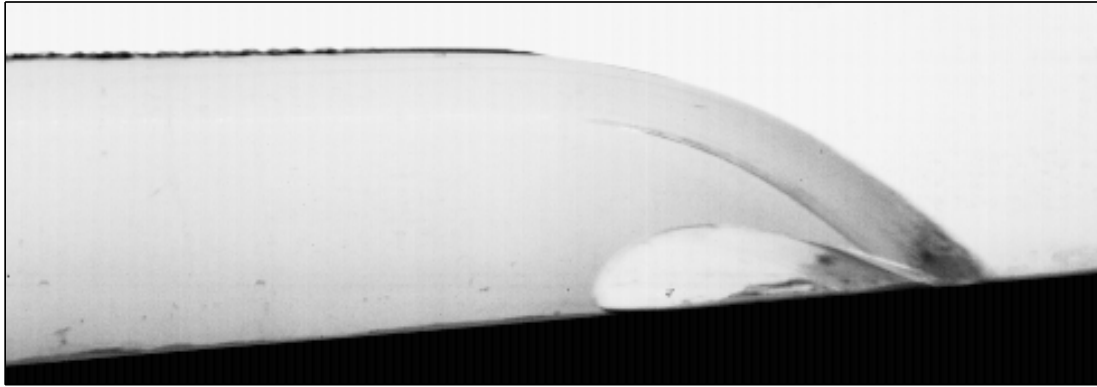
A flow visualisation technique was used to measure the solitary wave profile as it travelled up the slope, including the occurrence of wave overturning and breaking. This involved the use of a computer controlled DALSA DS-21-001M0150 area scan camera used with sufficient background lighting to obtain good quality images of both the evolving water surface and the sloping beach. This camera contains a one-megapixel (1024×1024) CMOS image sensor and was set to capture 100 frames per second. During each experimental run the camera was switched on for a total of eight seconds, such that 800 frames were captured per run. From a practical perspective, this represented the maximum possible duration of a single record due to computational memory constraints.

In undertaking the present tests, the camera was placed parallel to the glass walls of the wave flume in a pre-determined location chosen to ensure one particular aspect of the flow could be captured. In each wave case, the experimental run was repeated three times. The first was undertaken with the camera located to capture the overturning wave. In the second, the camera was located further up the slope, to capture the

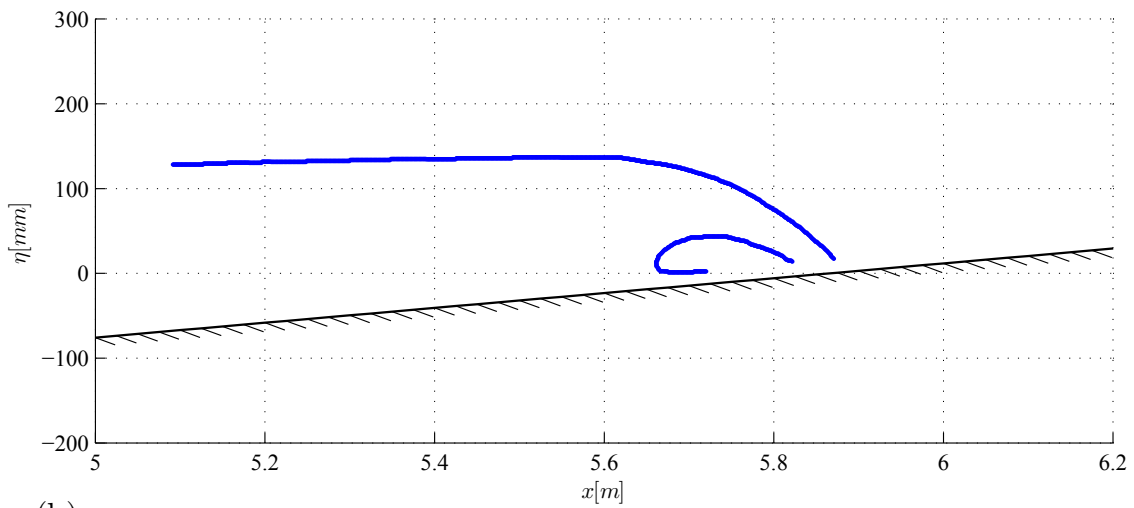
jet from the plunging breaker impacting on the slope. Finally, in the third run, the camera was located further down the slope and the start time of the recording delayed in order to capture the hydraulic jump which forms as a result of the supercritical wash-down running into the almost stationary water below. In addition to the quantitative flow visualisation, two wave gauges were located within the wave flume to ensure the repeatability of the wave conditions.

During the experimental runs, the background lighting was provided by shining several high-intensity flood-lights through a white, opaque perspex sheet mounted on the opposite side of the glass-walled flume. The sheet, which was 4mm thick, distributed the light evenly over the area visible to the camera. This was complimented by two further flood-lights: one below the flume, shining directly upwards, and the other secured above the flume and positioned to further illuminate the perspex sheet. This combination of lights was used to help reduce shadows which can become particularly prominent as the wave overturns, forming the distinctive plunging breaker shape. Figure 5.3a provides an example of the quality of the captured images; this particular image shows a solitary wave in the process of overturning.

Having recorded an appropriate section of the wave evolution, individual frames were selected and an edge detection technique developed by Canny (1986) was used to identify the water surface profile, including the development of any internal air pockets. This approach uses a Gaussian filter to search for local maxima within the intensity gradient and was found to be very effective. The next step in the analysis involved transforming the pixel coordinate system obtained from the digital photographs onto a two-dimensional spatial coordinate system. In order to achieve this, a 50mm $\times$ 50mm metal grid was placed on the front glass wall of the wave flume at the end of each experimental run. With the camera kept in exactly the same location, one further picture was taken defining the details of the metal grid. Using this information, the spatial location of each grid intersection can then be matched to the corresponding pixel in the photograph. The pixels can be defined in terms of their row and column coordinates, (i, j) , by the formulae



(a)



(b)

Figure 5.3: Results arising from the quantitative flow visualisation applied to an overturning solitary wave: (a) An example photograph taken in the laboratory study, indicating the high quality of the images and (b) the defined water surfaces, including the air void, based on the image given in (a) above.

$$i = a_1x + a_2y + a_3$$

$$j = b_1x + b_2y + b_3, \quad (5.8)$$

where the constants a_1 , a_2 , a_3 , b_1 , b_2 and b_3 are determined by a least squares fit. Using these values it is possible to transform any line, defined in terms of its pixel locations, onto a two-dimensional Cartesian grid. An example of the data that results is given in

Figure 5.3b, these particular results being based on the image given in Figure 5.3a. In developing this technique, Spentza (2006) found that the error due to refraction at the glass-air interface is less than 0.3% of the incident wave amplitude and that the absolute accuracy of the resulting data is $\pm 2\text{mm}$.

5.5 Discussion of results

5.5.1 Preliminary results

Initially, the *optimum* model developed in the previous chapter was used to generate a solitary wave with $H/d = 0.3$ in the flume described in § 5.3.4 and allowed to break on the 5° slope. This SPH simulation was then repeated with the improvements to the boundary model, the additional turbulence model and the boundary condition included, as described in § 3.9, § 5.3.2 and § 5.3.3. This version of the SPH model shall be referred to as the *improved* model in the remainder of this chapter. The results of these simulations are presented alongside each other and both are compared to laboratory observations from the experiments described in § 5.4. The results are presented first as a time-history of the water surface elevations, $\eta(t)$, at the wave gauge located 0.15m before the toe of the slope and, second, as a progression of visualisations showing the SPH water particles' positions and the location of the experimental water surface. These latter plots are presented for the solitary wave at its breaking point (as $\eta(t)$ becomes vertical), as it starts to overturn and as the overturning jet nears impact with the slope ahead. Photographs from the experiments will be presented alongside the final model results in the following section of this chapter. At this stage the experimental results are presented in their processed form, the water surface being represented as a blue line in the visualisations, as was the case in Figure 5.3b.

In order to directly compare the numerical and experimental results, it was first necessary to make the two comparable. The wave flumes employed in the separate experimental and numerical studies were of different lengths. Furthermore, the time taken for the experimental wave paddle to produce the desired wave form was considerably longer than the numerical wave maker due to its periodic motion, as described in § 5.4.2. In order to compare the generated wave events, the two sets of results have been aligned in both time and space. With both numerical and experimental wave gauges located

0.15m to the left of the toe of the slope, the arrival of a wave crest at this location was chosen as an appropriate position and time at which to enforce this alignment. As a result, this defines the origin for both the spatial location (x) and the time (t).

In undertaking the experiments, it was necessary to make a small alteration to the initial water depth in the laboratory wave flume and to programme the wave paddle to produce a slightly larger wave event in order to ensure that the generated wave had the desired height at $x = 0$ (assuming $d = 0.5\text{m}$, $H = 0.15\text{m}$ to maintain $H/d = 0.3$). The explanation for these changes is as follows. First, the water depth needed to be increased to compensate for the drawing-back of the wave paddle before it begins its forward motion. This increases the effective volume of the wave flume, thus reducing the water depth by a few millimetres. Second, it was necessary to produce a slightly larger wave to compensate for the effects of the viscous decay resulting from the glass walls of the wave flume. After a number of iterations, an initial water depth of $d = 0.509\text{m}$ and a height to depth ratio of $H/d = 0.302$ was found to give the desired wave height of $H = 0.15\text{m}$ and water depth of $d = 0.5\text{m}$ at the position of the wave gauge, 0.15m to the left of the toe of the slope. Having considered the numerical calculations, it was found that there was no need to increase the input wave height, the success of the model and the reduced length of the flume ensuring that any loss in wave height was negligible.

Comparisons between the recorded and predicted time histories of the water surface elevation, $\eta(t)$, at $x = 0$ are shown in Figure 5.4 using the *optimum* SPH model and in Figure 5.5 using the *improved* SPH model. These comparisons confirm that good agreement is achieved for both the *optimum* model and the *improved* model. Given the success of the results discussed in the previous chapter, based upon the *optimum* model, a very good level of agreement was to be expected. Nevertheless, the agreement is important for the validity of the forthcoming results. Furthermore, little difference between the two sets of results would be expected at this stage; the effects of the improvements to the model should not become important until the wave starts to break. However, some small discrepancies at the tail end of the solitary waves are apparent. These are most likely to have been caused by inaccuracies in the wave generation, as discussed in § 5.4.2.

Since the numerical and experimental waves compare well as they reach the toe of the slope, it is now possible to make comparisons of the breaking process as they run up

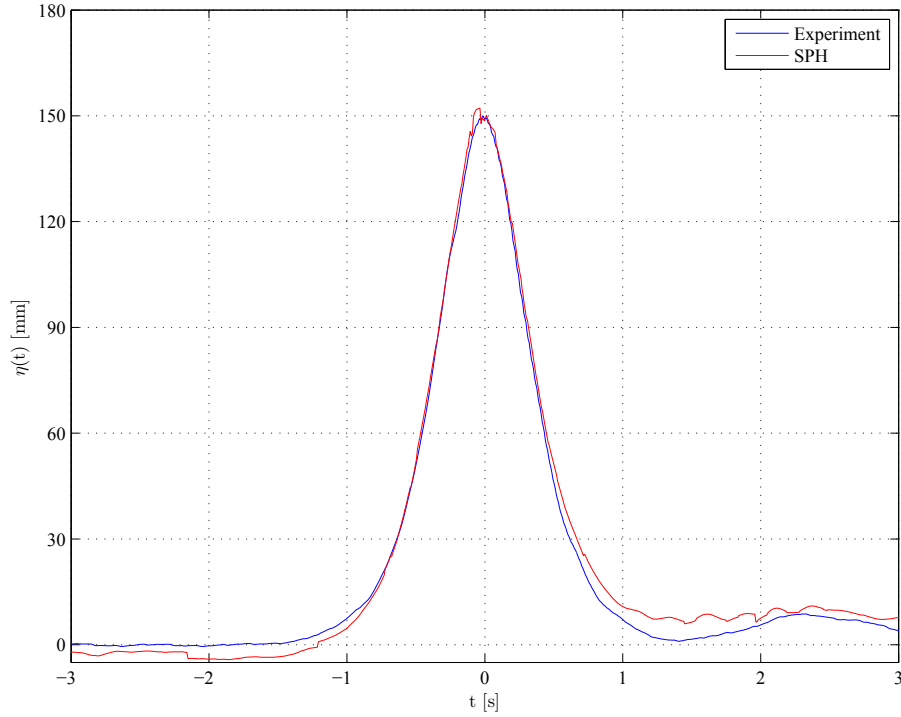


Figure 5.4: Time histories of the water surface elevation, $\eta(t)$, during solitary wave propagation with $H/d = 0.3$. Comparison between laboratory observations and initial SPH predictions based upon the *optimum* model at the wave gauge located 0.15m before the toe of the slope ($x = 0$).
— laboratory observations — SPH model predictions.

the 5° slope. The initial comparisons have been made at the breaking point, or the point at which the front face of the wave becomes vertical. The time taken for the wave to travel between the toe of the slope and breaking point was recorded both experimentally and numerically in the *optimum* model simulation and was found to be 1.96s and 2.06s respectively.

A comparison of the *optimum* model results with the laboratory observations of the solitary wave at the breaking point is shown in Figure 5.6. In order to facilitate subsequent comparisons the time, t , has been set to zero, both numerically and experimentally, at this point. The data presented in Figure 5.6 confirms that the wave heights and the wave profiles compare very well and their locations on the slope are similar. However, there is a small difference in the precise location of the breaking front; the SPH wave reached this point approximately 26mm further along the slope than the experimental wave. The SPH wave may have travelled further up the slope than the

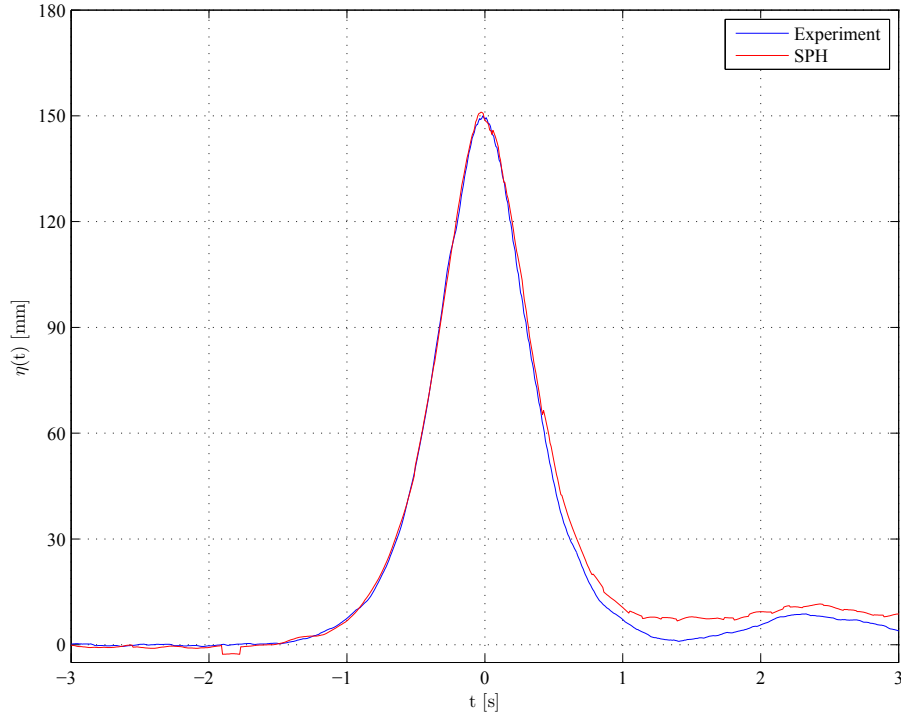


Figure 5.5: Time histories of the water surface elevation, $\eta(t)$, during solitary wave propagation with $H/d = 0.3$. Comparison between laboratory observations and SPH predictions, with improvements to the model, at the wave gauge located 0.15m before the toe of the slope ($x = 0$).
— laboratory observations — SPH model predictions.

experimental wave due to the lack of sidewalls in the two-dimensional SPH model. In the experimental observations, friction from the sidewalls is likely to prevent the wave from propagating as far up the slope.

The difference between the numerical predictions and experimental observations of the time taken to travel between the wave gauge near the toe of the slope and the wave breaking point is 0.1s, corresponding to a difference of approximately 5%. However, the difference between the distance the two waves travelled during this time is only around 0.5% of the total distance travelled. The difference in time is relatively small, particularly when considering the actual magnitude, while the difference in the distance travelled is negligible. As the SPH wave travelled further up the slope before reaching its breaking point, if the two waves were propagating at the same speed, the SPH wave would be expected to take longer to reach its breaking point. However, the small difference in the position of the breaking point on the slope can only account for around one tenth

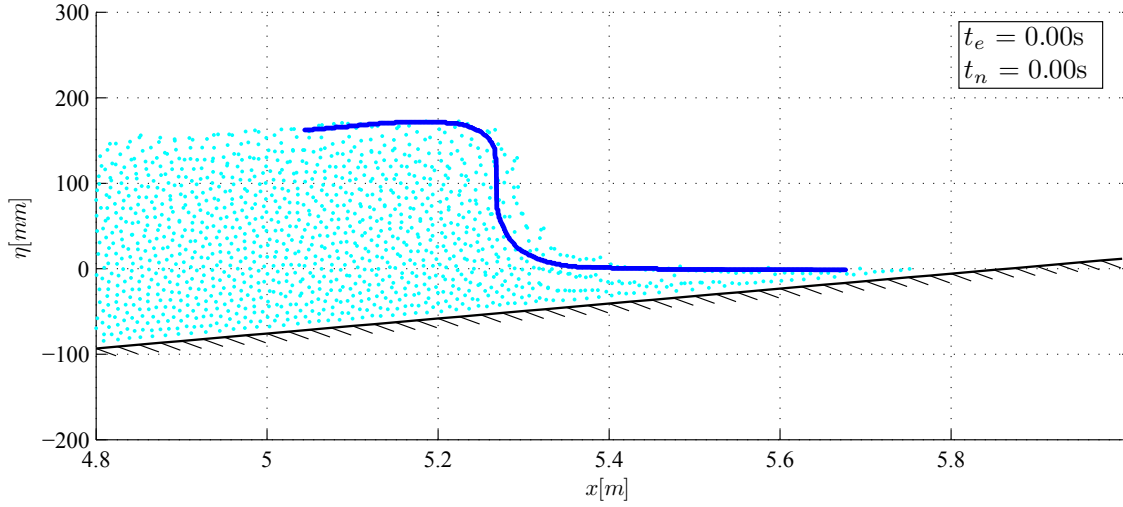


Figure 5.6: Visualisation of a solitary wave with $H/d = 0.3$ at the breaking point. Comparison between results produced with an initial SPH simulation and laboratory observations. • SPH, — experiment.

of the total difference in time, based on the propagation speed of the waves. Indeed, information from the wave gauge located 6m from the centre of the wave maker in the experimental flume and from wave gauges further to the left of the toe of the slope in the numerical flume, suggest that the solitary waves were propagating at approximately the same speed before reaching the toe of the slope. The following plots presented in this section also suggest that the two waves go through the breaking process at approximately the same rate. As a result, there is no clear reason why the SPH wave should take 0.1s longer than the experimental wave to go from just before the toe of the slope to the breaking point. However, it should be emphasised that the differences between the results are very small.

A comparison of the *improved* model results with the laboratory observations of the solitary wave at the breaking point is shown in Figure 5.7. As with the results using the *optimum* model, the numerical breaking point has been reached slightly further up the slope when compared to the experimental breaking point. This result is exactly as expected; the two sets of numerical results should not be different at this stage, since turbulent effects will not start to become important until the wave overturns. Indeed, this comparison is simply included to ensure that the additional (improved) modelling techniques have not introduced any new, unexpected problems. Since the two

visualisations are so similar, the *improved* model appears to have retained the validity of the *optimum* model established in the earlier work.

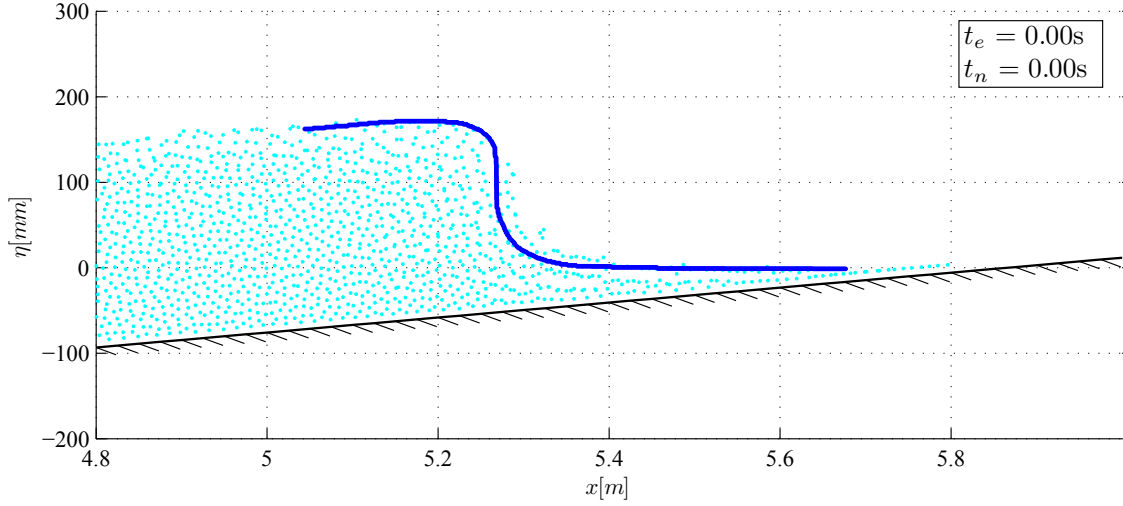


Figure 5.7: Visualisation of a solitary wave with $H/d = 0.3$ at the breaking point. Comparison between results produced with an SPH simulation, with improvements to the model, and laboratory observations. • SPH, — experiment

The next set of comparisons are made as the wave starts to overturn, Figure 5.8 using the *optimum* model and Figure 5.9, the *improved* model. For both numerical simulations the time taken to reach this stage from the breaking point is the same as in the laboratory observations. Furthermore, the surface profiles of both the SPH waves are very similar to the laboratory observations. The newly formed jet in the overturning SPH generated waves is at the same height and of a similar thickness to the experimental jet. However, at this stage of the breaking process, the advantages of using the *improved* model are becoming visible. The *improved* model produces a slightly narrower jet, which projects further forwards, rather than downwards, in comparison to the predictions of the *optimum* model. Although the differences are small, the shape of the jet produced with the *improved* model is closer to that of the experimental jet. It is believed that this improvement is due to the inclusion of the rate of strain tensor in the calculation of the eddy viscosity, within the turbulence model.

Although the numerically predicted overturning jets compare well with the experimental observations, there are nevertheless some small differences. In both SPH simulations the back wall of the developing cavity is further forward than the back wall of the

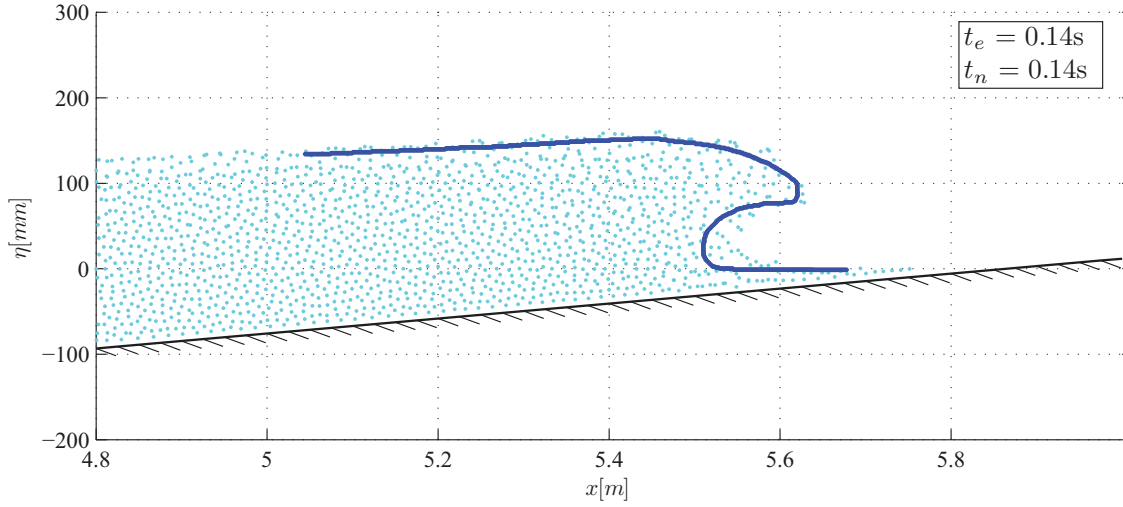


Figure 5.8: Visualisation of a solitary wave with $H/d = 0.3$ as the wave starts to overturn. Comparison between results produced with an initial SPH simulation and laboratory observations. • SPH, — experiment

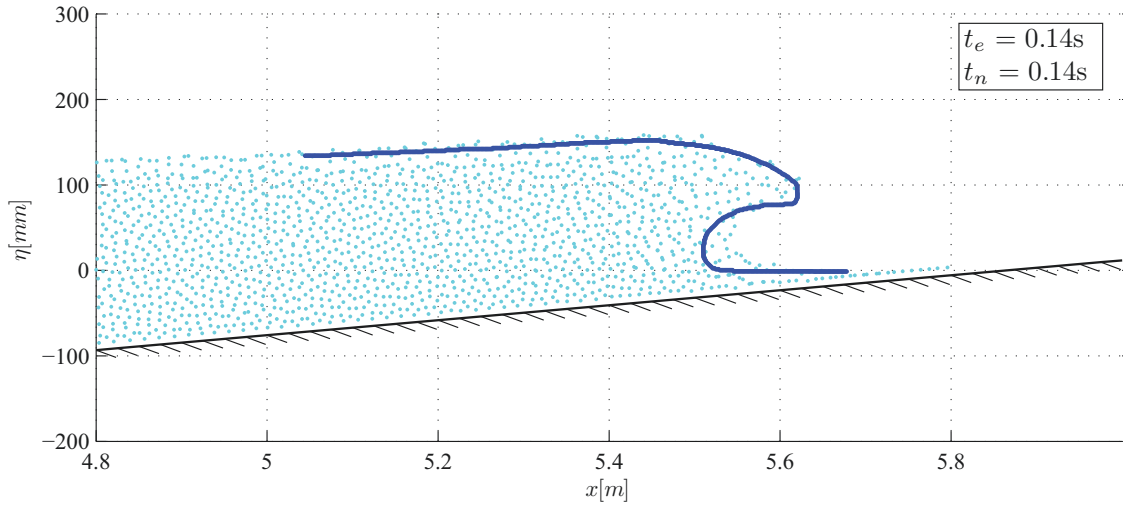


Figure 5.9: Visualisation of a solitary wave with $H/d = 0.3$ as the wave starts to overturn. Comparison between results produced with an SPH simulation, with improvements to the model, and laboratory observations. • SPH, — experiment

experimental cavity, yet the front of the overturning jets are inline with one another. The difference in the location of the back wall is around 40mm for the wave generated with the *optimum* SPH model and slightly less for the wave generated by the *improved* model. This is likely to have resulted from either the SPH waves continuing to break further up the slope, as was seen at breaking point, or because the particles are starting

to collapse inwards earlier in the breaking process than in the experiment.

If the first case is true, then even the *improved* model has not been able to project the jet forwards as far as in the experiment, suggesting a lack of forward momentum in the SPH generated waves as they start to break. This is the first time in the simulation so far where viscous forces start to become important. It is possible that the lack of forward momentum in the breaking SPH waves is related to the viscosity formulation in the model. If the shearing forces resulting from this formulation are too high, this may have the effect of slowing down the particle velocities in the overturning jet.

If the latter case is true, this could be due to the lack of a specific surface tension model being included in the SPH model or because the experimental images represent the water on the sidewall nearest the camera. In the middle of the flume water particles from the wave would be more likely to fall inwards, but the sidewalls provide increased surface tension, helping to retain the wave profile during overturning. Since the SPH model is two-dimensional it does not have sidewalls to provide this effect.

The final images in this section are taken as the overturning wave begins to touch-down on the slope in front. Comparisons between the numerical predictions and experimental observations are shown in Figure 5.10, using the *optimum* model, and Figure 5.11, using the *improved* model. The very bottom of the experimental jet is missing from the images as silicon holding the bed slope in place obscured the cameras view in the 2-3mm immediately above the bed. Once again, the wave in both numerical simulations takes the same time to reach this stage of the breaking process as in the laboratory observations and the jets touch down on the slope in the same location. As in the previous comparison, the jet produced by the *improved* model is slightly narrower than that produced by the *optimum* model. Furthermore, the thinner jet has also been projected slightly further forwards than that described by the *optimum* model predictions. Both these features indicate the beneficial effects of the additional modelling techniques used in the *improved* model as the surface profile at this stage is closer to the laboratory observations than those associated with the *optimum* model predictions. Although the jet profiles line up extremely well at this stage, unsurprisingly the back wall of the cavities are still further forwards in both numerical simulations when compared to the laboratory observations. The possible reasons for this were discussed previously.

Overall, both the *optimum* and the *improved* SPH models satisfactorily reproduced

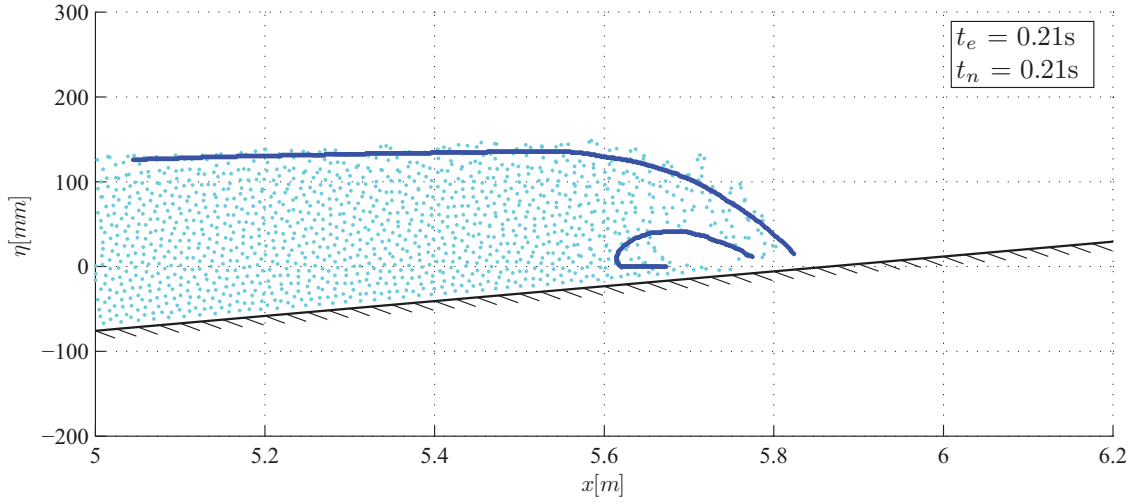


Figure 5.10: Visualisation of a solitary wave with $H/d = 0.3$ as the overturning jet nears impact with the bed slope ahead. Comparison between results produced with an initial SPH simulation and laboratory observations. • SPH, — experiment

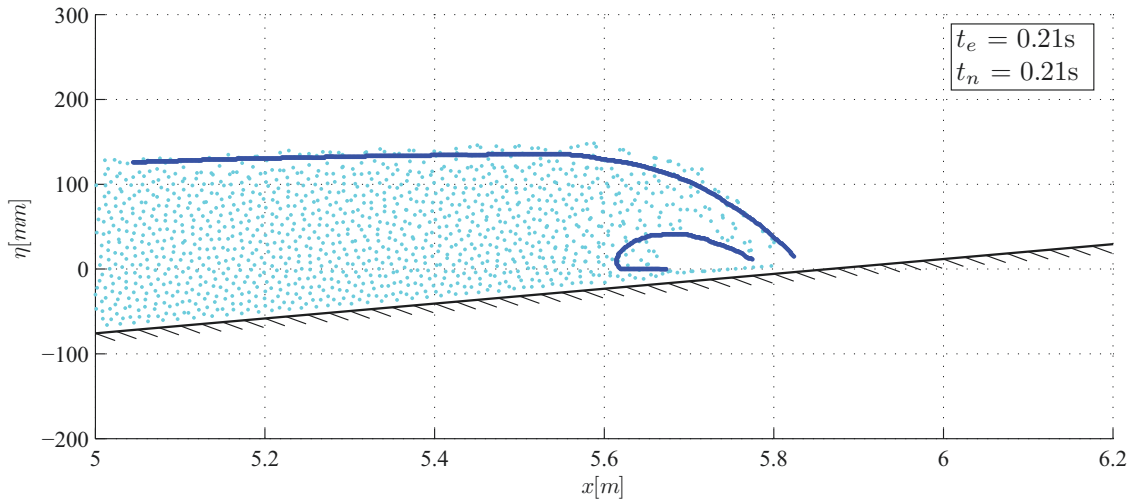


Figure 5.11: Visualisation of a solitary wave with $H/d = 0.3$ as the overturning jet nears impact with the bed slope ahead. Comparison between results produced with an SPH simulation, with improvements to the model, and laboratory observations. • SPH, — experiment

the main features of a plunging solitary wave overturning on a 5° slope. After a small adjustment to align the numerical and experimental times to coincide at breaking point, the subsequent wave breaking progressed at the same rate. In the latter stages, the introduction of a turbulence model in the *improved* SPH model has made a positive difference to the shape of the overturning jet.

The *optimum* model simulation was stopped at this stage in order to make the alterations to the numerical model which feature in the *improved* model. Although it is not shown in these images, the problem described in § 3.9, in relation to wetting previously dry boundaries, was known to cause further problems as the jet impacted with the slope and run up began. It was also anticipated that including the turbulence model would improve the quality of the water surface profile during the breaking process, particularly after the jet impacts with the slope. At this point, the intention was to allow the *improved* model simulation to run all the way through the breaking process. However, the simulation was stopped at the same stage of the breaking process as there were still a few details which would ideally be improved upon:

1. The SPH wave breaks slightly further up the slope in both model simulations.
2. The cavity is smaller in the SPH simulations, although some contribution from the sidewalls in the experiment may be responsible for this.
3. Even in the *improved* model simulation, the SPH wave seems to lack slightly in the degree of momentum necessary to drive the jet as far forwards as in the experiment, before it turns to impact on the slope.

In considering these points, it is possible that the particle number to depth ratio of 50 used in these simulations was not sufficient to fully capture all the details of the breaking process, particularly in the thin jet and the anticipated break-up of the water surface. Therefore, the resolution was increased to give a particle to depth ratio of 100 and further (final) SPH simulations were carried out. In undertaking these final simulations, the viscosity formulation was not altered and, importantly, no surface tension model was added. Indeed, it was believed that an increase in particle resolution was the most important factor in seeking to achieve the desired improvements.

5.5.2 Final model results

In undertaking the final calculations, two further solitary waves have been considered using the *improved* SPH model and an initial SPH particle to depth ratio of 100. The first, *Case 1*, is the same as the previous simulations, with $H/d = 0.3$ in a water depth of 0.5m. The second, *Case 2*, is a solitary wave with $H/d = 0.45$ in a water depth of 0.4m. As the same SPH particle to depth ratio of 100 was required, the initial particle spacing was reduced from $\Delta x = 5\text{mm}$ to $\Delta x = 4\text{mm}$ in *Case 2*.

Increasing the particle to depth ratio to 100 implies a significant increase in the total particle number and therefore causes a significant increase in the runtime. The time taken for the previous simulations to reach the breaking point was 13.5 hours for the *optimum* model and 21.5 hours for the *improved* model. However, the new *Case 1* simulation, with a particle to depth ratio of 100, took 5.75 days to reach the breaking point. Although this is a large increase in runtime, the benefits of the increased resolution were expected to outweigh this disadvantage.

The results of both the *Case 1* and *Case 2* simulations are compared to laboratory observations from the experiments described in § 5.4. As for the preliminary results described in the previous section of this chapter, results are presented as a time-history of the water surface at the wave gauge located 0.15m before the toe of the slope and as a progression of visualisations showing the SPH water particles' positions and the location of the experimental water surface. These plots begin whilst the solitary wave is at the breaking point and go through the process of the wave overturning, the resulting jet impacting with the slope ahead and the water splashing back up as run-up on the slope begins. In each case a table is provided giving the maximum run-up on the slope in both the SPH simulation and the experimental study. Finally, a visualisation of the hydraulic jump resulting from the water involved in the run-up washing back down the slope is presented. Again, visualisations comparing the SPH predictions with the laboratory observations are provided. At each stage of the breaking process a photograph from the experiment is also presented to give an indication of what the actual flow looks like.

A significant benefit of studying fluid flows numerically is the ease with which properties of the flow can be calculated. Accordingly, the final section of results describes the SPH predictions of the water particle velocity and the associated pressure.

Case 1: solitary wave with $H/d = 0.3$

Once again, the wave profiles were compared at the toe of the slope to ensure the correct input wave height was used both numerically and experimentally. This data is presented on Figure 5.12, which, once again, shows that the agreement is excellent.

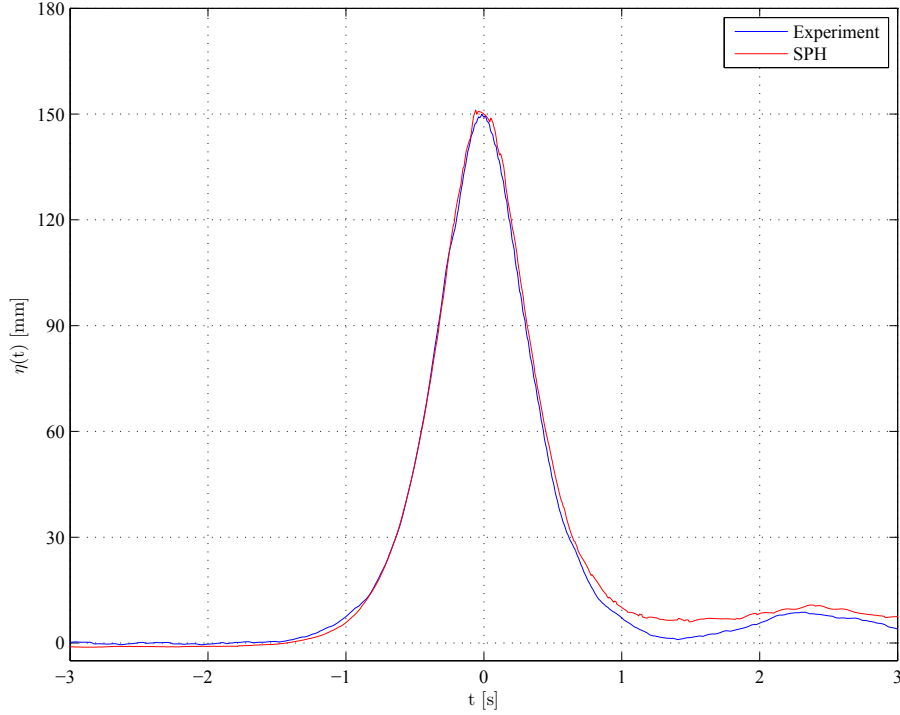


Figure 5.12: Time histories of the water surface elevation, $\eta(t)$, during solitary wave propagation with $H/d = 0.3$. Comparisons between laboratory observations and high resolution SPH predictions at a wave gauge located 0.15m before the toe of the slope. — laboratory observations — SPH model.

A comparison of the numerically predicted and experimentally observed waves at the breaking point (or where the water surface first becomes vertical) is shown in Figure 5.13. It can be seen that the higher resolution has produced much better definition in the water surface profile, $\eta(x)$. This provides a closer resemblance to the observed (experimental) water surface profile when compared to the lower resolution calculation. However, the higher resolution has not improved the accuracy of the predicted breaking position on the slope, the SPH wave front being approximately 47mm further along the slope. This is consistent with the increase in the time taken for the SPH wave to reach the breaking point after passing the toe of the slope, which took 2.08s in this simulation;

this is 0.02s longer than in the simulation with the lower resolution. The main reason for this increase in the distance travelled is, as discussed earlier, thought to be the influence of the sidewalls of the experimental flume. It would also seem that increasing the SPH particle resolution has allowed the water to move with slightly less resistance; this will be shown to be advantageous at later stages in the breaking process.

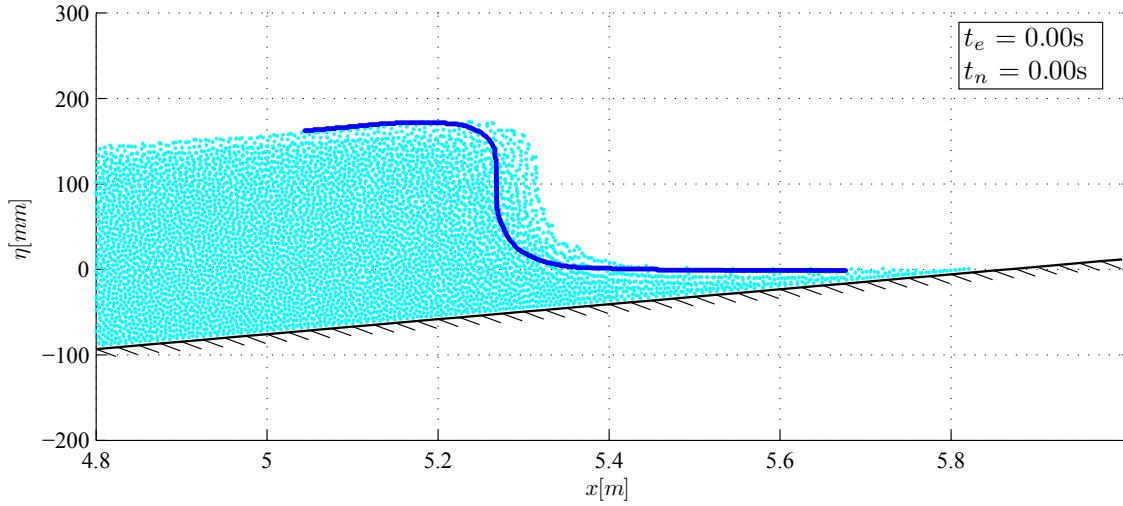
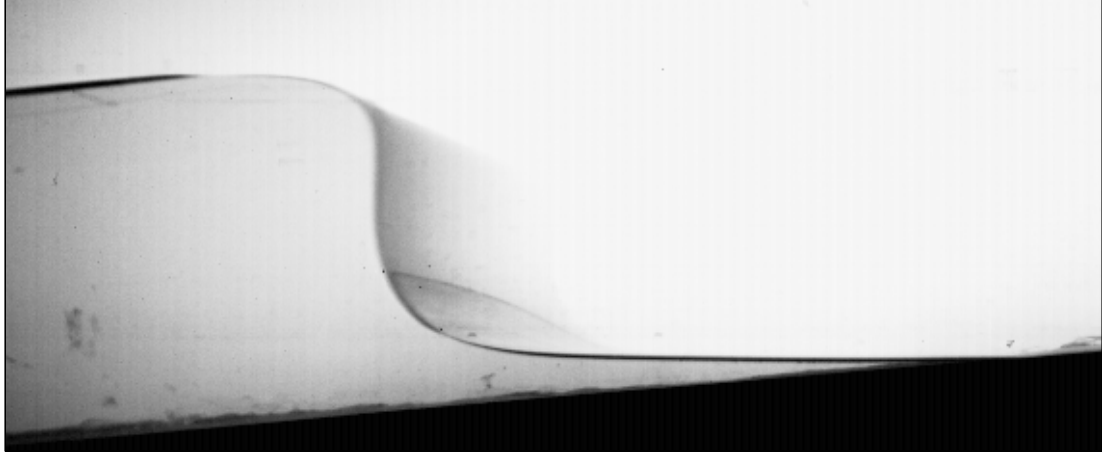


Figure 5.13: Visualisation of a solitary wave with $H/d = 0.3$ at the breaking point. Comparison between results produced with a high resolution SPH simulation and laboratory observations. — laboratory observations, • SPH model.

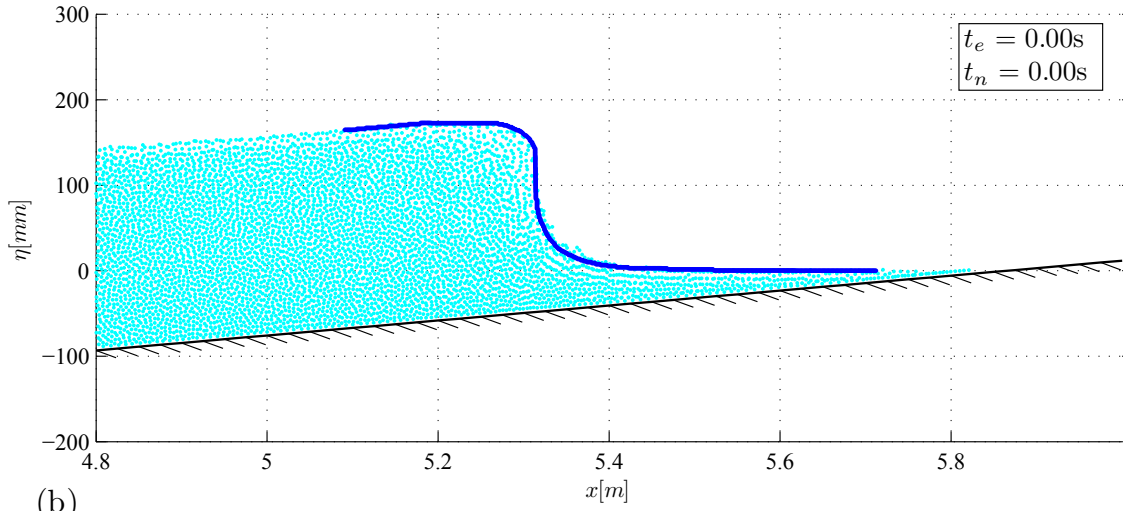
In order to aid subsequent comparisons at this resolution, the time was set to zero at the breaking point and the experimental water surface profile was shifted forwards 47mm in the x -direction to bring it in line with the numerical results. Having effectively zeroed the starting condition (relating to the breaking process), this shift has been maintained throughout the subsequent comparisons.

A photograph of the experimental wave at the breaking point is shown in Figure 5.14a, while comparisons between the SPH simulation and laboratory observations, again at the breaking point and including the shift described above, are shown in Figure 5.14b. These comparisons allow a refined comparison of the surface profiles to be made. At this stage, the predicted SPH profile is in excellent agreement with the experimental observations.

A photograph of the solitary wave as it starts to overturn is given in Figure 5.15a, with the corresponding comparison between the SPH simulation and laboratory obser-



(a)



(b)

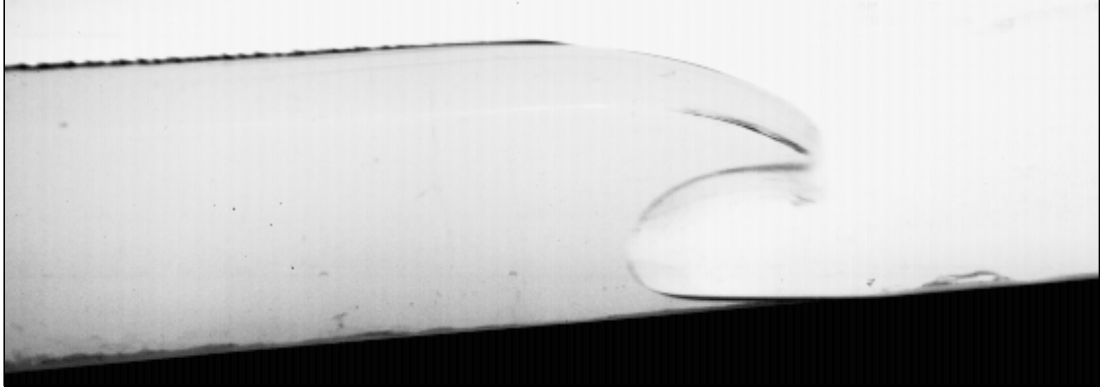
Figure 5.14: Visualisation of a solitary wave with $H/d = 0.3$ at breaking point: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations which have been spatially aligned with the SPH results: — laboratory observations, • SPH model.

vations being shown in Figure 5.15b. At this stage of the breaking process, the SPH surface profile matches the experiment very well. The shape and size of the newly-forming numerical jet have improved when compared to the lower resolution SPH results. In particular, the predicted jet is now the same length as the experimentally observed jet. However, a close inspection reveals that the predicted jet is still very slightly thicker than the experimental jet and pointing slightly downwards. Evidence of this can be

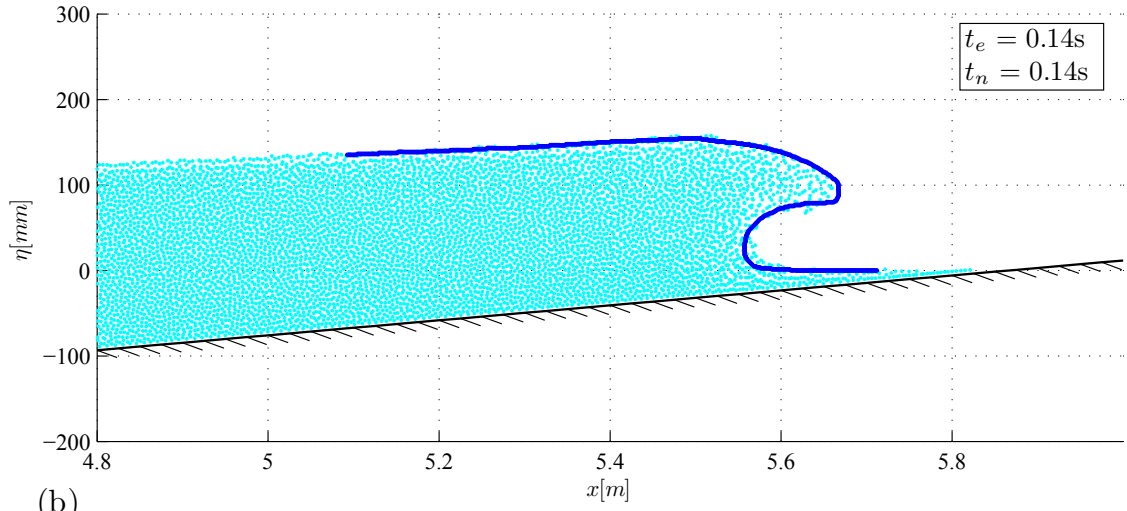
seen by the few particles which appear to be below the bottom of the experimental jet. Overall, the higher resolution has improved the predicted jet profile considerably. In particular, the wave now appears to have the degree of forward momentum necessary to project the jet forwards almost as far as in the experiment. Indeed, it is possible that by increasing the SPH resolution still further, the numerical and experimental jet profiles may provide an even closer match. Careful comparisons at the back wall of the developing cavity indicate that there are a small number of particles advancing ahead of the experimental observations. However, at this resolution there are fewer of them and they are confined to the bottom of the region. As indicated previously, this could be due to the enhanced roll of surface tension provided by the sidewalls of the experimental flume rather than necessarily resulting from errors in the two-dimensional SPH model.

Figure 5.16 shows a photograph of the experiment in (a) and a comparison between the numerical and experimental waves in (b) as the waves begin to touch down on the bed slope in front. At this stage, the numerically predicted water surface profile remains a good representation of the experimental observations. At this resolution, the location of the jet as it touches down on the bed slope is more accurately predicted compared to the lower resolution simulation. Indeed, it is only the tip of the jet which has turned a little too far downwards. The photograph appears to show some water splashing into the cavity immediately behind the jet, which has not been captured by the visualisation process. However, this effect appears not to occur on the side walls of the flume and, perhaps, emphasises the local effects of surface tension. It is possible that the reason the tip of the SPH jet appears to be directed more downwards (rather than forwards) is because the SPH jet is more representative of the cross-sectional centre of the experimental jet rather than the profile defined on the sidewall closest to the camera. Although there remain a few SPH particles ahead of the back wall of the experimental cavity, the overall size of the cavity has increased relative to the lower resolution predictions, improving the quality of the numerical predictions in this respect.

Increasing the resolution has improved the description of the forward momentum of the jet as the wave overturns. This suggests that the lack of detail possible in the movement of the fluid in a lower resolution calculation inhibited this motion, perhaps due to the viscosity being relatively too high. It is therefore possible that reducing any artificial numerical viscosity, adjusting parameters in the turbulence model or, indeed,



(a)

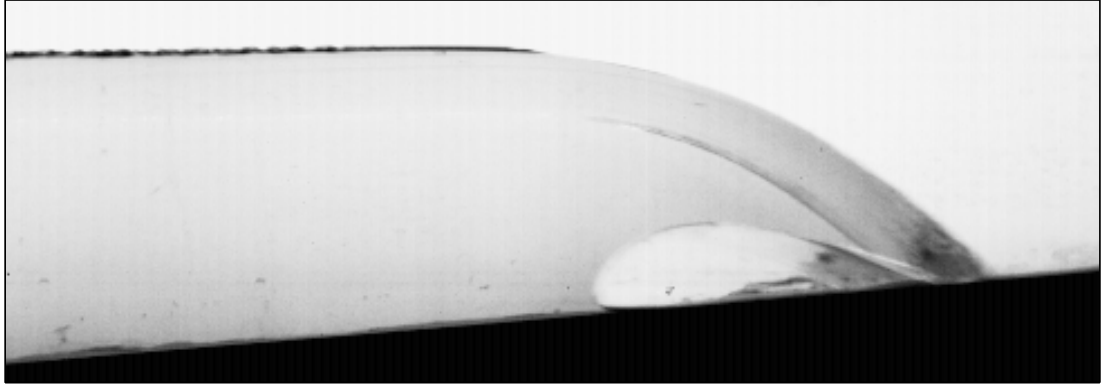


(b)

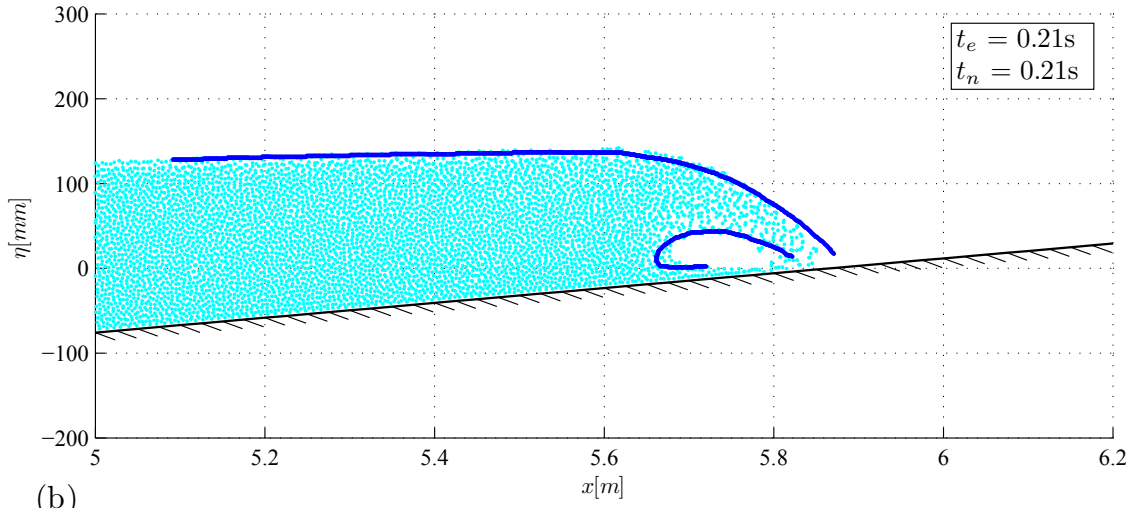
Figure 5.15: Visualisation of a solitary wave with $H/d = 0.3$ as the wave starts to overturn: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations: — laboratory observations, • SPH model.

employing a more complex turbulence model, may be beneficial in terms of increasing the forward momentum and bringing the SPH results closer to the experimental results.

Given that the results of the numerical simulation up to this point have provided very good quantitative comparisons with the experiment, the simulation has been continued throughout the whole breaking process. Once the overturning jet has reached the slope in front, the impact of the water on the slope results in high pressure and a break down of the water surface profile, which up until now has been relatively smooth. Figure 5.17



(a)

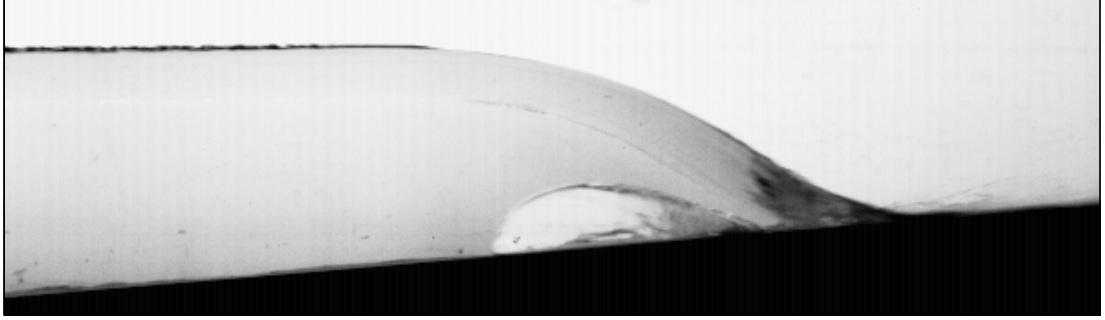


(b)

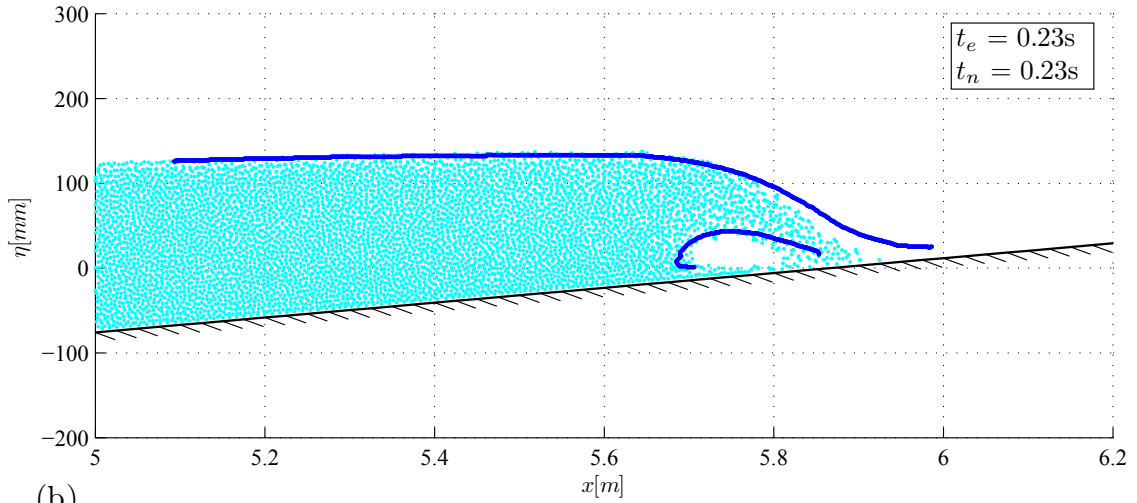
Figure 5.16: Visualisation of a solitary wave with $H/d = 0.3$ as the overturning jet nears impact with the bed slope ahead: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations: — laboratory observations, • SPH model.

shows the beginning of this process, a photograph of the experiment provided in (a) and a comparison between the numerical and experimental results in (b). This is the first point at which water from the back of the experimental cavity starts to push forwards into it. It is at this point that the forward momentum and gravitational forces begin to overcome the surface tension which has previously sustained the rounded cavity. At this stage of the breaking process, the predicted and observed water surface profiles remain in good agreement, although the tendency for the SPH jet to come down before the

experimental jet is more apparent, the experimental jet impacting slightly further up the slope. However, as with the previous comparison, the photograph appears to show water from the cross-sectional centre of the flume falling behind the water in the jet on the nearest sidewall of the flume. The centreline profile may therefore be closer to the SPH predictions.



(a)



(b)

Figure 5.17: Visualisation of a solitary wave with $H/d = 0.3$ as the wave starts to impact with the bed slope: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations: — laboratory observations, • SPH model.

As the process of wave breaking continues, the cavity gradually gets filled in with turbulent water both from the back of the cavity and the impacting jet. Meanwhile, water from what was the front of the jet begins to run up the slope with some turbulent splashes of water rebounding off the slope. The beginning of this run-up phase is shown

in Figure 5.18a with a photograph of the experiment and Figure 5.18b with a comparison between the SPH simulation and the laboratory observations. The increased forward momentum in the experimental observations has led the front of the run-up tongue to advance considerably further up the slope. Setting this aside, the shape of the water surface and the beginning of some water splashing up at the front of the tongue appear in good agreement. Considering the position of the cavity, or the remnants thereof, it is clear that this is further forwards in the experimental observations. This is consistent with the front of the run-up tongue being further up the slope. However, the size of the remaining cavities are broadly similar.

At this stage in the breaking process, it starts to become harder to make good quantitative comparisons. Not only does the turbulent nature of the flow result in somewhat random splashes of water, but three-dimensional effects become far more relevant. In particular, it is probable that small splashes of water in the experiment will only partially cover the horizontal cross-section of the flume. In contrast, the two-dimensional nature of the numerical simulation implies that all the turbulent flow, and any splashes of water, cover the entire width of the flume. This also emphasises the inappropriate nature of using a turbulence model in a two-dimensional simulation, turbulence being an inherently three-dimensional phenomenon. Another limitation of the SPH model may also be contributing to differences in the cavity at this stage and beyond. This relates to the fact that the SPH formulation is a single-phase model and, in reality, it is possible that the air provides some cushioning within the cavity, helping to sustain its shape for longer. Indeed, Colagrossi & Landrini (2003) conducted simulations of a backward plunging breaker, which entrapped air in a cavity, using a two-phase SPH model and found evidence to suggest that the inclusion of air in the model improved the quality of the predicted results.

The final comparison shows the continuation of the breaking procedure as the water from the overturned wave runs further up the slope. Once again, this is presented as a photograph of the experiment in Figure 5.19a and as a comparison between the numerical and experimental results in Figure 5.19b. This part of the breaking process results in a considerable upwards splash of water rebounding off the slope and a reduction in the size of the cavity. Although the SPH profile of the remaining jet and the developing run-up tongue are both similar in shape to the experiment, the numerical predictions

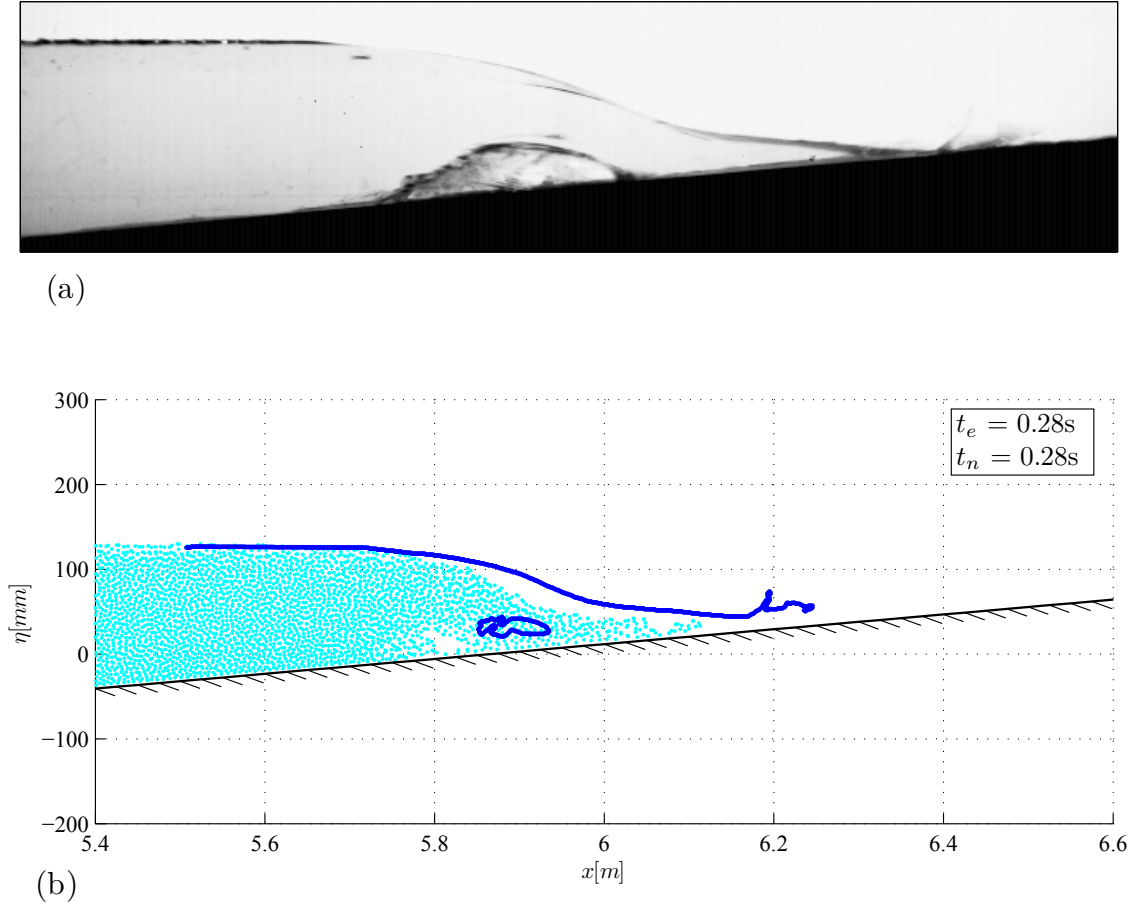


Figure 5.18: Visualisation of a solitary wave with $H/d = 0.3$ as the flow begins to run up the bed slope: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations: — laboratory observations, • SPH model.

lag behind the experimental observations and the splash of water rebounding off the bed slope is smaller. This is consistent with the reduced forward momentum seen in the SPH wave at previous stages of the breaking process and perhaps suggests a small reduction in the energy in comparison to the experimental observations. However, the photograph also shows that the profile of the splash captured by the visualisation technique does not stretch across the full width of the flume and therefore contains less water than a direct comparison between the experiment and numerical predictions suggests. Three-dimensional effects have clearly become important at this stage. As a result, it must be said that whilst qualitatively the SPH predictions are in good agreement with the laboratory observations, a true quantitative comparison is no longer possible.

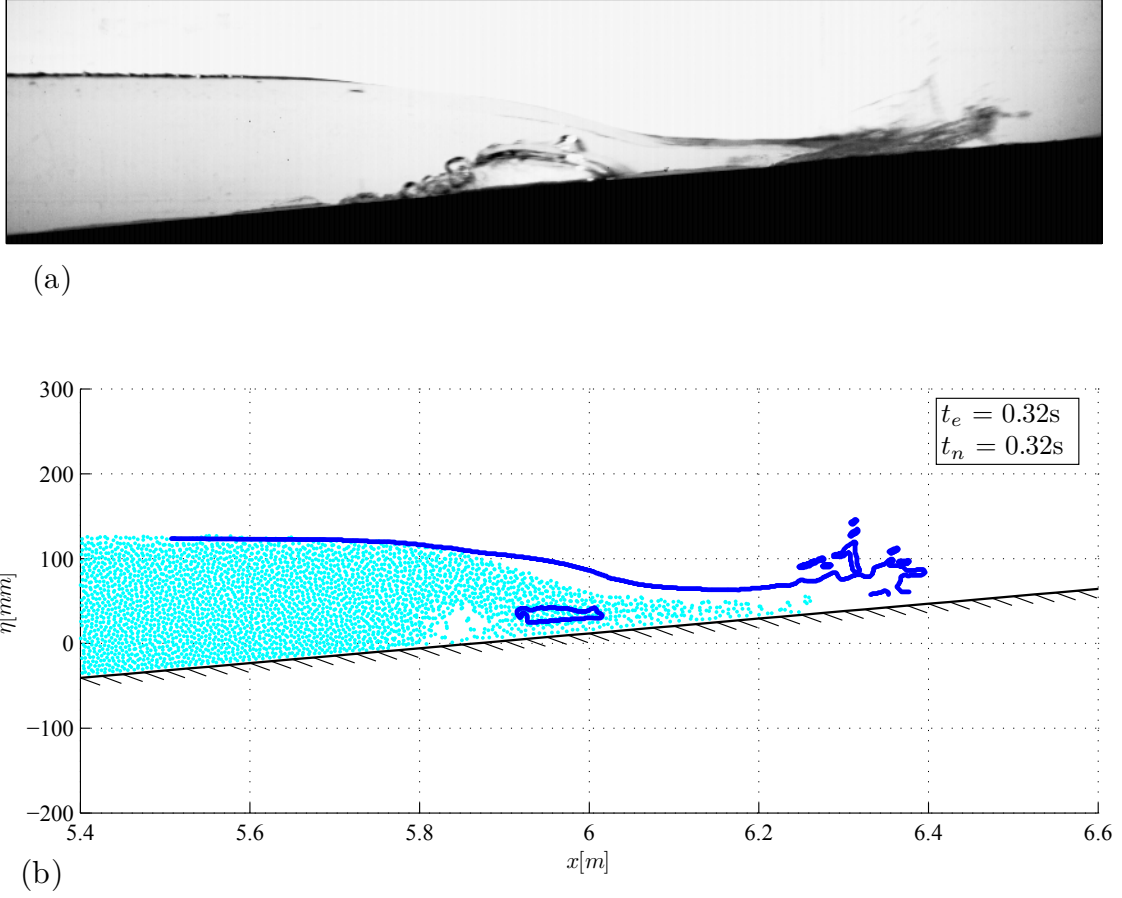


Figure 5.19: Visualisation of a solitary wave with $H/d = 0.3$ as the flow continues to run up the bed slope: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations: — laboratory observations, • SPH model.

After this last comparison, the water continues to run up the slope, with the water surface levelling and smoothing out, to produce a thin run-up tongue. Meanwhile, the cavity eventually becomes completely filled in. As the run-up tongue moves up the slope, it gradually thins out further as it loses momentum and approaches its maximum run-up height. A comparison of maximum run-up is given in Table 5.1. This data clearly suggest that the experimental run-up tongue travelled further up the slope than the numerical predictions, which only reached 80% of the observed run-up height. The time at which this maximum run-up occurred was not recorded experimentally, but the numerical results give an indication of how long it takes for the run-up to reach its maximum height.

	Maximum run-up height above still water level [mm]	x -location of maximum [m]	Time of maximum [s]
SPH model	324	9.50	2.20
Laboratory observations	400	10.43	unknown

Table 5.1: Maximum run-up on a 5° slope for a solitary wave with $H/d = 0.3$, comparison between results produced with a high resolution SPH simulation and laboratory observations.

Although there is a large discrepancy between the observed and predicted run-up, this was not entirely unexpected for several reasons. First, at the maximum run-up, the SPH predicted run-up tongue becomes only one particle thick at the tip. Even further down, where it is four or five particles thick, this is insufficient to give a high level of accuracy. However, this problem is not restricted to the present SPH model. The run-up tongue naturally continues until it is very thin, so it would be difficult for any numerical model to simulate it accurately right up to its maximum extent. One possibility for improving the SPH results would be to use an adaptive particle resolution in this region. However, this brings additional complications to the model in terms of the conservation properties and has not been considered herein.

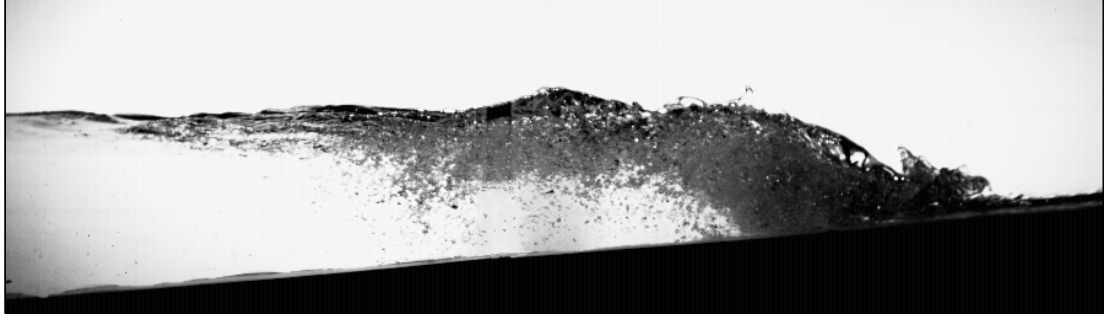
Second, both experimentally and numerically the boundary layer becomes increasingly important in the thin run-up tongue, and the properties of the boundary layer depend strongly on the fluid viscosity and the bed roughness. The roughness of the experimental slope was not measured, although it was assumed to be hydraulically smooth. However, as discussed in § 5.3.3, it proved difficult to specify the turbulent boundary condition precisely in the SPH model. It is therefore likely that the viscous effects arising in the SPH boundary were over-estimated in the thin run-up tongue. This will be a major factor in accounting for the difference in the maximum run-up height. During the development of the present SPH model, simulations were carried out with and without the turbulent boundary condition and with various boundary and viscosity models. Some of these model variations resulted in the run-up tongue continuing well beyond that observed in the experiment. Given that the maximum run-up depends so strongly on the boundary conditions on the slope, it did not seem appropriate to make further

comparisons.

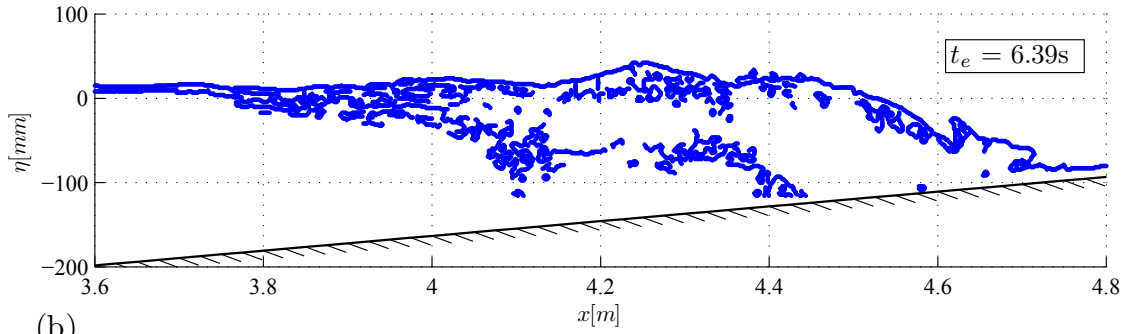
After reaching its maximum run-up height, the water begins to wash back down the slope. As it does so, its speed greatly increases, resulting in a hydraulic jump forming as the supercritical flow meets the subcritical flow below. The hydraulic jump is transitory, lasting until all the water that ran up the slope has washed back down and joined the almost stationary flow at the bottom. Since the hydraulic jump is a highly turbulent feature and three-dimensional effects are undoubtedly important, it is difficult to make quantitative comparisons for this part of the wave breaking process.

A photograph of the hydraulic jump observed in the experimental study is shown in Figure 5.20a. The flow is clearly three-dimensional and a lot of air is entrained in the flow in the form of small bubbles. Although the SPH model can readily describe cavities and small air pockets in the flow, it is unable to capture these tiny bubbles as they are considerably smaller than the particle resolution.

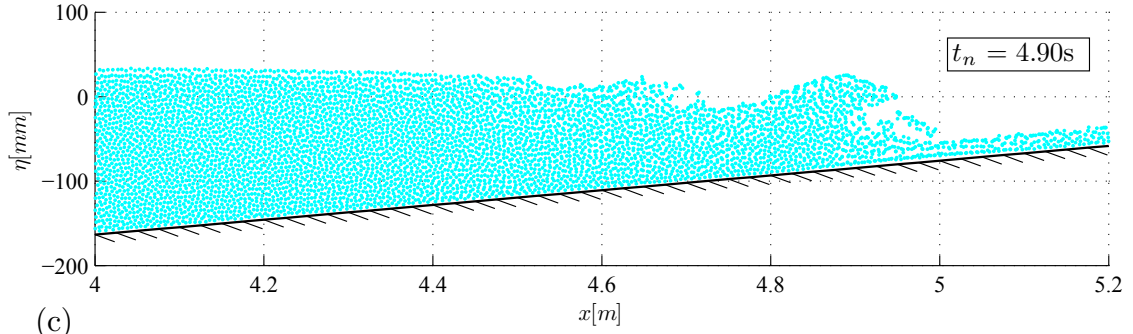
Figure 5.20b shows a processed image of the observed hydraulic jump based upon the flow visualisation technique described previously. This is followed by an image of the numerically predicted hydraulic jump in Figure 5.20c. Both images were taken whilst the hydraulic jump was at its strongest, although it is virtually impossible to properly compare the surface profiles as they are constantly evolving. The time in the two images is very different and the location of the jump also differs by about 40cm. However, these were not expected to be the same for several reasons. First, the images simply show a snap-shot from both the experiment and the SPH simulation; the hydraulic jump lasts for approximately four seconds in both and during that time the front location moves around significantly. Second, the experimental and numerical flumes are of different lengths, so any reflected waves will interact with the hydraulic jump at different times. Indeed, the depth of the subcritical water at the time shown in Figure 5.20 is greater in the SPH simulation than in the experiment. Third, due to differences in the run-up tongue described above, the quantity of water available to wash back down the slope as supercritical flow will be different between the experimental observations and the numerical predictions. Nevertheless, given that the hydraulic jump formed in a similar location on the slope, and over a similar period of time, it can be argued that the SPH predictions are still in good qualitative agreement with the laboratory observations.



(a)



(b)



(c)

Figure 5.20: Visualisation of the hydraulic jump resulting from a solitary wave with $H/d = 0.3$ washing back down the bed slope: (a) photograph of the experiment, (b) laboratory observations, and (c) results produced with a high resolution SPH simulation: — laboratory observations, • SPH model.

Case 2: solitary wave with $H/d = 0.45$

In order to produce a solitary wave with $H/d = 0.45$ in the laboratory wave flume it was necessary to reduce the water depth to $d = 0.4\text{m}$. As the size of the wave and the water depth increases, the movement of the wave maker necessary to produce the required

wave (specified in terms of H/d) must increase, the target being a high quality solitary wave with minimal oscillations in the tail of the wave. The range required for a wave with $H/d = 0.45$ in a water depth of $d = 0.5\text{m}$ was slightly larger than the laboratory wave maker could achieve. Accordingly, the water depth was reduced to $d = 0.4\text{m}$ for this wave case. In practice, the desired wave could have been generated in a water depth of $d = 0.5\text{m}$ by using a shorter range of motion, but this would have increased the amplitude of the trailing disturbances, thereby decreasing the accuracy of the results. It was therefore preferable to reduce the water depth, which in turn reduces the required motion of the wave maker.

As for *Case 1*, the wave gauge data was used to compare the profiles at the toe of the slope and ensure that the desired wave height, H , was produced both numerically and experimentally. Data confirming this are shown in Figure 5.21. These results show that the numerical predictions exactly overlaid the experimental data, except for some minor discrepancies in the trailing wave.

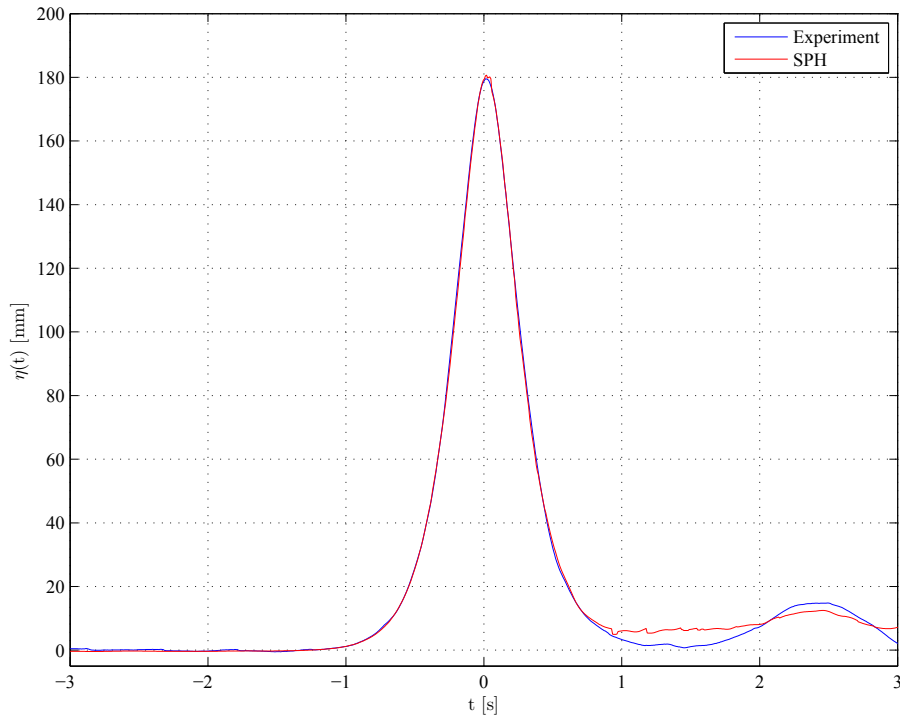


Figure 5.21: Time histories of the water surface elevation, $\eta(t)$, during solitary wave propagation with $H/d = 0.45$. Comparisons between laboratory observations and high resolution SPH predictions at a wave gauge located 0.15m before the toe of the bed slope. — laboratory observations — SPH model.

In Figure 5.22 the numerical and experimental waves are again compared at the breaking point. The time taken for the laboratory generated wave to travel from the toe of the slope to the breaking point was 1.51s. In contrast, the numerically predicted wave took 1.60s, corresponding to a 6% error in the propagation time between these two points. This difference is of a similar magnitude to that observed in *Case 1* and discussed above.

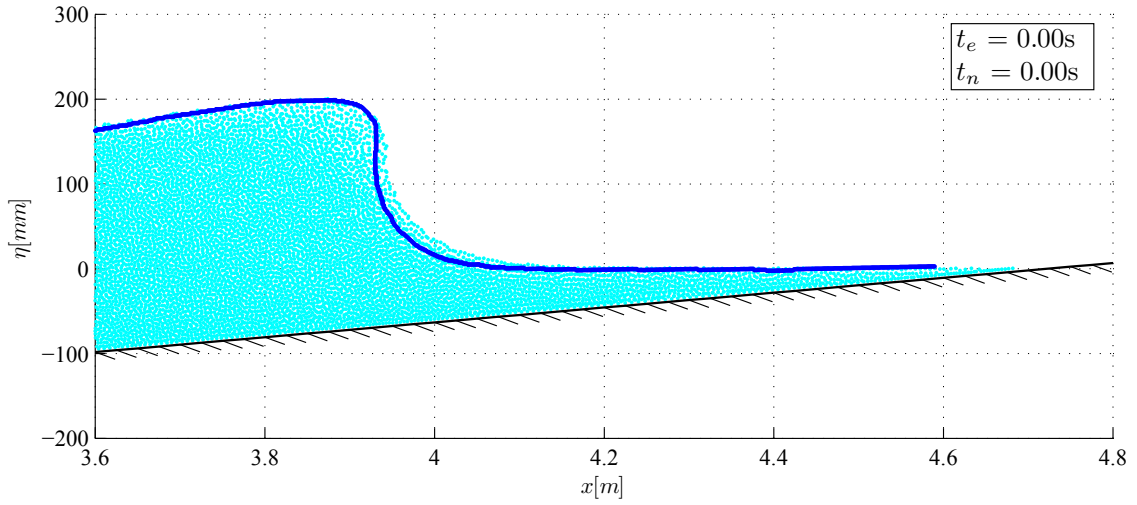


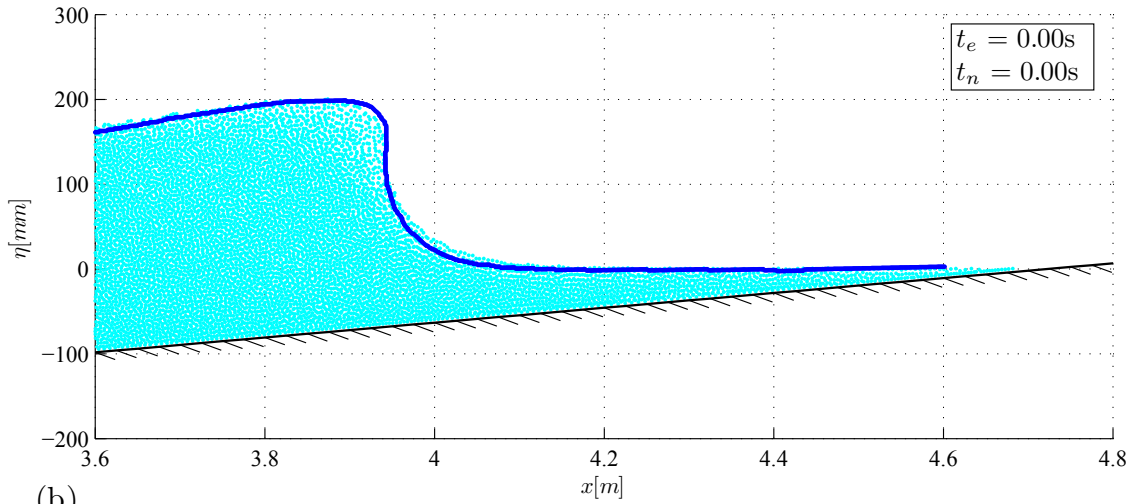
Figure 5.22: Visualisation of a solitary wave with $H/d = 0.45$ at the breaking point. Comparison between results produced with a high resolution SPH simulation and laboratory observations: — laboratory observations, • SPH model.

Comparisons between the observed and predicted water surface profiles at the breaking point (Figure 5.22) are very good; the location of the wave front is much closer to the experimental results than in *Case 1*. Nevertheless, the numerical wave front is still approximately 12mm further forward. In order to facilitate comparisons of the entire breaking process, and to be consistent with earlier comparisons of *Case 1*, the experimental results have been shifted forwards 12mm in the sequence of comparisons that follow. Figure 5.23a provides a photograph of the experimental observations, while a comparison between the numerical results and the shifted laboratory observations is presented in Figure 5.23b. In this case, the water surface profile predicted by the SPH model is in excellent agreement with the experimental observations.

As the wave starts to overturn, the experimental and numerical profiles were again compared. Figure 5.24a provides a photograph of the experiment and Figure 5.24b a



(a)

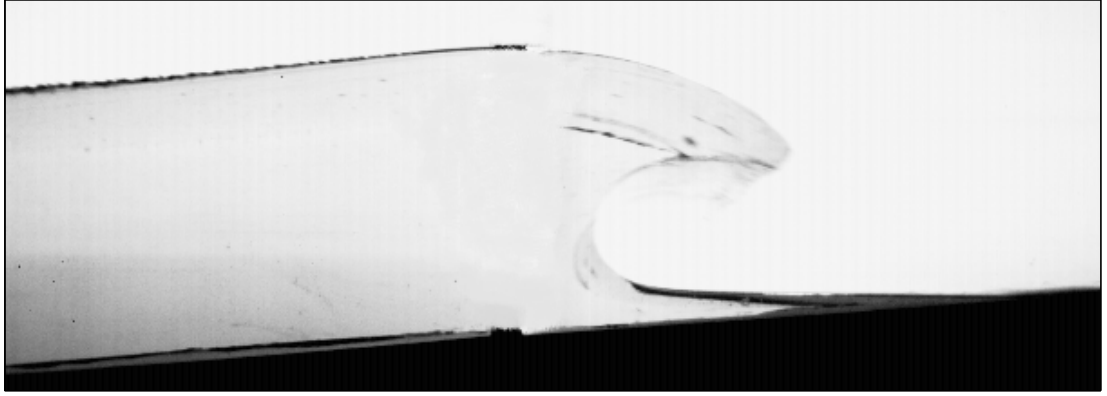


(b)

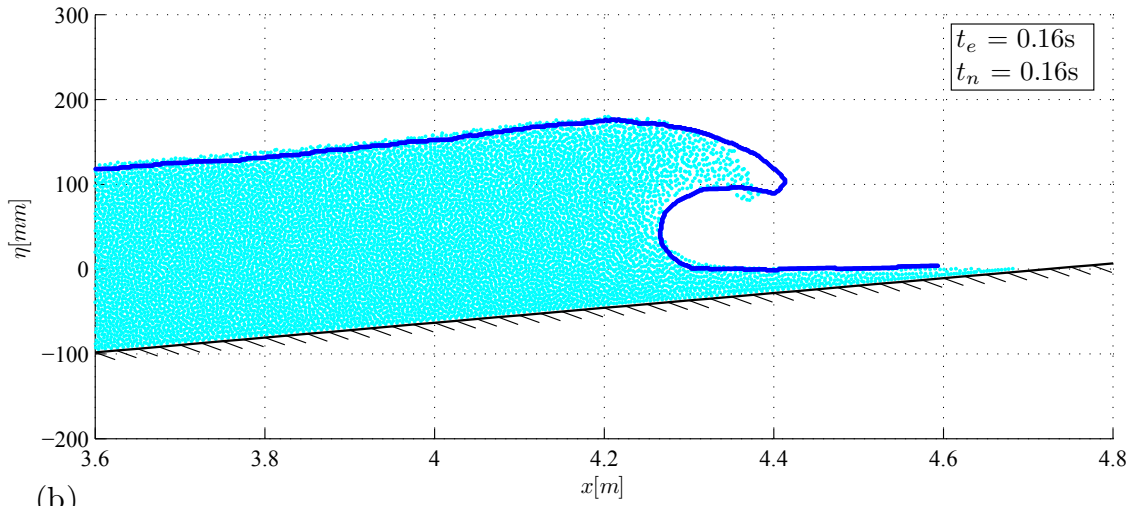
Figure 5.23: Visualisation of a solitary wave with $H/d = 0.45$ at the breaking point: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations which have been spatially aligned with the SPH results: — laboratory observations, • SPH model.

comparison of the SPH simulation and the laboratory observations. At this stage, the predicted back wall of the developing cavity is in very good agreement with the experimental observations. Furthermore, both the numerical and experimental overturning jets are of a similar thickness and length, and the shape in the numerically predicted jet tip provides a good description of the experimental jet tip. However, although aspects of the numerical jet are well defined, it is clearly turning downwards before the experimental jet, which is still predominantly projecting forwards at this time. As was

suggested for the lower resolution simulation of the solitary wave with $H/d = 0.3$, it is possible that the viscosity formulation within the numerical model leads to an over prediction of the viscous effects. This, in turn, may contribute to the numerical wave lacking the forward momentum necessary to create the degree of projection observed in the experimental wave. This effect appears to be more important in the larger wave, perhaps because there is more momentum needed to drive it forwards.



(a)



(b)

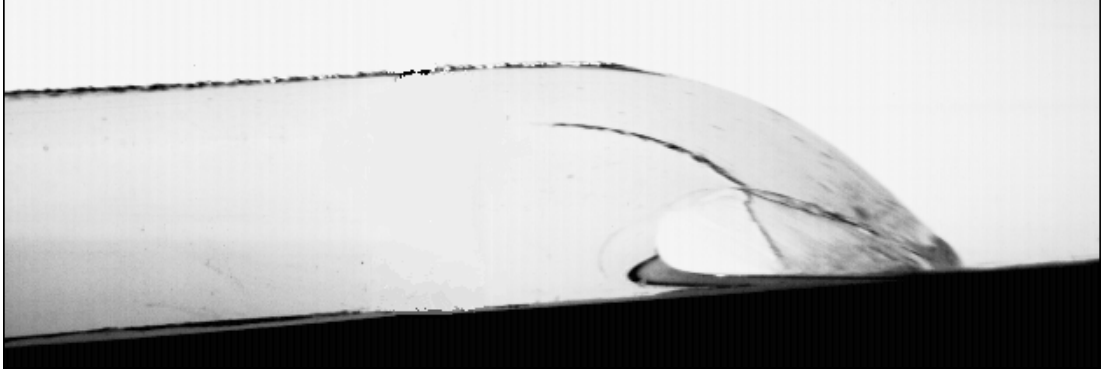
Figure 5.24: Visualisation of a solitary wave with $H/d = 0.45$ as the wave starts to overturn: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations: — laboratory observations, • SPH model.

Figure 5.25a shows a photograph of the experimental jet as it touches down on the

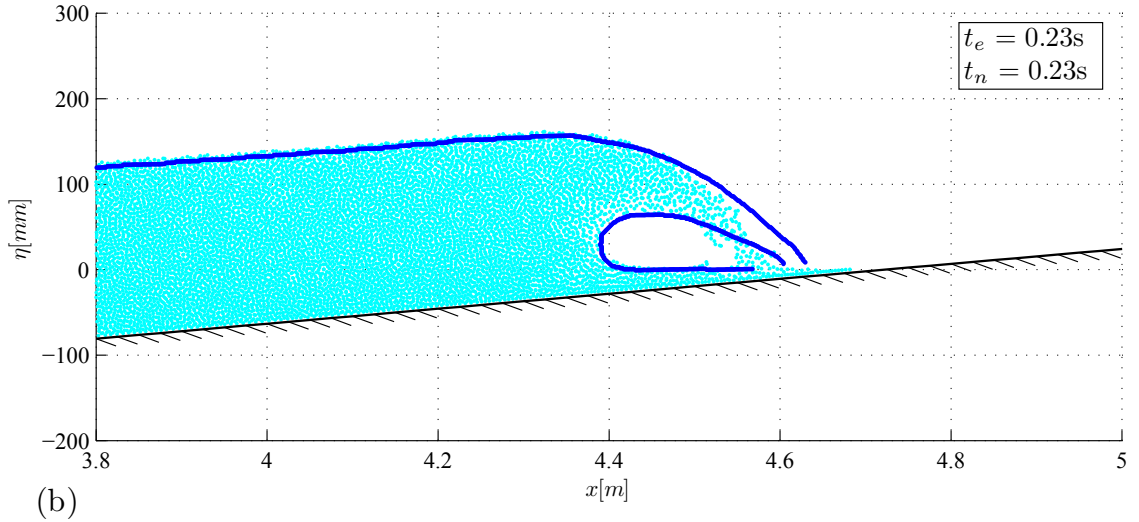
slope ahead. This happened at the same time in the numerical simulation as in the experiment, a comparison between the results being presented in Figure 5.25b. The back wall of the cavity in the SPH simulation has remained in the correct position when compared to the experimental observations, the resulting shape of the cavity being in excellent agreement. However, the discrepancies seen at the previous stage have continued, and as a result, the numerical jet has touched down on the bed slope earlier than the experimental jet. Whereas the experimental jet has just cleared the still water lying in front of the over-turning jet, the numerical jet has impacted into this water at the point where the water depth is approximately 13mm deep.

The consequences of this difference can be seen in the following comparison, shown as a photograph of the experiment in Figure 5.26a and as a comparison between the numerical and experimental results in Figure 5.26b. Within the numerical calculations, the SPH particles from the impact have bounced off the still water, causing water to be splashed upwards at a similar angle to the approaching jet. This type of solitary wave breaking is similar to that described in the laboratory experiments of Khayyer *et al.* (2008), when carrying out studies looking into solitary waves breaking on a slope with an incompressible SPH method, and also observed by Issa & Violeau (2009) when comparing to the experiments of Li & Raichlen (2003). In contrast, the impact from the experimentally observed wave jet in the present case was very similar to that observed in *Case 1*, the water splashing off the bed slope being directed forwards and up the slope. The SPH predicted wave is breaking in a realistic manner, with qualitatively the same features as the experimentally observed wave. However, the tendency for the SPH predictions to describe a wave jet which turns downwards sooner than that observed experimentally, appears to have led the SPH wave to break in the manner of experimental waves with a slightly different setup; for example, this may be achieved by using a slightly different height to depth ratio.

As the breaking process continues, the cavity begins to fill with water and the impacting jet starts to run up the slope. This is shown in a photograph of the experiment given in Figure 5.27a and as a comparison between the SPH simulation and laboratory observations in Figure 5.27b. The water driven into the cavity is remarkably similar in both the numerical and the experimental results. However, it is also clear that the experimental jet has propagated a lot further up the slope. Furthermore, the splash



(a)

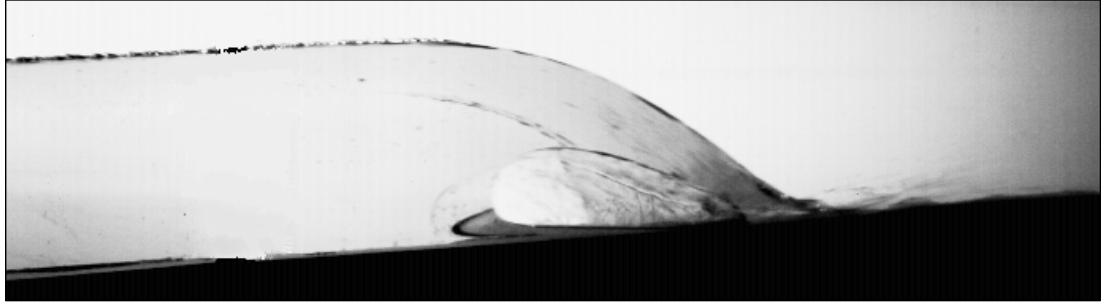


(b)

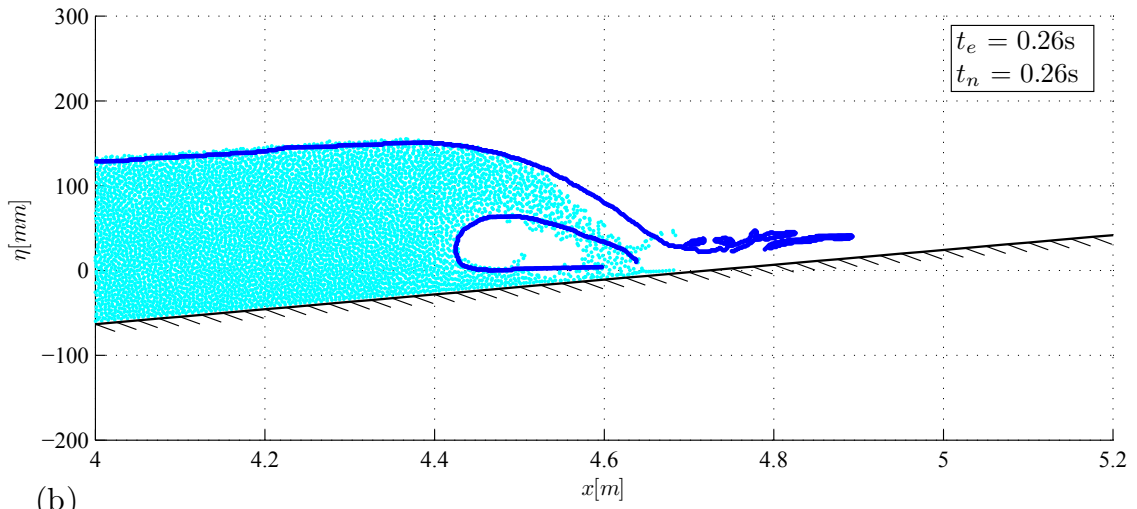
Figure 5.25: Visualisation of a solitary wave with $H/d = 0.45$ as the overturning jet nears impact with the bed slope ahead: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations: — laboratory observations, • SPH model.

up from the numerical jet impacting on the bed slope is travelling more upwards than forwards, and in this sense is very different to the experimental observations.

A photograph of the experiment as the wave progressed further up the slope is given in Figure 5.28a and a comparison between the SPH simulation and the laboratory observations is presented in Figure 5.28b. In comparing these results, it can be seen that the numerically simulated breaking wave has progressed further up the slope with little change in the profile, other than some reduction in the size of the cavity. In



(a)



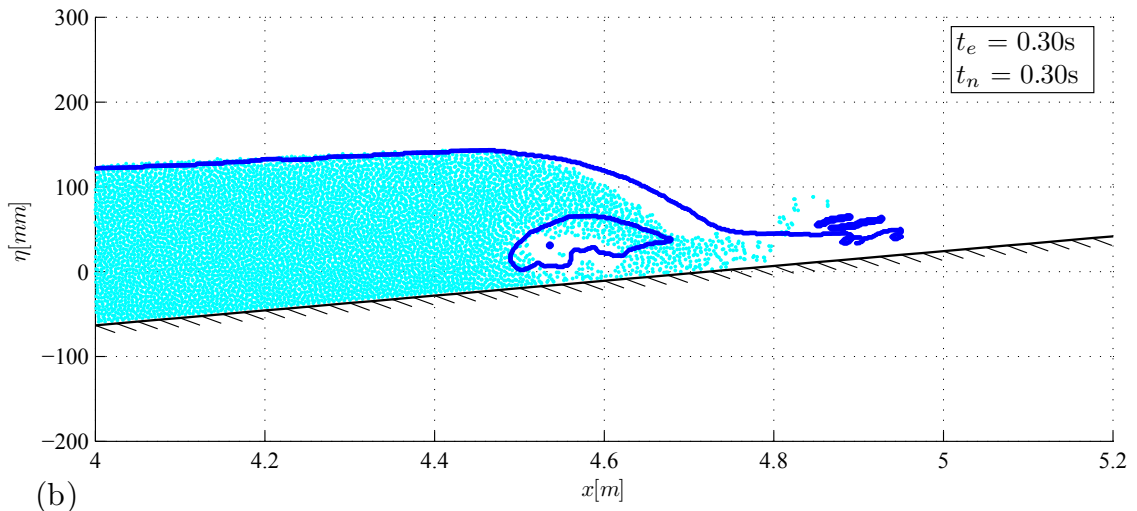
(b)

Figure 5.26: Visualisation of a solitary wave with $H/d = 0.45$ as the wave starts to impact with the bed slope: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations: — laboratory observations, • SPH model.

contrast, the experimental wave now has a large quantity of water splashing upwards at the front of the developing run-up tongue and the cavity has started to break up and move towards the water surface. This movement of the cavity continues, like a rising air bubble, until it breaks through the top of the water surface, as can be seen in a photograph of the experiment in Figure 5.29a and as a comparison of the numerical and experimental results in Figure 5.29b. This is accompanied by an increase in the quantity and height of the water splashing up at the head of the run-up tongue. At this stage, the front of the SPH predicted run-up tongue has more or less flattened out. The reduced forward momentum in the numerically predicted jet during the earlier stages



(a)



(b)

Figure 5.27: Visualisation of a solitary wave with $H/d = 0.45$ as the cavity starts to fill with water: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations: — laboratory observations, • SPH model.

of the wave breaking seems to have translated into a reduction of energy, momentum and turbulence at this stage of breaking in comparison to the experimental observations. The cavity has kept its size from the last comparison, but has moved forwards rather than upwards and, unlike the experimental cavity, has not yet broken through the water surface.

Many of the differences between the SPH simulation and the experimental observations in the latter stages of the wave breaking process can be attributed to the SPH wave breaking onto the still water in front rather than the dry slope. However, it also seems that the ferocity of the experimental wave at this increased height to depth ratio has led

to an increase in turbulent mixing once the jet has impacted on the slope. Unfortunately, the SPH model appears unable to capture this effect. There are several possible reasons for this. First, many of the small bubbles in the experimental observations are much smaller than the SPH particles. As a result, these effects cannot be resolved. Second, in its present form, the SPH model does not include particles representing the air, and this may affect the degree of air entrapment. Third, the SPH simulation is two dimensional and so cannot fully represent the three dimensional turbulent features. In effect, the turbulence model may be insufficient for the flow encountered in this simulation.

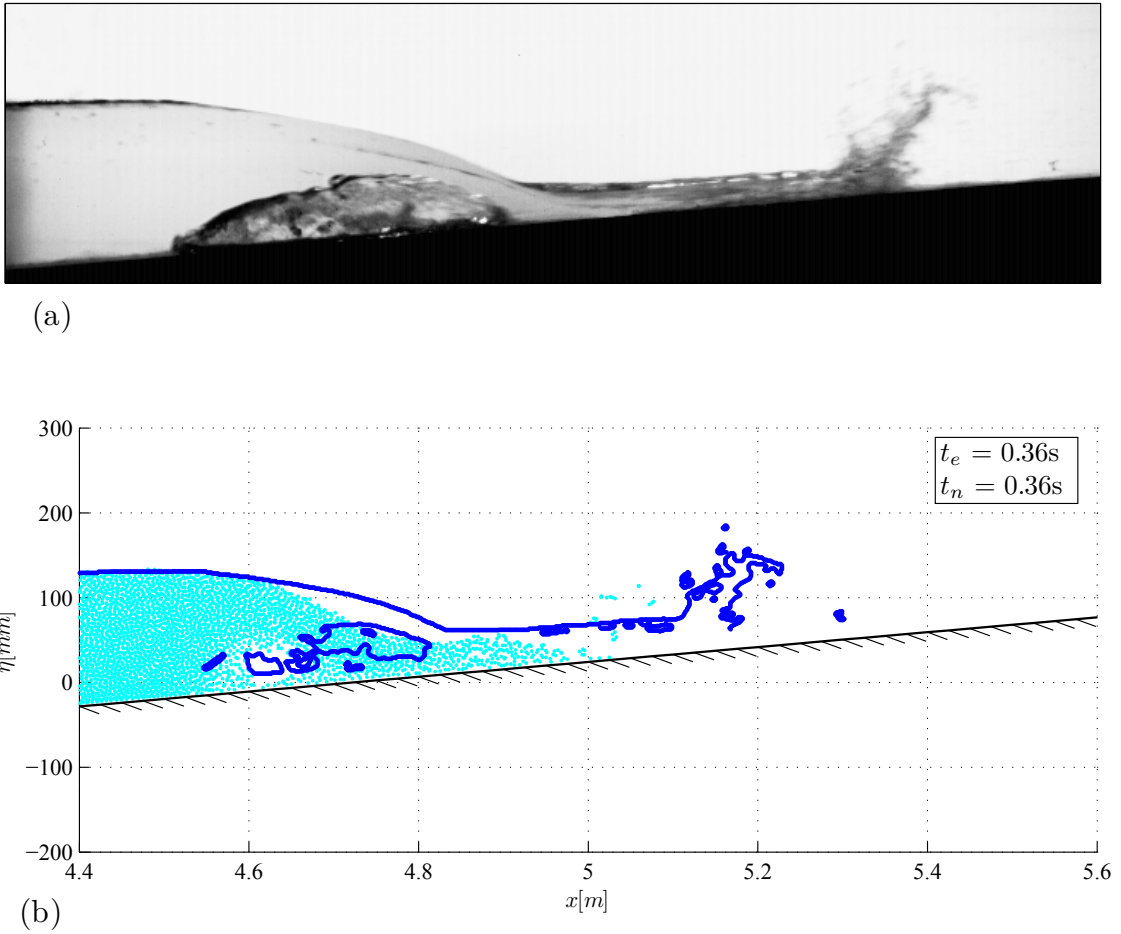


Figure 5.28: Visualisation of a solitary wave with $H/d = 0.45$ as the flow begins to run up the bed slope: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations: — laboratory observations, • SPH model.

Data describing the maximum run-up for this wave case are given in Table 5.2.

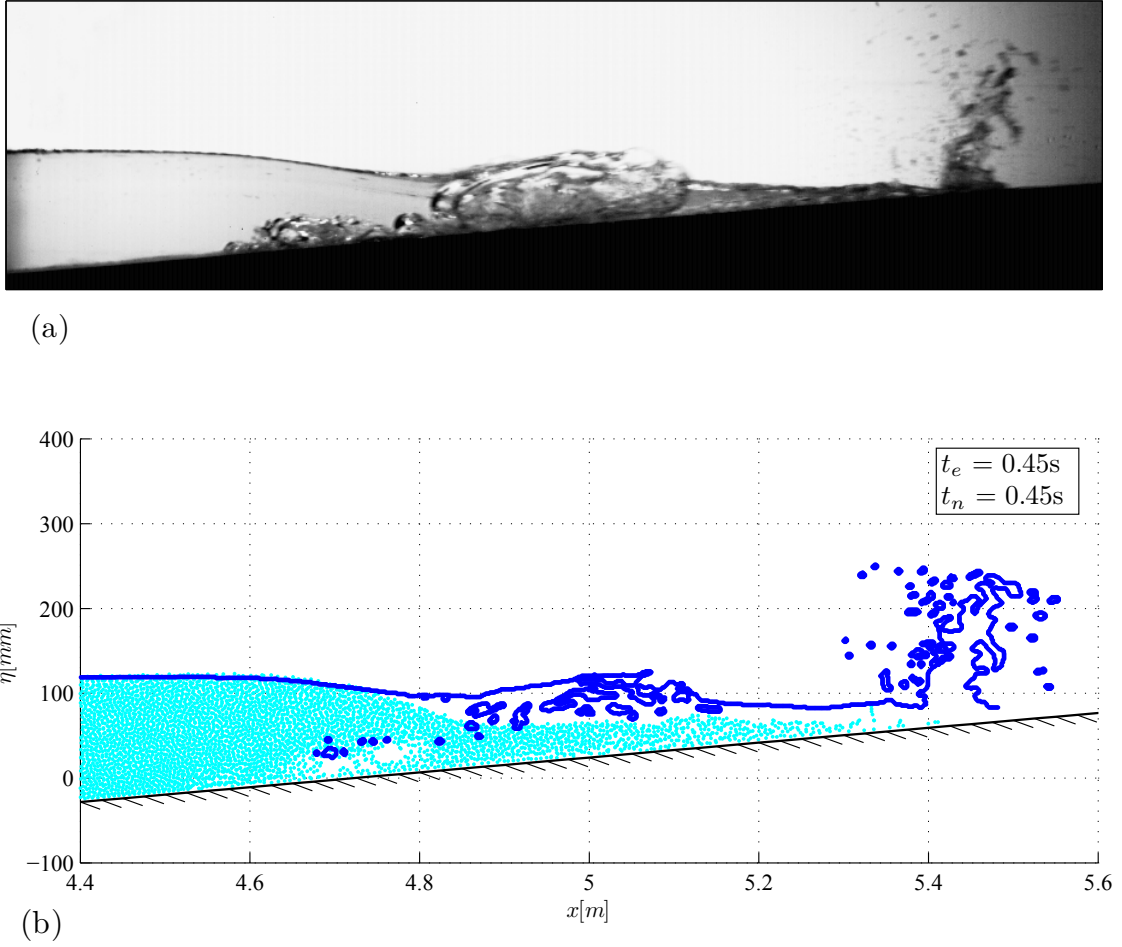


Figure 5.29: Visualisation of a solitary wave with $H/d = 0.45$ as the flow continues to run up the bed slope: (a) photograph of the experiment and (b) comparison between results produced with a high resolution SPH simulation and laboratory observations: — laboratory observations, • SPH model.

As in *Case 1*, the maximum run-up was substantially greater in the experiment; the numerically predicted maximum run-up reached around 80% of the observed value. This difference is very similar to that observed in *Case 1*, the explanation for this difference being as described earlier.

In considering the hydraulic jump caused by the supercritical wash-down, images of the hydraulic jump have again been selected whilst it is at its strongest, a similar selection having been made in respect of both the numerical predictions and the experimental observations. The full turbulent processes can be seen in the photographic image of the experiment shown in Figure 5.30a. The corresponding processed image is given

	Maximum run-up height above still water level [mm]	x -location of maximum [m]	Time of maximum [s]
SPH model	357	8.74	2.46
Laboratory observations	431	9.65	unknown

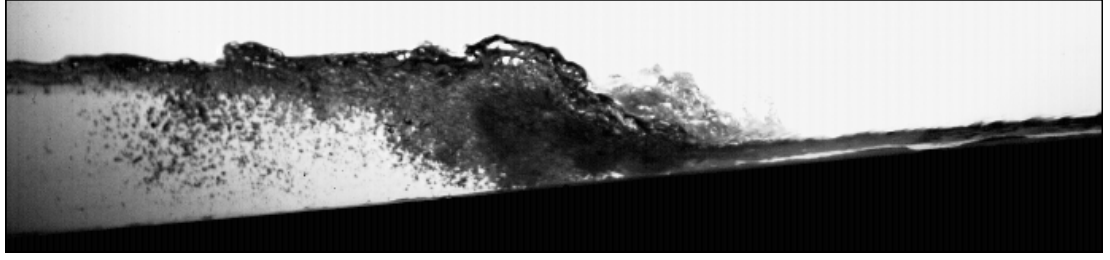
Table 5.2: Maximum run-up on a 5° slope for a solitary wave with $H/d = 0.45$, comparison between results produced with a high resolution SPH simulation and laboratory observations.

in Figure 5.30b, while Figure 5.30c provides the results of the SPH simulation. These results are very similar to those relating to *Case 1*, (Figure 5.20), the time and location of the hydraulic being notably different for reasons given previously. In this case, both the observed and the predicted hydraulic jumps lasted for approximately 5s. Once again, it can be concluded that the SPH predictions are in good qualitative agreement with the laboratory observations.

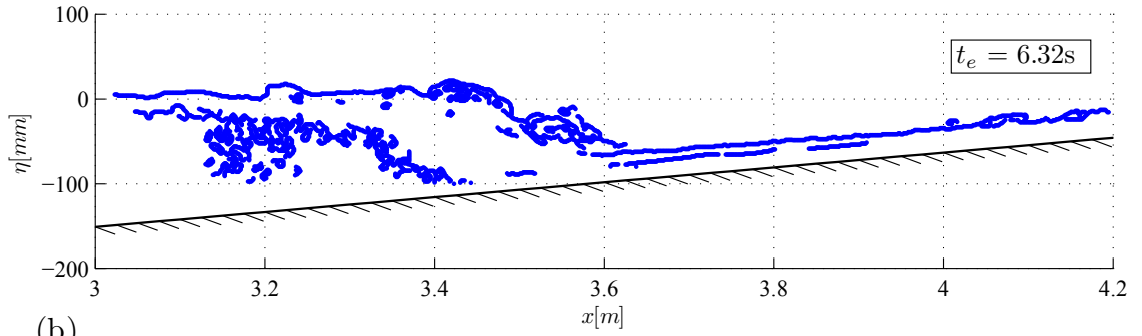
Velocity and pressure in breaking solitary waves

This section discusses results taken from the SPH simulations of the breaking solitary waves, providing descriptions of the water particle velocity and pressure during the breaking process. They are not compared with experiments, as it is difficult to obtain such measurements in the laboratory, but are included to highlight the ease with which such properties can be captured numerically. SPH modelling is particularly useful for describing these properties after the wave has broken, when other numerical methods are either inappropriate or fall down. The results arising from the two cases described in the previous sections of this chapter have shown that the SPH results are in excellent agreement with experimental observations as the wave begins to break, but show some discrepancies during the breaking process. The primary source of this difference lies in the angle of the overturning jet as it touches down on the bed slope. Despite these differences, the SPH profiles throughout the breaking process are realistic and the overall features of the breaking wave are captured at each stage. This earlier validation adds to the credibility of the velocity and pressure results shown herein.

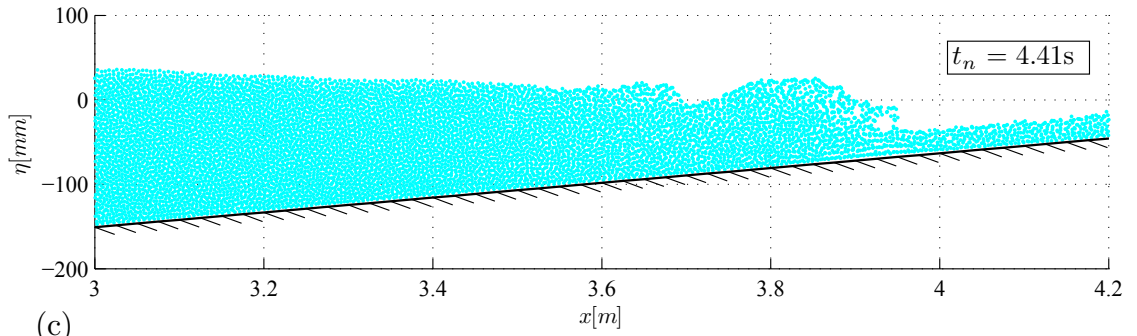
Figures 5.31 and 5.32 show velocity vectors describing the solitary waves from *Case 1*



(a)



(b)



(c)

Figure 5.30: Visualisation of the hydraulic jump resulting from a solitary wave with $H/d = 0.45$ washing back down the bed slope: (a) photograph of the experiment, (b) laboratory observations and (c) results produced with a high resolution SPH simulation: — laboratory observations, • SPH model.

($H/d = 0.3$) and *Case 2* ($H/d = 0.45$), respectively, as they approach and move through the breaking point. The velocity vectors are coloured according to the particles' speeds, $|u|$, non-dimensionalised with respect to the wave speed, c , which was calculated from the numerical results. The images show that as the waves approach the breaking point, $|u|/c$ is less than one. As the waves move through breaking point, $|u|/c$ increases and

finally $|u|/c$ exceeds one as the wave begins to break. This corresponds with the well-established notion that waves begin to break when the water particle velocities in the wave crest exceed the wave speed. It can also be seen in these images that behind the wave front the particle velocity is almost uniform with depth. This is exactly as expected according to the theory outlined in § 4.2.1. Finally, the water in front of the breaking wave remains virtually undisturbed, as was observed in the experiments.

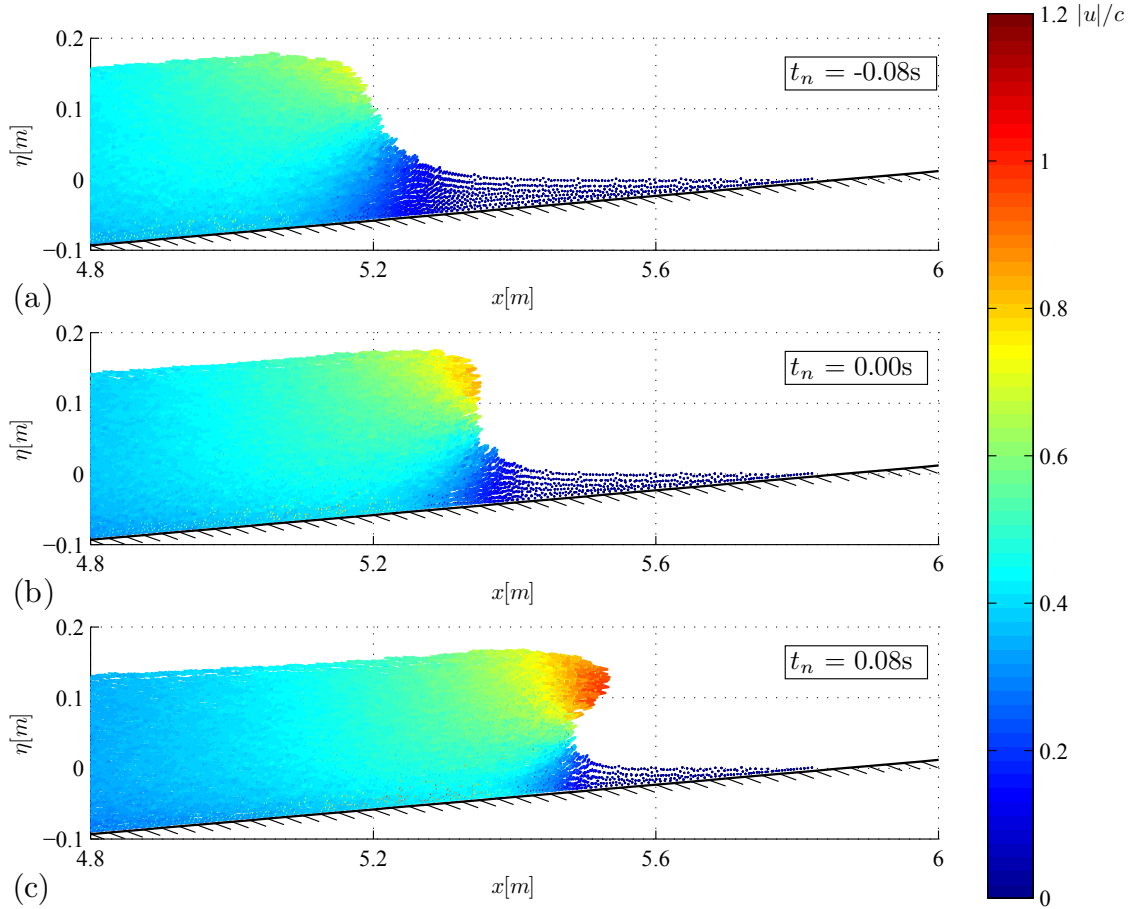


Figure 5.31: Velocity vector plots of a solitary wave with $H/d = 0.3$ at three times when the wave is close to the breaking point, or the point at which the wave face becomes vertical: (a) 0.08s before reaching the breaking point, (b) at the breaking point and (c) 0.08s after reaching the breaking point.

The next two figures concern *Case 1* (Figure 5.33) and *Case 2* (Figure 5.34) respectively and show both the velocity vectors and pressure as the waves progress through the breaking process. In both figures, the left column describes the velocity vectors,

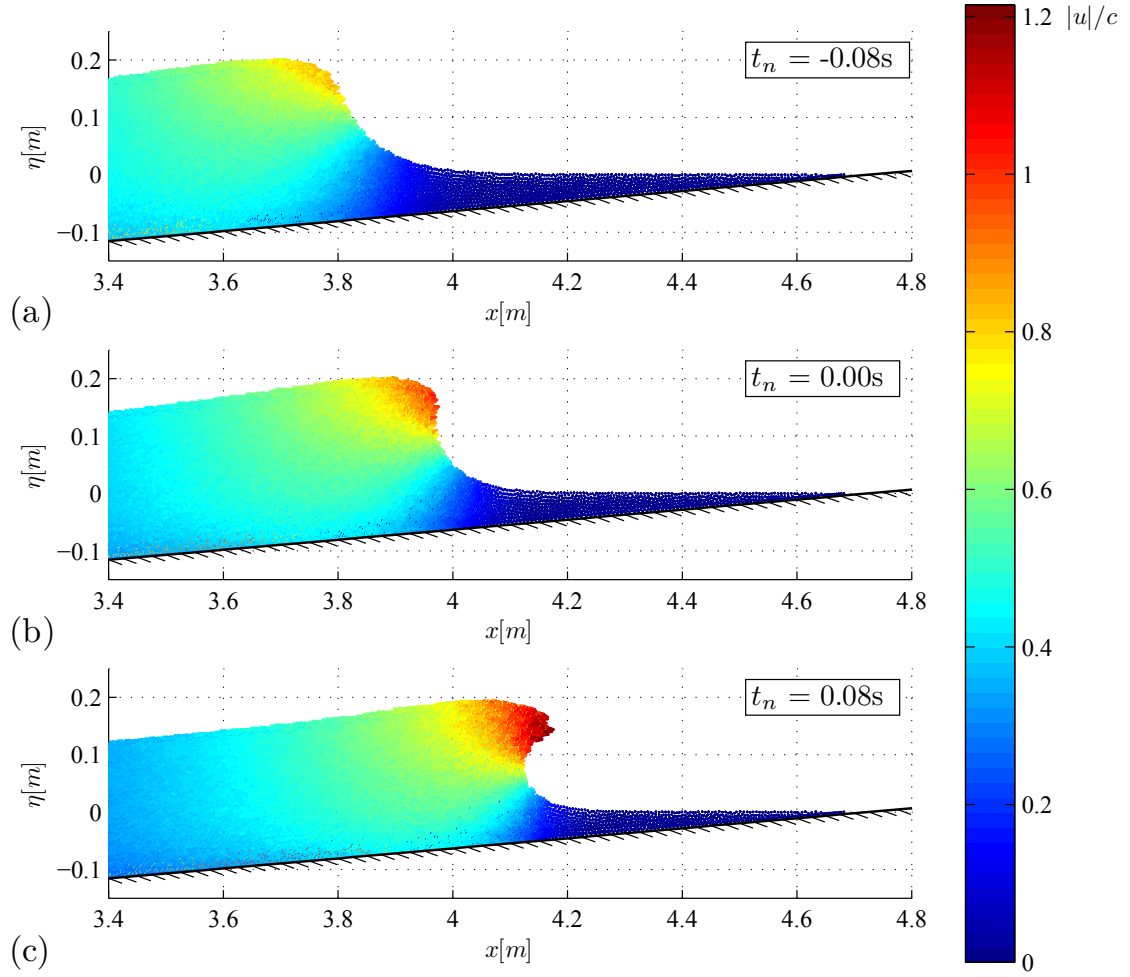


Figure 5.32: Velocity vector plots of a solitary wave with $H/d = 0.45$ at three times when the wave is close to the breaking point, or the point at which the wave face becomes vertical: (a) 0.08s before reaching the breaking point, (b) at the breaking point and (c) 0.08s after reaching the breaking point.

normalised with respect to the wave speed, while the right column shows the pressure, relative to atmospheric pressure, measured in Pascals.

In both cases, an increase in particle speed is observed as the jet touches down on the bed slope and begins the process of wave run-up. Likewise, at the point of wave impact the pressure increases dramatically. For example, in the third image of Figure 5.33, relating to *Case 1*, the maximum pressure is approximately 31000Pa. This is an enormous increase in the pressure relative to that recorded in the jet prior to its impact on the bed slope. In *Case 2* the maximum predicted pressure was lower, at

approximately 20000Pa. At first sight this may not have been what was expected with a larger wave. However, the jet is thinner in this case and the jet impacted on the water in front, rather than on the dry bed slope.

In the first three images for each case, the pressure behind the breaking wave front is approximately hydrostatic. This is exactly as expected. However, in the last image of each case, after the jet has impacted on the bed slope, the flow has become turbulent. The fluctuations associated with this, together with the collapse of the air cavity, causes a fluctuation, or modulation, in the pressure field in the water behind the cavity.

In both cases, after the jet impacts on the bed slope, there is a large increase in the particle velocity as the water begins to run up the slope. In *Case 2* the speed of the water in the jet rebounding off the still water in front is even higher than when it starts to travel parallel to the slope in the final image. This emphasises the effect that this aspect of the jet impact has on the rest of the breaking process.

The final images in this section show the velocity vectors associated with the hydraulic jump in each of the two wave cases. In both of these figures the actual velocity is presented in m/s. The hydraulic jump produced in *Case 1* is shown in Figure 5.35, while that produced in *Case 2* is shown in Figure 5.36. In both images the fast and shallow supercritical flow running down the slope is clearly observed. This results in the development of a turbulent hydraulic jump as the supercritical flow meets the deep and slow-moving subcritical flow in the flume below. A brief definition states that a hydraulic jump will only occur when flow moving under these two regimes meets, resulting in highly turbulent mixing and considerable energy loss. Although it is difficult to see in these images, the deep water is still moving slowly down the slope at this stage.

The two additional values (other than 0, 1 and 2m/s) marked on the colour scales of Figures 5.35 and 5.36 give the critical velocity for both the shallow and deep regions of the flow. The critical velocity occurs when the Froude number is equal to one or, in other words, when the flow speed, $|u|$, equals the propagation speed of a long wave within the flow, $\sqrt{gd}$. For *Case 1*, in Figure 5.35, the depth in the shallow water flow at $x = 5\text{m}$ is $d = 0.02\text{m}$, which ensures that if the water was travelling at 0.45m/s the flow would be critical. However, the flow is clearly travelling considerably faster than this, making it supercritical. In the deep water at $x = 4.4\text{m}$ the water depth is $d = 0.15\text{m}$ which ensures that if $u = 1.23\text{m/s}$ the flow would be critical. However, at this location

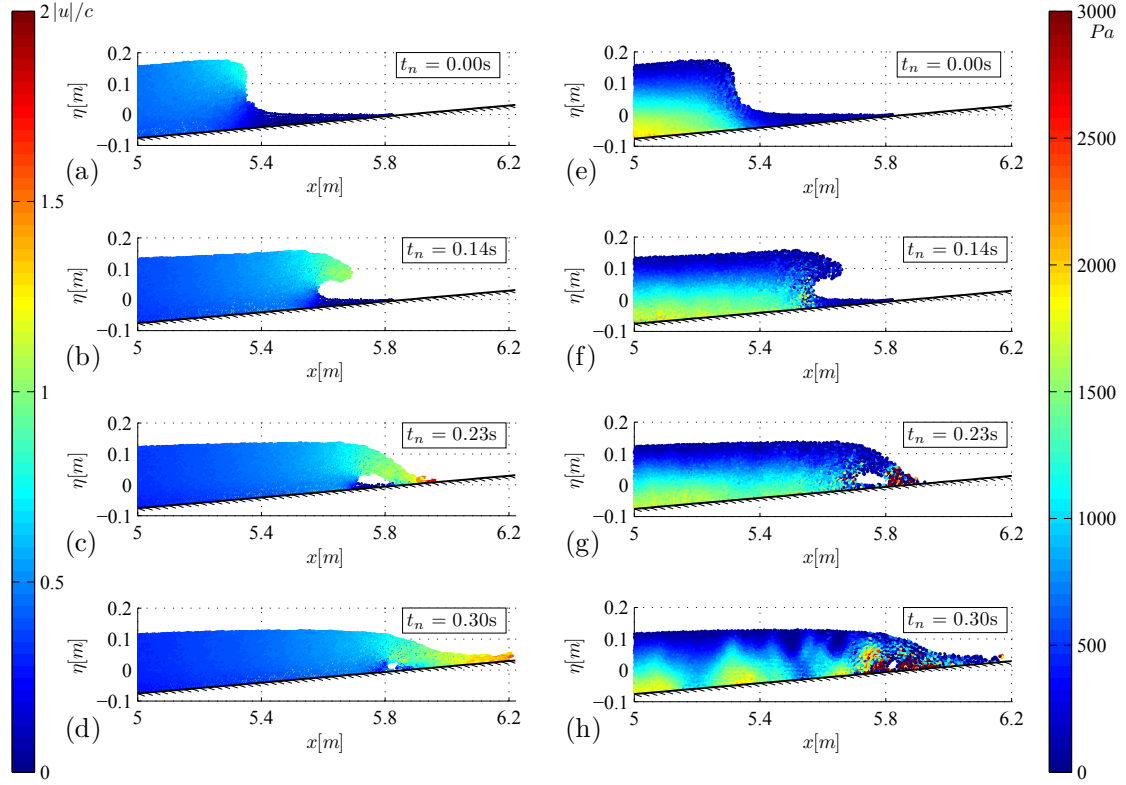


Figure 5.33: Velocity vector plots (left column) and pressure plots (right column) of a solitary wave with $H/d = 0.3$ at four times during breaking: (a) velocity vectors at $t = 0.00s$, (b) velocity vectors at $t = 0.14s$, (c) velocity vectors at $t = 0.23s$, (d) velocity vectors at $t = 0.30s$, (e) pressure at $t = 0.00s$, (f) pressure at $t = 0.14s$, (g) pressure at $t = 0.23s$ and (h) pressure at $t = 0.30s$. Times are relative to breaking point.

the water is clearly travelling much slower than this, making it subcritical. Likewise for *Case 2*, shown in Figure 5.36, the depth and speed of the shallow water flow at $x = 4m$ ($d = 0.03m$, $u > 0.54m/s$) means that this flow is supercritical, while the depth and speed of the deep water at $x = 3.4m$ ($d = 0.14m$, $u < 1.16m/s$) ensures that this flow is subcritical. Given the necessary transition from supercritical to subcritical flow, a hydraulic jump has to form in both wave cases. This is successfully predicted by the SPH model.

It is also interesting to note the increased velocities of the water particles riding over the back of the main wave in each of the hydraulic jumps. These local maxima occurring at $x = 4.7m$ on Figure 5.35 and $x = 3.7m$ on Figure 5.36, are the cause of a secondary (smaller) wave breaking further down the slope, and lead to a disturbance of the flow

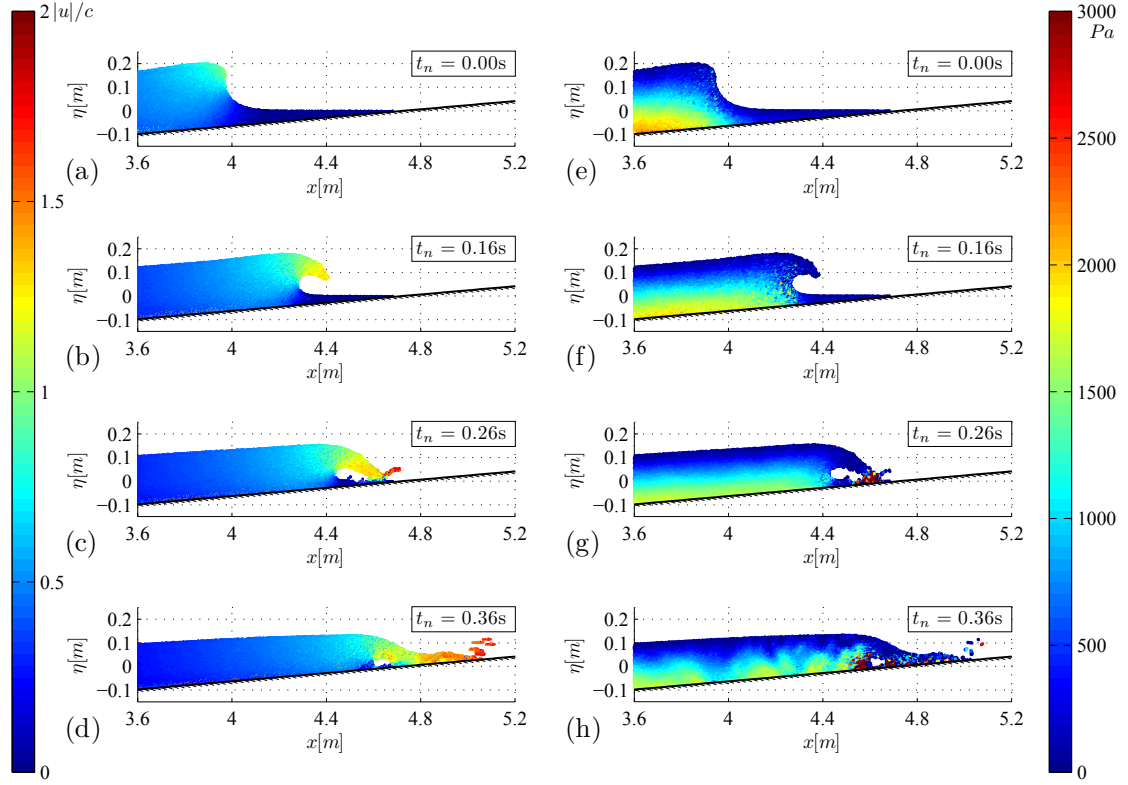


Figure 5.34: Velocity vector plots (left column) and pressure plots (right column) of a solitary wave with $H/d = 0.45$ at four times during breaking: (a) velocity vectors at $t = 0.00s$, (b) velocity vectors at $t = 0.16s$, (c) velocity vectors at $t = 0.26s$, (d) velocity vectors at $t = 0.36s$, (e) pressure at $t = 0.00s$, (f) pressure at $t = 0.16s$, (g) pressure at $t = 0.26s$ and (h) pressure at $t = 0.36s$. Times are relative to breaking point.

underneath.

These images both represent single snapshots of constantly changing, turbulent flow. At other times within the evaluation of the hydraulic jump, it would be possible to see many other turbulent effects within the SPH results. For this reason, SPH has proved to be an excellent tool for studying this type of flow.

5.6 Conclusions

The SPH model, developed in the context of propagating solitary waves over a long distance in Chapter 4, was used to consider the full process of solitary waves breaking on a 5° plane slope. Initial calculations of a solitary wave with $H/d = 0.3$ in a water depth of $d = 0.5m$ and a resolution of 50 particles over the initial water depth, suggested

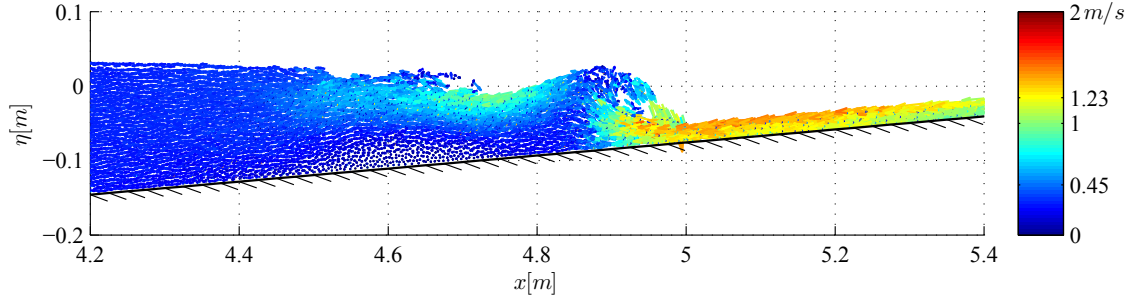


Figure 5.35: Velocity vector plot of a solitary wave with $H/d = 0.3$ during the hydraulic jump.

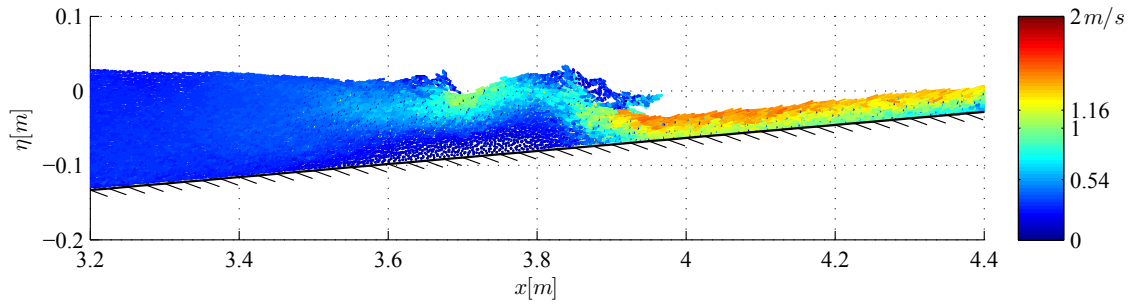


Figure 5.36: Velocity vector plot of a solitary wave with $H/d = 0.45$ during the hydraulic jump.

that this SPH model was unable to capture some of the flow phenomena involved in wave breaking. In particular, the turbulent processes involved in the post-breaking wave behaviour were not thought to be satisfactorily modelled and the run-up and wash down of a thin layer of water on the slope was not described sufficiently. Accordingly, a mechanism for properly modelling flow running over previously dry boundaries and for the drying out of previously wet boundaries was introduced. In addition, an SPS turbulence model and a turbulent boundary condition were introduced into the model.

Preliminary results, incorporating these additions to the model, showed some improvement in the shape of the emerging jet as the solitary wave began to overturn. However, it was clear that the resolution of 50 particles over the initial water depth, corresponding to an initial particle spacing of $\Delta x = \Delta y = 0.01\text{m}$, was inadequate to capture all the details of the wave breaking. For this reason, the simulation was only continued beyond breaking once the resolution was increased to 100 particles over the initial water depth. Two wave cases were considered in detail: *Case 1* corresponding to $H/d = 0.3$ and *Case 2*, $H/d = 0.45$.

Overall, the results showed that the SPH model was capable of reproducing the main processes involved as the solitary waves began to overturn, producing plunging breakers which impacted on the slope in front, with subsequent run-up and wash-down on the slope, finally resulting in a hydraulic jump. Once the experimental results were aligned with the numerical predictions at the breaking point, in both wave cases, as the wave passed through breaking point and began to overturn, the SPH predictions of the wave profiles were in excellent agreement with the laboratory observations. However, as the overturning jets began to form, some minor differences started to appear in the details of the breaking procedure. The most significant difference, resulting in the greatest consequences, was a lack of forward momentum in the crest of the overturning wave.

Water particle velocities and the associated pressure field during the breaking process were also presented from the SPH results. These images demonstrate the ease with which properties of the flow can be calculated from a validated numerical model.

In order to improve upon the results of these quantitative comparisons, several adaptations need to be made to the SPH model. First, the viscosity model requires further testing in order to improve upon the behaviour of particles in the overturning wave. Second, the cavity that is formed when the wave touches down on the slope, and subsequent entrapped air, may be better represented in the SPH model if the air phase was also modelled by SPH particles. This would also allow smaller bubbles to be visualised as single SPH air particles within the turbulent waters.

As the breaking continues, and particularly during the hydraulic jump, turbulence becomes increasingly important. The SPS turbulence model employed here is relatively basic and the results may well be improved upon by using a state-of-the-art turbulence model, although this would increase the complexity of the SPH model and runtime of the simulations. The scheme used to create the turbulent boundary condition should also be considered in more detail to improve upon the results of the maximum run-up on the bed slope.

It is also in the latter stages of breaking that three-dimensional effects start to become important. As a result, improvements are likely to be seen by extending this SPH model to three dimensions. However, this would significantly increase runtimes. As the results at the higher resolution of 100 particles over the initial depth were substantially more accurate than at the lower resolution, increasing the resolution further may result

in greater improvements. Therefore, a convergence test should be carried out at increasingly high resolutions. However, this too would greatly increase runtime. For these reasons, one of the most important developments for the SPH model would be to adapt the program for parallel computing. This is the only method by which the extensions to the model can be incorporated whilst keeping the runtimes manageable.

Conclusions and further work

The work outlined in this thesis has comprised the development of an SPH model for studying free-surface flows. Initially developed through tests on unbroken solitary waves propagating in a flume, the model was then extended to be able to satisfactorily simulate flows involving surface break-up, turbulence and regions of thin-layered flow. A study involving the full process of solitary waves breaking on a plane beach was chosen to demonstrate the models capabilities. Where possible, validation was carried out through quantitative comparisons with available theory and laboratory experiments. Comparisons of the solitary waves breaking on a plane beach were shown to be in good agreement with the experiments. Some principal achievements of this work are given below, along with their practical relevance in engineering and some suggestions for further research.

6.1 Principal achievements

The work carried out for this thesis has:

1. **Developed an SPH model for simulating a wide range of free-surface flows.**

A thorough review of the SPH literature was carried out in order to research the various modelling techniques currently in use. The most promising of these techniques were then investigated further through a sequence of rigorous tests.

Individual techniques were analysed in isolation to assess their contribution to the overall model accuracy. The best of these were compiled into the current SPH model through selection of the most appropriate techniques working in combination with one another. The model was originally developed and optimised for simulating non-violent, free-surface flows through a case study focusing on propagating solitary waves. Building on this success, the model was further developed and validated through studies of violent free-surface flows with large deformations from the initial surface geometry.

2. Demonstrated the SPH model's ability to propagate solitary waves over long distances with only minimal and explainable losses to the wave height.

The SPH model was used to propagate a solitary wave, with a height to depth ratio of 0.3, over a distance in the order of a hundred times the wave height. This was achieved with only minimal losses to the wave height. The model results were compared with a number of theories and found to compare extremely well to McCowan's first-order theory. These comparisons considered time histories of the free-surface, $\eta(t)$, wave profiles, $\eta(x)$, and the horizontal particle velocities under the wave crest, $u(z)$. In order to add further support to the validation, the model results were also compared to detailed experimental observations. In undertaking these comparisons further SPH simulations were carried out with wave height to depth ratios of 0.1, 0.2 and 0.45.

3. Implemented improvements to the modelling of solid boundaries with SPH in regions which had previously been dry or had dried out having previously been wet.

As boundaries dry out or re-wet the *Stationary Ghost* boundary method was found to cause significant problems. However, having previously been selected as the best boundary method for most flows there was a reluctance to remove the method entirely. As a result, a combination of the *Repulsive Force* and *Stationary Ghost* boundary methods was adopted. A switch to change between the methods under specified conditions was introduced within the SPH model.

4. Demonstrated the SPH model's ability to describe breaking solitary

waves.

A study was carried out to investigate solitary waves with height to depth ratios of 0.3, in 0.5m of water, and 0.45, in 0.4m of water. In both cases the waves broke on a plane slope orientated at 5° to the horizontal. The results demonstrated that the SPH model was capable of describing the main stages involved in the entire breaking process. These included the initial evolution of the overturning wave, the plunging breaker impacting on the slope in front, the subsequent run-up and wash-down, and the formation of an unsteady, or transitory, hydraulic jump due to the supercritical flow associated with the wash down.

5. Validated the SPH model for solitary wave breaking on a slope by providing quantitative comparisons with laboratory experiments.

Detailed and quantitative comparisons have been made between the numerical predictions of the SPH model and the laboratory data, describing each stage in the breaking process. In particular, initial comparisons were made at the breaking point, followed by comparisons addressing both the shape of the overturning wave and the projection angle and shape of the developing jets. Although minor discrepancies existed between the observed and predicted behaviour, the SPH model predictions were generally found to be in good agreement with the laboratory data and produced realistic results at all stages. The maximum run-up on the slope and the location and severity of the subsequent hydraulic jump were also considered. In these cases the differences between the model predictions and the observed behaviour were larger. This, in part, reflects the increased complexity of these predictions. In addition, the later stages of the breaking process are intrinsically linked to the development of a very narrow layer of fluid which first runs up the previously dry bed slope and then washes down to produce the hydraulic jump. Unfortunately, the maximum achievable particle resolution was such that these aspects of the flow suffered from inadequate particle resolution. In such cases the SPH model cannot be expected to reproduce the laboratory observations with a high degree of accuracy. Nevertheless, the SPH model was generally shown to be in reasonably good agreement with the observed motions for all aspects of the breaking and post-breaking behaviour.

6.2 Engineering significance and practical implications

Mesh-based numerical models often struggle to simulate free-surface flows when significant break-up and turbulence affects the state of the free-surface. The work carried out in this thesis has demonstrated the SPH model's ability to deal with such flows. These studies highlighted the ease with which the SPH model is able to capture complete break-up of the free-surface, as well as regions of high turbulence and mixing. Thorough validation of the SPH model, based upon quantitative comparisons with laboratory experiments, has demonstrated the usefulness of the SPH model. Indeed, the success of the comparisons suggests that SPH could become an important part of model testing for engineering developments involving this type of flow.

Using a suitable numerical model as part of design testing or feasibility studies can reduce costs in comparison to using laboratory testing alone. Numerical models also allow greater flexibility; changes to an SPH model layout are usually easier to implement than changes in an experimental layout. The simplicity of obtaining a wide range of flow measurements across the whole flow field using the SPH model has also been demonstrated. This includes predictions of the pressure field and flow velocity. These types of measurements are often useful in initial studies to get a general idea of the qualitative nature of a flow or to highlight any outstanding or abnormal results. However, ideally all the numerical predictions should be quantitatively validated by comparison to observed data.

In its present state, there are many opportunities for applying the SPH model to complex free-surface flow problems. However, there remain some tricky problems to overcome before it can be said to be quantitatively valid for modelling all aspects of the flows described in this thesis. The results of the studies carried out in this thesis suggest that SPH can add significantly to design calculations, but must do so in conjunction with other methods, such as laboratory tests.

6.3 Suggestions for further work

In developing and validating the SPH model described in this thesis, it is clear that a number of areas require further work. These include additional developments to the

model itself, improvements and extensions to the validation work, and an expansion of the range of flows that the model can be used to simulate. These points are discussed below.

1. Viscosity modelling

Further testing should be carried out concerning the viscosity formulation used in the SPH model. This is particularly relevant to the behaviour of SPH particles in an overturning wave, where the accuracy of this aspect of the model is highly important. Although three viscosity formulations were tested for use in this SPH model, it seems likely that the selected formulation is still causing some inaccuracies in the model predictions which could be improved through further research and development.

2. Turbulence modelling

Unfortunately, within the present scope of work it was not possible to test more than one turbulence model; a relatively basic SPS turbulence model has been employed throughout. However, there are other turbulence models available, such as the $k - \epsilon$ model, which may have produced better results. Ongoing research into turbulence modelling is continuously providing more accurate solutions, some of which have or could be adapted for SPH modelling. However, improved turbulence models tend to be more complex and are therefore more computationally expensive. In order to truly benefit from an improved turbulence model it would also be necessary to use a three-dimensional SPH model (since turbulence is an inherently three-dimensional process) and combine this with an increase in the particle resolution. As these extensions are also more computationally expensive, it would be necessary to parallelise the SPH model code. These changes are discussed separately below.

3. Turbulent boundary conditions, bed roughness and friction

The necessity to accurately model friction or bed roughness first became clear when studying the run-up on a plane slope from a broken solitary wave. This case study can be seen as a much simplified representation of a tsunami coming ashore. If the SPH model were to be used for this purpose, being able to simulate the correct maximum run-up height of water on the land would be essential. In

addition to being able to establish the desired bed roughness for particular cases, it is also important to be able to obtain the correct roughness by using meaningful parameters as input to the SPH model. Alternatively, a calibration process could be used to match the roughness calculated by the model predictions to laboratory or field observations. However, this is unsatisfactory as it does not provide a general solution nor a genuinely predictive solution.

The bed roughness is also the key factor in establishing the correct turbulent boundary condition for producing the desired flow characteristics within larger bodies of water. It is therefore important for many turbulent flows, not just thin-layer flows. Given its significance in terms of key flow characteristics, ensuring the numerically predicted boundary conditions are in good agreement with the experimental observations is of great importance.

4. Multi-phase flow

Some of the model predictions obtained within this thesis indicated that improvement may be gained by modelling the air phase as well as the water phase. For example, once the solitary wave touches down on the slope after overturning, a cavity is formed. This contains air which becomes entrapped beneath the water as it starts to run up the slope. This may be better represented within the SPH model if the air phase was modelled with SPH particles. Including the air phase would also allow smaller bubbles to be visualised as single SPH air particles within turbulent water. In addition, the introduction of the air phase into the SPH model would be beneficial when seeking to capture the considerable effects of aeration in the hydraulic jump. However, unless the particle resolution is increased to be fine enough to include air bubbles on the scale displayed by the experiments, this may introduce further problems resulting from oversized pockets of air being swept into the turbulent mixing zone.

The ability to include additional phases in the SPH model would also make SPH a useful tool for studying density stratified flows or visualising the spread of a tracer released into the flow with either the same or a different density to the main fluid. However, no work of this type was undertaken in the present study. Another useful development of the SPH model would be to add in a second solid phase.

This would allow the movement of sediment particles to be tracked within a flow, for example, within a hydraulic jump. In particular, a study of this nature could be used to gain a better understanding of sediment mixing. If the SPH model was coupled with an external sediment transport model, the effects of scour on a riverbed could also be considered.

5. **Increase in particle resolution, particularly within thin-layered flows**

One of the most obvious improvements to the accuracy of any numerical model is likely to come from increasing the resolution of the mesh or, in this case, increasing the number of SPH particles. The studies described within this thesis have demonstrated that a relatively low particle resolution is particularly problematic in regions of thin-layered flow, where the particle to depth ratio was often reduced to less than ten particles. A sensitivity test relating to the particle resolution was carried out in respect of the solitary wave breaking on a slope. The simulation with the higher of the two particle resolutions was found to give far superior results, providing a better definition in the shape of the overturning wave and more detail in the jet formation. This suggests that increasing the resolution still further may give additional improvements. A convergence test, performing simulations of this flow at increasingly higher particle resolutions, could be used to demonstrate this effect.

6. **Three-dimensional simulations**

The studies carried out in this thesis were deliberately chosen because the flow characteristics are predominantly two-dimensional. Nevertheless, some aspects of these flows displayed three-dimensional characteristics. Specifically, it was felt that a three-dimensional model would have been beneficial in simulating the latter stages of wave breaking. In addition, it would also be desirable to be able to model fully three-dimensional free-surface flows, such as directionally spread wave groups, particularly those that are on the verge of breaking.

Several of the cases considered within this thesis involved regions of highly turbulent flow. As turbulence is an inherently three-dimensional process, a three-dimensional SPH model would be necessary to truly capture this aspect of the flow. It is therefore proposed that an important future development would be to

extend the SPH model to three dimensions.

7. Parallelisation of the code and computational efficiency

In considering the suggestions for further work described above, nearly all require a greater computational effort, resulting in substantially increased runtimes. For example, improving the turbulence model could potentially involve the need to solve more equations within the model formulation and may even require adaptations to the time-stepping scheme, resulting in more computational effort per time-step. If the air phase was included within the model, this is likely to require in the order of at least double the number of SPH particles, as the chosen domain would need to be entirely filled with particles. Obviously, increasing the particle resolution would require more computational effort. In particular, increasing the particle resolution to a level suitable for thin-layered flows would currently involve a vast increase to the particle resolution everywhere else. In addition, not only would extending the model to three dimensions increase the number of SPH particles required, it would also mean solving the momentum equation in three-dimensions, with more particle pairs per particle. This would undoubtedly lead to an increased effort when performing every single calculation. As the current SPH model has already stretched the limits of viable simulation times, the only way to be able to include these improvements to the model and still obtain results within a reasonable time frame, is to parallelise the code. Indeed, this is a critical factor in the model's future development. As a result, it is probably the first adaptation that should be undertaken.

8. Extension to the range of model testing and validation - further applications

As well as improving various aspects of the SPH model, it would also be valuable to extend the range of free-surface flows used for testing and validating the model. Other wave applications could include regular waves breaking on a beach, both focused waves and extreme wave events involving breaking or overtopping, and interactions of waves with solid structures. The model could also be used to study complex hydraulic flows such as steady-state hydraulic jumps and spillway flow. As well as adding to the validation, these tests would further demonstrate the

capabilities of the SPH model, particularly if the model is adapted to include the improvements described above.

References

- AMERICAN INSTITUTE OF AERONAUTICS AND ASTRONAUTICS 1998 *AIAA Guide for the Verification and Validation of Computational Fluid Dynamics Simulations*. American Institute of Aeronautics and Astronautics.
- ARCHIBALD, S. 2011 Modelling of extreme ocean waves using high performance computing. PhD thesis, Department of Civil & Environmental Engineering, Imperial College London.
- ATA, R. R. 2005 A stabilized SPH method for inviscid shallow water flows. *International Journal for Numerical Methods in Fluids* **47** (2), 139–159.
- ATAIE-ASHTIANI, B. & SHOBEYRI, G. 2008 Numerical simulation of landslide impulsive waves by incompressible smoothed particle hydrodynamics. *International Journal for Numerical Methods in Fluids* **56** (2), 209–232.
- ATAIE-ASHTIANI, B., SHOBEYRI, G. & FARHADI, L. 2008 Modified incompressible sph method for simulating free surface problems. *Fluid Dynamics Research* **40** (9), 637 – 661.
- BALSARA, D. S. 1995 von Neumann stability analysis of Smooth Particle Hydrodynamics – suggestions for optimal algorithms. *Journal of Computational Physics* **121**, 357–372.
- BATCHELOR, G. K. 1974 *An Introduction to Fluid Mechanics*. Cambridge University Press.
- BLIN, L., HADJADJ 3, A. & VERVISCH, L. 2003 Large eddy simulation of turbulent flows in reversing systems. *Journal of Turbulence* p. N1.

- BONET, J. & LOK, T. S. L. 1999 Variational and momentum preservation aspects of Smooth Particle Hydrodynamic formulations. *Computer Methods in Applied Mechanics and Engineering* **180** (1-2), 97–115.
- BORTHWICK, A., FORD, M., WESTON, B., TAYLOR, P. & STANSBY, P. 2006 Solitary wave transformation, breaking and run-up at a beach. *Proceedings of the ICE - Maritime Engineering* **159** (3), 97–105.
- BOUSSINESQ, J. 1872 Theorie des ondes et des remous qui se propagent le long d'un canal rectangulaire horizontal, en communiquant au liquide contenu dans ce canal des vitesses sensiblement pareilles de la surface au fond. *Journal de Mathematique Pures et Appliquees, Deuxieme Serie* **17**, 55–108.
- BYATT-SMITH, J. 1970 An exact integral equation for steady surface waves. *Procedure of the Royal Society A* **315**, 405–418.
- CANNY, J. J. 1986 A computational approach to edge detection. *IEEE Transactions on Pattern Analysis and Machine Intelligence* **PAMI-8** (6), 679–698.
- CASSIDY, J. J. 1965 Irrotational flow over spillways of finite height. *Journal of Engineering Mechanics Division, ASCE* **91**(6), 155–173.
- CHRISTOU, M. 2008 Fully nonlinear computations of waves and wave-structure interaction. PhD thesis, Department of Civil & Environmental Engineering, Imperial College London.
- CHRISTOU, M., SWAN, C. & GUDMESTAD, O., T. 2007 The description of breaking waves and the underlying water particle kinematics. In *26th International Conference on Offshore Mechanics and Arctic Engineering*.
- COLAGROSSI, A. & LANDRINI, M. 2003 Numerical simulation of interfacial flows by Smoothed Particle Hydrodynamics. *Journal of Computational Physics* **191** (2), 448–475.
- COLAGROSSI, A., LUGNI, C. & BROCCINI, M. 2010 A study of violent sloshing wave impacts using an improved sph method. *Journal of Hydraulic Research* **48** (sup1), 94–104.

- CORRIGNAN, P. 1994. Analyse physique des phenomenes associes au ballotement de liquide dans des reservoirs (sloshing). PhD thesis, Ecole Centrale de Nantes.
- CRESPO, A. C., DOMINGUEZ, J. M., BARREIRO, A., GMEZ-GESTEIRA, M. & ROGERS, B. D. 2011 Gpus, a new tool of acceleration in cfd: Efficiency and reliability on smoothed particle hydrodynamics methods. *PLoS ONE* **6** (6).
- CRESPO, A. J. C., GOMEZ-GESTEIRA, M. & DALRYMPLE, R. A. 2008 Modeling dam break behavior over a wet bed by a SPH technique. *Journal of Waterway, Port, Coastal, and Ocean Engineering* **134** (6), 313–320.
- CUMMINS, S. J. & RUDMAN, M. 1999 An sph projection method. *Journal of Computational Physics* **152** (2), 584 – 607.
- DALRYMPLE, R. & ROGERS, B. 2006 Numerical modeling of water waves with the SPH method. *Coastal Engineering* **53** (2-3), 141–147.
- DALRYMPLE, R., ROGERS, B., NARAYANASWAMY, M., ZOU, S., GESTEIRA, M., CRESPO, A. & PANIZZO, A. 2007 Smoothed Particle Hydrodynamics for water waves. In *Proceedings of the 26th International Conference on Offshore Mechanics and Arctic Engineering*.
- DALRYMPLE, R. A., GÓMEZ-GESTEIRA, M., ROGERS, B. D., PANIZZO, A., ZOU, S., CRESPO, A. J. C., CUOMO, G., NARAYANASWAMY, M. & MA, Q. 2009 Smoothed particle hydrodynamics for water waves. *Advances in numerical simulation of nonlinear water waves* .
- DAS, K., JANETZKE, R., BASU, D., GREEN, S. & STAMATAKOS, J. 2009 Numerical simulations of tsunami wave generation by submarine and aerial landslides using rans and sph models. *ASME Conference Proceedings* **2009** (43451), 581–594.
- DE LEFFE, M., LE TOUZ, D. & ALESSANDRINI, B. 2008 Coastal flow simulation using a SPH formulation modelling the non-linear shallow water equations. In *ERCOFTAC SIG SPHERIC IIIrd International Workshop*.
- DORING, M., ANDRILLON, Y., ALESSANDRINI, B. & FERRANT, P. 2003 SPH free surface flow simulation. In *Electronic Proceedings of the 18th International Workshop on Water Waves & Floating Bodies*.

- FENTON, J. 1972 A ninth-order solution for solitary wave. *Journal of Fluid Mechanics* **53** (MAY23), 257–271.
- FERRAND, M., VIOLEAU, D., ROGERS, B. & LAURENCE, D. 2011 Consistent wall boundary treatment for laminar and turbulent flows in SPH. In *6th International SPHERIC Workshop*.
- FERRARI, A., DUMBSER, E., TORO, A. & ARMANINI, A. 2007 A new stable and consistent version of the SPH method in Lagrangian coordinates. In *SPHERIC 2nd Internation Conference*.
- FERRARI, A., DUMBSER, M., TORO, E. & ARMANINI, A. 2010 A new parallel sph method for 3d free surface flows. In *High Performance Computing on Vector Systems 2009* (ed. M. Resch, S. Roller, K. Benkert, M. Galle, W. Bez & H. Kobayashi), pp. 179–188. Springer Berlin Heidelberg.
- FERZIGER, J. H. & PERIC, M. 2002 *Computational Methods for Fluid Dynamics, 3rd Edition*. Springer.
- FOUREY, G., LE TOUZE, D., ALESSANDRINI, B. & OGER, G. 2010 SPH-FEM coupling to simulate fluid-structure interactions with complex free-surface flows. In *5th International SPHERIC Workshop*.
- GINGOLD, R. A. & MONAGHAN, J. J. 1977 Smoothed Particle Hydrodynamics theory and application to non-spherical stars. *Monthly Notices of the Royal Astronomical Society* **181** (2), 375–389.
- GOMEZ-GESTEIRA, M., CERQUEIRO, D., CRESPO, C. & DALRYMPLE, R. 2005 Green water overtopping analyzed with a SPH model. *Ocean Engineering* **32** (2), 223–238.
- GOMEZ-GESTEIRA, M., ROGERS, B. D., DALRYMPLE, R. A. & CRESPO, A. J. 2010 State-of-the-art of classical SPH for free-surface flows. *Journal of Hydraulic Research* **48** (sup1), 6–27.
- GONG, K., LIU, H. & LONG WANG, B. 2009 Water entry of a wedge based on sph model with an improved boundary treatment. *Journal of Hydrodynamics, Ser. B* **21** (6), 750 – 757.

- GOPALA, V. & VAN WACHEM, B. 2008 Volume of fluid methods for immiscible-fluid and free-surface flows. *Chemical Engineering Journal* **141** (13), 204 – 221.
- GORING, D. G. 1979 Tsunamis—the propagation of long waves onto a shelf. PhD thesis, California Institute of Technology Pasadena, California.
- GOTOH, H., IKARI, H., MEMITA, T. & SAKAI, T. 2005 Lagrangian particle method for simulation of wave overtopping on a vertical seawall. *Coastal Engineering Journal* **47** (2-3), 157–181.
- GOTOH, H. & SAKAI, T. 2006 Key issues in the particle method for computation of wave breaking. *Coastal Engineering* **53**, 171–179.
- GOTOH, H., SHIBIHARA, T. & SAKAI, T. 2001 Sub-particle-scale model for the MPS method - Lagrangian flow model for hydraulic engineering. *Computational Fluid Dynamics Journal* **9** (4), 339–347.
- GRENIER, N., ANTUONO, M., COLAGROSSI, A., TOUZ, D. L. & ALESSANDRINI, B. 2009 An hamiltonian interface sph formulation for multi-fluid and free surface flows. *Journal of Computational Physics* **228** (22), 8380 – 8393.
- GRILLI, S., GILBERT, R., LUBIN, P., VINCENT, S., ASTRUC, D., LEGENDRE, D., DUVAL, M., KIMMOUN, O., BRANGER, H., DEVRARD, D., FRAUNIE, P. & ABADIE, S. 2004 Numerical modeling and experiments for solitary wave shoaling and breaking over a sloping beach. In *Proceedings of the Fourteenth International Offshore and Polar Engineering Conference*.
- GRILLI, S., SVENDSEN, I. & SUBRAMANYA, R. 1997 Breaking criterion and characteristics for solitary waves on slopes. *Journal of Waterway, Port, Coastal and Ocean Engineering-ASCE* **123** (3), 102–112.
- GRILLI, S., SVENDSEN, I. & SUBRAMANYA, R. 1998 Closure on: Breaking criterion and characteristics for solitary waves on slopes. *Journal of Waterway, Port, Coastal and Ocean Engineering, ASCE* **124** (6), 333–335.
- GRIMSHAW, R. 1971 The solitary wave in water of variable depth. Part 2. *Journal of Fluid Mechanics* **46** (3), 611–622.

- GUIGNARD, S., MARCER, R., REY, V., KHARIF, C. & FRAUNIE, P. 2001 Solitary wave breaking on sloping beaches: 2-D two phase flow numerical simulation by SL-VOF method. *European Journal of Mechanics* **20** (1), 57–74.
- HAGUE, C. H. 2006 Fully nonlinear computations of directional waves, including wave breaking. PhD thesis, Department of Civil & Environmental Engineering, Imperial College London.
- HAGUE, C. H. & SWAN, C. 2007 On the advantages of a multiple flux boundary element method applied to the description of surface water waves. *Submitted to the Journal of Computational Physics* .
- HAGUE, C. H. & SWAN, C. 2009 A multiple flux boundary element method applied to the description of surface water waves. *Journal of Computational Physics* **228** (14), 5111–5128.
- HALL, J. V. & WATTS, G. M. 1953 Laboratory investigation of the vertical rise of solitary waves on impermeable slopes. U.S. Army Corps of Engineers, Beach Erosion Board, Technical Memo no. 33.
- HARLOW, F. & WELCH, J. 1965 Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface. *The Physics of Fluids* **8** (12), 2182–2189.
- HIRT, C. W. & NICHOLS, B. D. 1981 Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics* **39** (1), 201–225.
- HUGHES, J. P. & GRAHAM, D. I. 2010 Comparison of incompressible and weakly-compressible sph models for free-surface water flows. *Journal of Hydraulic Research* **48** (sup1), 105–117.
- INUTSUKA, S.-I. 2002 Reformulation of Smoothed Particle Hydrodynamics with a Riemann solver. *Journal of Computational Physics* **179**, 238–267.
- ISSA, R. & VIOLEAU, D. 2009 Modelling a plunging breaking solitary wave with eddy-viscosity turbulent sph models. *Computers, Materials & Continua* **8**(3), 151–164.

- JASAK, H. & WELLER, H. 1995 Interface tracking capabilities of the inter-gamma differencing scheme. *Tech. Rep.*. Imperial College of Science, Technology and Medicine, University of London.
- KASSIOTIS, C., FERRAND, M., VIOLEAU, D., ROGERS, B., STANSBY, P. & BENOIT, M. 2011 Coupling SPH with a 1-d boussinesq-type wave model. In *6th International SPHERIC Workshop*.
- KEULAGAN, G. 1948 Gradual damping of solitary waves. *Journal of Research of the National Bureau of Standards* **40**, 487–498.
- KHAYYER, A., GOTOH, H. & SHAO, S. 2008 Corrected incompressible SPH method for accurate water-surface tracking in breaking waves. *Coastal engineering* **55** (3), 236–250.
- KIM, S., LIU, P.-F. & LIGGET, J. A. 1983 Boundary integral equation solutions for solitary wave generation, propagation and run-up. *Coastal Engineering* **7**, 299–317.
- KOSHIZUKA, S., TAMAKO, H. & OKA, Y. 1995 A particle method for incompressible viscous flow with fluid fragmentation. *Computational Fluid Dynamics Journal* **4**, 29–46.
- LAITONE, E. V. 1960 The second approximation to cnoidal and solitary waves. *Journal of Fluid Mechanics* **9** (3), 430–444.
- LEE, E.-S., MOULINEC, C., XU, R., VIOLEAU, D., LAURENCE, D. & STANSBY, P. 2008 Comparisons of weakly compressible and truly incompressible algorithms for the SPH mesh free particle method. *Journal of Computational Physics* **227** (18), 8417–8436.
- LEE, E.-S., VIOLEAU, D., REZA, I. & PLOIX, S. 2010 Application of weakly compressible and truly incompressible SPH to 3-D water collapse in waterworks. *Journal of Hydraulic Research* **48** (Extra Issue).
- LI, Y. 2000 Tsunamis: Non-breaking and breaking solitary wave run-up. PhD thesis, Division of Engineering and Applied Science, California Institute of Technology.

- LI, Y. & RAICHLEN, F. 1998 Discussion on: Breaking criterion and characteristics for solitary waves on slopes. *Journal of Waterway, Port, Coastal and Ocean Engineering, ASCE* **124** (6), 329–333.
- LI, Y. & RAICHLEN, F. 2001 Solitary wave run-up on plane slopes. *Journal of Waterway, Port, Coastal and Ocean Engineering, ASCE* **127** (1), 33–44.
- LI, Y. & RAICHLEN, F. 2003 Energy balance model for breaking solitary wave runup. *Journal of Waterway, Port, Coastal and Ocean Engineering-ASCE* **129** (2), 47–59.
- LIND, S., XU, R., STANSBY, P. & ROGERS, B. 2012 Incompressible smoothed particle hydrodynamics for free-surface flows: A generalised diffusion-based algorithm for stability and validations for impulsive flows and propagating waves. *Journal of Computational Physics* **231** (4), 1499 – 1523.
- LIU, G. R. & LIU, M. B. 2003 *Smoothed Particle Hydrodynamics: a meshfree particle method*. World Scientific.
- LO, E. Y. & SHAO, S. 2002 Simulation of near-shore solitary wave mechanics by an incompressible SPH method. *Applied Ocean Research* **24** (5), 275–286.
- LONGUET-HIGGINS, M. S. & STEWART, R. W. 1960 Changes in the form of short gravity waves. *Journal of Fluid Mechanics* **8**, 565–583.
- LUCY, L. B. 1977 A numerical approach to the testing of the fission hypothesis. *The Astronomical Journal* **82**, 1013–1024.
- MAHMOOD, O., KASSIOTIS, C., VIOLEAU, D., FERRAND, M. & DENIS, C. 2011 Effect of wall boundary treatment in sph for modelling turbulence flows with inlet/outlet. In *6th International SPHERIC Workshop*.
- MARTIN, J. & MOYCE, W. 1952 Part iv. An experimental study of the collapse of liquid columns on a rigid horizontal plane. *Royal Society of London Philosophical Transactions Series A* **244**, 312–324.
- MCCOWAN, J. 1891 On the solitary wave. *London, Edinburgh and Dublin Philosophical Magazine and Journal of Science* **Ser. 5, 32**, 45–58.

- MILNE-THOMSON, L. 1964 An exact integral equation for the solitary wave. *Revue Roumaine des Sciences Techniques-Serie Mecanique Applique* **9**, 1189–1194.
- MOIN, P., SQUIRES, K., CABOT, W. & LEE, S. 1991 A dynamic subgrid-scale model for compressible turbulence and scalar transport. *Physics Of Fluids A Fluid Dynamics* **3** (11), 2746–2757.
- MONAGHAN, J. 1992 Smoothed Particle Hydrodynamics. *Annual Review of Astronomy and Astrophysics* **30**, 543–574.
- MONAGHAN, J. 2007 Time-stepping. In *Note given at SPHERIC 2nd International Workshop, Madrid*.
- MONAGHAN, J. & LATTANZIO, J. 1985 A refined particle method for astrophysical problems. *Astronomy and Astrophysics* **149** (1), 135–143.
- MONAGHAN, J. J. 1989 On the problem of penetration in particle methods. *Journal of Computational Physics* **82** (1), 1–15.
- MONAGHAN, J. J. 1994 Simulating free surface flows with SPH. *Journal of Computational Physics* **110** (2), 399–406.
- MONAGHAN, J. J. 2000 SPH without a tensile instability. *Journal of Computational Physics* **159** (2), 290–311.
- MONAGHAN, J. J. 2005 Smoothed Particle Hydrodynamics. *Reports on Progress in Physics* **68** (8), 1703–1759.
- MONAGHAN, J. J. 2011 A turbulence model for Smoothed Particle Hydrodynamics. *European Journal of Mechanics - B/Fluids* **30** (4), 360–370.
- MONAGHAN, J. J. & GINGOLD, R. A. 1983 Shock simulation by the particle method SPH. *Journal of Computational Physics* **52** (2), 374–389.
- MONAGHAN, J. J. & KOS, A. 1999 Solitary waves on a Cretan beach. *Journal of Waterway, Port, Coastal and Ocean Engineering-ASCE* **125** (3), 145–154.
- MONAGHAN, J. J. & KOS, A. 2000 Scott Russell’s wave generator. *Physics of Fluids* **12** (3), 622–630.

- MONAGHAN, J. J., KOS, A. & ISSA, N. 2003 Fluid motion generated by impact. *Journal of Waterway, Port, Coastal and Ocean Engineering-ASCE* **129** (6), 250–259.
- MONAGHAN, J. J. & PONGRACIC, H. 1985 Artificial viscosity for particle methods. *Applied Numerical Mathematics* **1** (3), 187–194.
- MORRIS, J. P., FOX, P. J. & ZHU, Y. 1997 Modeling low Reynolds number incompressible flows using SPH. *Journal of Computational Physics* **136** (1), 214–226.
- OGER, G., DORING, M., ALESSANDRINI, B. & FERRANT, P. 2006 Two-dimensional SPH simulations of wedge water entries. *Journal of Computational Physics* **213** (2), 803–822.
- OSHER, S. & SETHIAN, J. 1988 Fronts propagating with curvature-dependent speed: Algorithms based on hamilton-jacobi formulations. *Journal of Computational Physics* **79**, 12–49.
- PAKOZDI, C. & GRACZYK, M. 2009 Validation of an sph sloshing simulation by experiments. *ASME Conference Proceedings* **2009** (43451), 729–739.
- PEDERSON, G. & GJEVIK, B. 1983 Run-up of solitary waves. *Journal of Fluid Mechanics* **135**, 283–299.
- RAYLEIGH, L. 1876 On waves. *London, Edinburgh and Dublin Philosophical Magazine and Journal of Science* **Ser. 5, 1** (4), 257–279.
- ROACHE, P. 1998 *Verification and Validation in Computational Science and Engineering*. Hermosapublishers.
- RODRIGUEZ-PAZ, M. & BONET, J. 2005 A corrected Smooth Particle Hydrodynamics formulation of the shallow-water equations. *Computers & Structures* **83** (17-18), 1396–1410.
- ROGERS, B. & DALRYMPLE, R. 2004 SPH modeling of breaking waves. *Coastal Engineering* **1-4**, 415–427.
- ROGERS, B. D. & DALRYMPLE, R. A. 2008 SPH modeling of tsunami waves. *Advanced numerical models for simulating tsunami waves and runup, Advances in coastal and ocean engineering* **10**, 75–100.

- ROGERS, B. D., DALRYMPLE, R. A. & STANSBY, P. K. 2010 Simulation of caisson breakwater movement using 2-d sph. *Journal of Hydraulic Research* **48** (sup1), 135–141.
- ROUBTSOVA, V. & KAHAWITA, R. 2006 The SPH technique applied to free surface flows. *Computers & Fluids* **35** (10), 1359–1371.
- SANDER, J. & HUTTER, K. 1991 On the development of the theory of the solitary wave - a historical essay. *Acta Mechanica* **86** (1-4), 111–152.
- SAVAGE, B. & JOHNSON, M. 2001 Flow over ogee spillway: Physical and numerical model case study. *Journal of Hydraulic Engineering* **127** (8), 640–648.
- SCHOENBERG, I. J. 1946 Contributions to the problem of approximation of equidistant data by analytic functions: part A. *The Quarterly of Applied Mathematics* **IV**, 45–99.
- SCOTT RUSSELL, J. 1838 Report of the committee on waves. *Report of the 7th Meeting of the British Association for the Advancement of Science, Liverpool* pp. 417–496.
- SCOTT RUSSELL, J. 1845 On waves. *Report of the 14th Meeting of the British Association for the Advancement of Science, York* pp. 311–390.
- SHAO, S. 2006 Incompressible SPH simulation of wave breaking and overtopping with turbulence modelling. *International Journal for Numerical Methods in Fluids* **50**, 597–621.
- SHAO, S. & LO, E. Y. M. 2003 Incompressible SPH method for simulating Newtonian and non-Newtonian flows with a free surface. *Advances in Water Resources* **26** (7), 787–800.
- SHUTO, N. 1985 The Nihonkai-Chubu earthquake tsunami on the North Akita coast. *Coastal Engineering in Japan* **28**, 255–264.
- SPENTZA, E. 2006 Laboratory study of 3-D breaking waves. Internal report. Department of Civil & Environmental Engineering, Imperial College London.
- STANSBY, P. 2003 Solitary wave run up and overtopping by a semi-implicit finite-volume shallow-water Boussinesq model. *Journal of Hydraulic Research* **41** (6), 639–647.

- STOKES, G. G. 1847 On the theory of oscillatory waves. *Transactions of the Cambridge Philosophical Society* **8**, 441–455.
- STOKES, G. G. 1880 *On the theory of oscillatory waves, Appendix B*, , vol. 1, pp. 314–326. Cambridge University Press.
- SUBRAMANYA, K. 2001 *Flow In Open Channels, Second Edition*. Tata McGraw-Hill Education.
- SYNOLAKIS, C. 1987 The runup of solitary waves. *Journal of Fluid Mechanics* **185**, 523–545.
- TANUKA, M. 1986 The stability of solitary waves. *Physics of Fluids* **29**, 650–655.
- UBBINK, O. 1997 Numerical prediction of two fluid systems with sharp interfaces. PhD thesis, Imperial College of Science, Technology and Medicine, University of London.
- VACONDIO, R., MIGNOSIA, P., ROGERS, B. & STANSBY, P. 2010 SPH shallow water equation solver for real flooding simulation. In *5th International SPHERIC Workshop*.
- VACONDIO, R., ROGERS, B. D. & STANSBY, P. K. 2012 Smoothed particle hydrodynamics: Approximate zero-consistent 2-d boundary conditions and still shallow-water tests. *International Journal for Numerical Methods in Fluids* **69** (1), 226–253.
- VAUGHAN, G. L., HEALY, T. R., BRYAN, K. R., SNEYD, A. D. & GORMAN, R. M. 2004 An SPH numerical wave tank. *Civil Engineering in the Oceans VI* **182**, 95–107.
- VILA, J. P. 1999 On particle weighted methods and Smoothed Particle Hydrodynamics. *Mathematical models and Methods in Applied Sciences* **9** (2), 161–209.
- VIOLEAU, D. & ISSA, R. 2007 Numerical modelling of complex turbulent free-surface flows with the SPH method: an overview. *International Journal for Numerical Methods in Fluids* **53** (2), 277–304.
- XU, R., STANSBY, P. & LAURENCE, D. 2009 Accuracy and stability in incompressible SPH (ISPH) based on the projection method and a new approach. *Journal of Computational Physics* **228** (18), 6703 – 6725.

- YATSUDA, T., MUTSUDA, H. & MIZUTANI, N. 1997 Kinematics of overturning solitary waves and their relations to breaker types. *Coastal Engineering* **29**, 317–346.
- ZELT, J. A. 1991 The run-up of nonbreaking and breaking solitary waves. *Coastal Engineering* **15**, 205–246.
- ZHENG, J., SOE, M. M., ZHANG, C. & HSU, T. 2010 Numerical wave flume with improved smoothed particle hydrodynamics. *Journal of Hydrodynamics, Ser. B* **22** (6), 773 – 781.