Modelling Multi-phase Non-Newtonian Flows using Incompressible SPH

A thesis submitted to the University of Manchester for the degree of Doctor of Philosophy in the Faculty of Engineering and Physical Sciences

2015

By
Antonios Xenakis
School of Mechanical, Aerospace and Civil Engineering
## Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contents</td>
<td>2</td>
</tr>
<tr>
<td>List of Published Works</td>
<td>7</td>
</tr>
<tr>
<td>List of Tables</td>
<td>8</td>
</tr>
<tr>
<td>List of Figures</td>
<td>10</td>
</tr>
<tr>
<td>List of Symbols and Operators</td>
<td>17</td>
</tr>
<tr>
<td>List of Abbreviations</td>
<td>20</td>
</tr>
<tr>
<td>Abstract</td>
<td>21</td>
</tr>
<tr>
<td>Declaration</td>
<td>22</td>
</tr>
<tr>
<td>Copyright</td>
<td>23</td>
</tr>
<tr>
<td>Acknowledgements</td>
<td>24</td>
</tr>
<tr>
<td>1 Introduction</td>
<td>26</td>
</tr>
<tr>
<td>1.1 Background and Motivation</td>
<td>26</td>
</tr>
<tr>
<td>1.2 Aims and Objectives</td>
<td>28</td>
</tr>
<tr>
<td>1.3 Thesis Structure</td>
<td>29</td>
</tr>
<tr>
<td>2 Fluid Dynamics Background</td>
<td>31</td>
</tr>
<tr>
<td>2.1 Introduction</td>
<td>31</td>
</tr>
<tr>
<td>2.2 Governing Equations</td>
<td>31</td>
</tr>
</tbody>
</table>
5.4.1 Bingham fluid dam-break .......................... 99
5.4.2 Power-law dam-break case .......................... 102
5.5 Power-law moulding flow .............................. 103
5.6 Cross model complex fillings ........................... 105
5.7 Quantitative pressure comparisons with WCSPH .............. 109
5.7.1 Still power-law fluid ............................... 110
5.7.2 Dam-break case .................................. 111
5.8 The importance of shifting for non-Newtonian flows .......... 113
5.9 Summary ............................................. 116

6 Extension of INNSPH to Multi-phase Flows .......................... 118
6.1 Introduction ......................................... 118
6.2 Multi-phase formulation ................................. 119
6.3 Shifting treatment for multi-phase flows ..................... 122
6.3.1 Multi-phase shifting formulation ..................... 122
6.3.2 Treatment of the interface .......................... 123
6.4 Limitations ........................................... 125
6.5 INNSPH multi-phase validation and applications .............. 125
6.5.1 Introduction ...................................... 125
6.5.2 Two-phase Poiseuille flows ....................... 126
6.5.2.1 Newtonian flows ................................ 127
6.5.2.2 Newtonian/ Non-Newtonian interaction .......... 129
6.5.3 Two-phase gravity waves ........................... 131
6.5.4 Rayleigh-Taylor instability .......................... 133
6.5.5 Submarine landslide ................................ 139
6.6 Summary ............................................. 144

7 Concluding Environmental Application - Lituya Bay Tsunami and Landslide .................................. 146
7.1 Introduction ......................................... 146
7.2 Real-scale simulation with gravitational acceleration input .... 149
7.3 Pneumatic accelerated rock entry ........................ 156
7.4 Turbulence modelling

7.4.1 Boundary conditions

7.4.2 Validation cases for the $k-\epsilon$ model

7.4.2.1 Turbulent Poiseuille flow

7.4.2.2 Schematic fish-pass

7.4.2.3 Turbulent dam-break

7.5 The Lituya Bay experiment with turbulence implementation

7.6 Saturation of the landslide

7.7 Summary

8 Conclusions & Future Work

8.1 Introductory comments

8.2 Conclusions

8.2.1 Modelling of inelastic non-Newtonian flows using the ISPH with shifting method

8.2.2 Extension to multi-phase flows

8.2.3 The Lituya Bay tsunami and landslide

8.3 Proposed future work

8.3.1 Suggestions for future applications

8.3.2 Suggestions for improvement of the INNSPH method

References

A Derivation of non-Newtonian Analytical Solutions for Single-phase Plane Poiseuille flows


B.1 Bingham fluid

B.2 Power-law fluid

Word count: 63121
List of Published Works

Parts of this research project have been published, as follows:

In peer-reviewed journals:


In specialist conferences:


List of Tables

5.1 Approximate computational times for the different non-Newtonian models with particle spacing of $dx = 0.02$ m and for a physical time period of $1$ s. .............................................................. 90
5.2 $\epsilon_{L_2}$ errors in steady-state velocity of all different non-Newtonian models for particle spacing of $dx = 0.02$ m ............................................................. 95
5.3 $\epsilon_{L_2}$ relative error in steady-state velocity for increasing shear-thinning behaviour for a power-law fluid. ............................................................... 96

6.1 Deviation of the calculated five periods $T_5$ from the analytical solution for the standing gravity waves. Theoretical five periods duration $T_5 = 6.94$ s. ............................................................... 133
6.2 The set of viscosity and yield stress values used in the relevant literature 141
6.3 Estimated deviation of the water free-surface height at $t_1 = 0.4$ s, $t_2 = 0.8$ s and total error for both timesteps. ............................................................... 143

7.1 Rheological parameters for the three inelastic models tested for the Lituya Bay test case and their discrepancy with the findings of Fritz et al. (2001). ............................................................... 152
7.2 Rheological parameters for the three inelastic models tested, with density $\rho = 1650$ kg/m$^3$ and the discrepancy with the findings of Fritz et al. (2001). ............................................................... 159
7.3 $k - \epsilon$ model constants ............................................................... 164
7.4 Rheological parameters for the three inelastic models tested for the Lituya Bay test case at an experimental scale and the discrepancy with the findings of Fritz et al. (2001). ............................................................... 177
7.5 Transition of sediment’s rheological parameters from dry to saturated state.
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Types of inelastic flow behaviour (Chhabra and Richardson, 2008)</td>
<td>38</td>
</tr>
<tr>
<td>3.1</td>
<td>Representation of a typical kernel function</td>
<td>60</td>
</tr>
<tr>
<td>4.1</td>
<td>Computational configuration for Couette flow: Arrows represent the expected velocity profile at steady state.</td>
<td>81</td>
</tr>
<tr>
<td>4.2</td>
<td>Comparative results of transient Couette flow with a steady state Reynolds number $Re = 1$: • Current study results, – series solution results</td>
<td>81</td>
</tr>
<tr>
<td>4.3</td>
<td>Computational configuration for Poiseuille flow: Arrows represent the expected velocity profile.</td>
<td>82</td>
</tr>
<tr>
<td>4.4</td>
<td>Comparative results of transient Poiseuille flow with a steady state Reynolds number $Re = 1$: • Current study results, – series solution results</td>
<td>82</td>
</tr>
<tr>
<td>4.5</td>
<td>Dry dam-break configuration, with $a = 0.1$ m as presented in (Lind et al., 2012)</td>
<td>83</td>
</tr>
<tr>
<td>4.6</td>
<td>Dry bed dam-break: Left: ISPH results of Lind et al. (2012), Right: INNSPH with new stress formulation.</td>
<td>84</td>
</tr>
<tr>
<td>4.7</td>
<td>Wet dam-break configuration as presented in (Lind et al., 2012), with $h_1 = 1$ m, $x_1 = x_2 = 2$ m and $h_2 = 0.1$ m.</td>
<td>85</td>
</tr>
<tr>
<td>4.8</td>
<td>Comparative results of wet bed dam-break case: Overlay INNSPH free-surface particles (purple colour) with results from Lind et al. (2012)</td>
<td>86</td>
</tr>
<tr>
<td>4.9</td>
<td>Detail of the free-surface particles at an evolved stage of the wet dam-break case: Purple colour: INNSPH free-surface particles.</td>
<td>86</td>
</tr>
</tbody>
</table>
5.1 Configuration of Poiseuille flow computational domain: Arrows show the expected velocity profile. .................................................. 90
5.2 Normalized velocity comparison between analytical and INNSPH results for symmetric Bingham Poiseuille flows: • INNSPH results, — analytical solution. .................................................. 91
5.3 Time convergence results, conducted for Bingham model \( (Bn = 3) \). The solid line indicates an order of convergence of approximately 1.2. ............................... 92
5.4 Comparison between original Bingham formulation and bilinear model proposed by Hosseini et al. (2007): • INNSPH results, — analytical solution .................................................. 93
5.5 Normalized velocity comparison between analytical and INNSPH results for symmetric Poiseuille flows: • INNSPH results, — analytical solution. ........................................ 94
5.6 Convergence results of INNSPH, conducted for a power-law model \( (N = 0.5 \text{ and } N = 3) \). The solid line indicates an order of spatial convergence of approximately 0.9 in both cases. ............................... 95
5.7 Configuration of the annular viscometer case ........................................ 97
5.8 Tangential velocity comparison between analytical and INNSPH results for the case of the annular viscometer .................................................. 97
5.9 Tangential velocity comparison between analytical, WCSPH (Capone et al., 2010) and INNSPH results for a Bingham fluid ............................... 98
5.10 Configuration of dam-break computational domain: grey area represents the initial configuration of the Bingham fluid, while the dashed line shows its predicted profile evolution. ........................................ 99
5.11 Comparison between experimental results (Komatina and Jovanovic, 1997) and INNSPH results for a Bingham fluid: ■ experimental validation results, solid line — INNSPH results. ............................... 100
5.12 Pressure contours of INNSPH method for a Bingham dam-break case (Komatina and Jovanovic, 1997), and comparisons with the free-surface profiles of (Hosseini et al., 2007): a) INNSPH method, b) computational results of (Hosseini et al., 2007). ............................... 101
5.13 Pressure contours and free-surface profiles comparison between INNSPH and CVFEM: a) INNSPH method, b) CVFEM. ............................ 101

5.14 Dam-break propagation comparisons with computational results (Hosseini et al., 2007): ■ computational results (Hosseini et al., 2007), solid line – INNSPH results. ........................................ 102

5.15 Configuration of the power-law moulding flow computational domain as described in (Fan et al., 2010) ........................................ 103

5.16 Detail of the pressure field predicted by the INNSPH methodology for the case of the power-law fluid moulding flow, at time $t = 0.05$ s .................. 104

5.17 Representation of flow profiles and pressure contours of INNSPH method for a power-law moulding flow case (Fan et al., 2010) and comparisons with computational results (Fan et al., 2010): a) INNSPH method, b) computational results of Fan et al. (2010) .................. 104

5.18 Free-surface and pressure contour comparison between INNSPH and CVFEM for the case of power-law moulding flow at 1) $t = 0.01$ s and 2) $t = 0.03$ s: a) INNSPH method, b) CVFEM. ............................. 104

5.19 Configuration of the Cross-model moulding flow geometry as presented in Ren et al. (2012). .................................................. 106

5.20 Comparison of the pressure contours of the WCSPH (Ren et al., 2012) and the INNSPH methods during the filling process of a symmetric mould with a Cross model shear-thinning liquid: a) INNSPH method, b) computational results of Ren et al. (2012). .................. 107

5.21 Comparison of the pressure contours of the WCSPH (Ren et al., 2012) and the INNSPH methods during the filling process of an asymmetric mould with a Cross model shear-thinning liquid: a) INNSPH method, b) computational results of Ren et al. (2012). .................. 107

5.22 Comparison of the relative $L_2$ error evolution for a period of 10 s between INNSPH and WCSPH techniques for still power-law liquids. ............. 111

5.23 Comparison of free-surface profiles by INNSPH and WCSPH for the Komatina and Jovanovic dam-break case (Komatina and Jovanovic, 1997). 112
5.24 Detail of calculated pressures by INNSPH and WCSPH at bottom left corner of dam-break flow presented in Figure 5.23: a) INNSPH, b) WCSPH. 112

5.25 Detail of calculated pressures by INNSPH and WCSPH as in Figure 5.24 for the pressure range $800 \text{ Pa} \leq P \leq 1000 \text{ Pa}$: a) INNSPH, b) WCSPH. 112

5.26 Particle distribution for power-law Poiseuille flow ($N = 0.1$ and $F = 80 \text{ m/s}^2$) with and without shifting for $Re = 10$. 114

5.27 Particle distribution for the power-law moulding flow first presented by Fan et al. (2010) with and without shifting for time $t = 0.035 \text{ s}$. 115

5.28 Particle distribution for the Cross complex filling moulding flow (Ren et al., 2012) using INNSPH with and without shifting for time $t = 0.06 \text{ s}$. 116

6.1 Comparison of the effect of density and viscosity smoothing (equations 6.3 and 6.4) in the evolution of the flow for a Rayleigh-Taylor instability case (described in Section 6.5.4). 121

6.2 Comparison of three different shifting approaches for the Rayleigh-Taylor instability case (discussed in detail in Section 6.5.4). 123

6.3 Shifting treatment near and on the interface. 123

6.4 Comparison of the proposed multi-phase method with and without the interface shifting treatment. 124

6.5 Configuration of the Poiseuille flow of two immiscible fluids with $\mu_b > \mu_a$ as presented in Bird et al. (2007): Arrows represent the expected velocity profiles. 127

6.6 Velocity comparisons between ISPH and analytical solutions for two-phase Poiseuille flows: • ISPH, – analytical results. 128

6.7 Convergence results of ISPH, conducted for a viscosity ratio of $\mu_b/\mu_a = 4$. The solid line indicates an order of spatial convergence of approximately 1.0. 128

6.8 Velocity comparisons between ISPH and analytical solutions for two-phase Newtonian/Bingham Poiseuille flows: • ISPH, – analytical results. 130
6.9 Velocity comparisons between ISPH and semi-analytical solutions for two-phase Newtonian/power-law Poiseuille flows: • ISPH, – semi-analytical results. ................................................................. 130
6.10 Gravity waves configuration of the computational domain .............. 132
6.11 Deviation of the calculated five periods $T_5$ from the analytical solution. Solid line represents rate of convergence, which in this case is approximately 1.0. ................................................................. 132
6.12 Configuration of the Rayleigh-Taylor instability computational domain (Cummins and Rudman, 1999) ................................................................. 135
6.13 Comparison of different resolutions for the for $\bar{t} = 5.0$: a) $dx = 0.02$ m $dx = 0.01$ m $dx = 0.005$ m ................................................................. 136
6.14 Comparison of different resolutions for $\bar{t} = 5.0$ ......................... 136
6.15 Evolution of the Rayleigh-Taylor instability of $Re = 420$, $dx = 0.005$ m. 137
6.16 Comparison of different methodologies for Rayleigh-Taylor of $Re = 420$ at $\bar{t} = 5.0$ ................................................................. 138
6.17 Comparison between current study and Hu and Adams (2007) results for similar number of particles at $\bar{t} = 5.0$ ................................. 138
6.18 Detail comparison with Szewc et al. (2015) for lower and higher particle resolution at $\bar{t} = 5.0$ ................................................................. 139
6.19 Configuration of the submarine landslide computational domain (Assier-Rzadkiewicz et al., 1997) ................................................................. 140
6.20 Comparison between INNSPH, experimental results (Assier-Rzadkiewicz et al., 1997), the WCSPH (Capone et al., 2010) and the density invariant ISPH (Ataie-Ashtiani and Shobeyri, 2008) results. ................................................................. 142
6.21 Water free-surface comparison between INNSPH, experimental results (Assier-Rzadkiewicz et al., 1997), the WCSPH (Capone et al., 2010) and the density invariant ISPH (Ataie-Ashtiani and Shobeyri, 2008) results. ................................................................. 142
6.22 Mud phase comparison between INNSPH, Nasa-Vod2D results (Assier-Rzadkiewicz et al., 1997) and WCSPH results (Capone et al., 2010). . . 144
7.1 Graphical representation of the Lituya Bay event overlaid on an aerial photograph of the Lituya Bay (Fritz et al., 2001; Basu et al., 2010).
7.2 The computational configuration of Lituya Bay landslide (Fritz et al., 2001; Basu et al., 2010).
7.3 Wave run-up for three different rheological models.
7.4 Wave height at \( x = 885 \) m for three different rheological models.
7.5 The shear-rate vs shear-stress relationship of the rock phase using a Hershel-Bulkley rheological model.
7.6 Comparison between INNSPH, the FLOW-3D results by Basu et al. (2010) and the experimental results of Fritz et al. (2001).
7.7 Comparison between INNSPH, the WCSPH method of Schwaiger (Schwaiger, 2007; Schwaiger and Higman, 2007) and the experimental results of Fritz et al. (2001).
7.8 The experimental configuration of Lituya Bay landslide (Fritz et al., 2001; Fritz and Moser, 2003).
7.9 Configuration of the rock phase in the pneumatic accelerator relevant to Figure 7.8.
7.10 Velocity profile of the pneumatic accelerator used in the current work against the velocity profile presented in the experimental work of Fritz and Moser (2003).
7.11 Comparison between INNSPH and experimental results of Fritz et al. (2001) for the rock-phase entry profile versus time at \( x = -67 \) m.
7.12 The flow profile of both phases and the estimation of wave run-up at \( t = 32 \) s after impact.
7.13 Comparison of the turbulent Poiseuille flow with \( Re = 640 \), between the DNS results of Kawamura et al. (2000), the ISPH results of Leroy et al. (2014) and the current work.
7.14 Comparison of the velocity contours for the schematic fish-pass test-case against the experimental results of Tarrade et al. (2008) after time \( t = 20 \) s.
7.15 Comparison of the velocity contours for the schematic fish-pass test-case against the ISPH method of Leroy et al. (2014) for time $t = 20$ s.

7.16 Comparison of the velocity contours for the schematic fish-pass test-case against the WCSPH method of Ferrand et al. (2013) for time $t = 20$ s.

7.17 Comparison of the velocity contours with the WCSPH dam-break case of Violeau and Issa (2007).

7.18 Comparison of the propagation and height of the dam-break against dimensionless time $t^* = t\sqrt{2g/a}$ against the WCSPH of Violeau and Issa (2007) and the experiment of Koshizuka and Oka (1996).

7.19 Comparison between INNSPH and experimental results of Fritz et al. (2001) for the rock-phase entry profile versus time for the experimental scale.

7.20 Estimation of wave run-up including the $k - \epsilon$ turbulence model.

7.21 The shear-rate/shear-stress over yield stress comparison of the rheological models for dry and saturated sediment.

7.22 Comparison between INNSPH with saturation and the experimental results of Fritz et al. (2001).

7.23 Wave run-up at 1.73s intervals: i) The INNSPH results, ii) the experimental results of Fritz et al. (2001).

7.24 Impact of the two phases at 1.73s intervals: i) The INNSPH results, ii) the experimental results of Fritz et al. (2001).
### List of Symbols and Operators

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \mathcal{A} )</td>
<td>Atwood number</td>
</tr>
<tr>
<td>( Bn )</td>
<td>Bingham number</td>
</tr>
<tr>
<td>( C )</td>
<td>Particle concentration</td>
</tr>
<tr>
<td>( c )</td>
<td>Speed of sound</td>
</tr>
<tr>
<td>( c_i )</td>
<td>Colour function of particle ( i )</td>
</tr>
<tr>
<td>( \tilde{c}_i )</td>
<td>Smoothed colour function of particle ( i )</td>
</tr>
<tr>
<td>( C_\mu, C_{\epsilon,1}, C_{\epsilon,2} )</td>
<td>( k - \epsilon ) model constants</td>
</tr>
<tr>
<td>( D )</td>
<td>Shear rate tensor</td>
</tr>
<tr>
<td>( \mathcal{D}' )</td>
<td>Shifting diffusion coefficient</td>
</tr>
<tr>
<td>( dx )</td>
<td>Particle spacing</td>
</tr>
<tr>
<td>( F )</td>
<td>Body-force vector</td>
</tr>
<tr>
<td>( Fr )</td>
<td>Froude number</td>
</tr>
<tr>
<td>( g )</td>
<td>Gravitational acceleration</td>
</tr>
<tr>
<td>( G )</td>
<td>Young’s shear modulus</td>
</tr>
<tr>
<td>( h )</td>
<td>Smoothing length</td>
</tr>
<tr>
<td>( J )</td>
<td>Particle flux</td>
</tr>
<tr>
<td>( k )</td>
<td>Turbulent kinetic energy</td>
</tr>
<tr>
<td>( K )</td>
<td>Cross-model fitting parameter</td>
</tr>
<tr>
<td>( k_l )</td>
<td>Similarity scale factor</td>
</tr>
<tr>
<td>( l )</td>
<td>Characteristic length</td>
</tr>
<tr>
<td>( L )</td>
<td>Channel height</td>
</tr>
<tr>
<td>( M )</td>
<td>Mach number</td>
</tr>
<tr>
<td>( m_j )</td>
<td>Weight of particle ( j )</td>
</tr>
<tr>
<td>Symbol</td>
<td>Definition</td>
</tr>
<tr>
<td>--------</td>
<td>------------</td>
</tr>
<tr>
<td>$N$</td>
<td>Power-law exponent</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
</tr>
<tr>
<td>$P_i$</td>
<td>Production term for $k - \epsilon$ turbulence model</td>
</tr>
<tr>
<td>$R$</td>
<td>Reynolds stress tensor</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$r$</td>
<td>Particle position</td>
</tr>
<tr>
<td>$r_i^*$</td>
<td>Intermediate position of particle $i$</td>
</tr>
<tr>
<td>$r_i^n$</td>
<td>Position of particle $i$ at timestep $n$</td>
</tr>
<tr>
<td>$S$</td>
<td>Mean strain rate tensor</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
</tr>
<tr>
<td>$\bar{t}$</td>
<td>Dimensionless time</td>
</tr>
<tr>
<td>$T$</td>
<td>Period</td>
</tr>
<tr>
<td>$u$</td>
<td>Velocity vector</td>
</tr>
<tr>
<td>$u_i^*$</td>
<td>Intermediate velocity of particle $i$</td>
</tr>
<tr>
<td>$u_i^n$</td>
<td>Velocity of particle $i$ at timestep $n$</td>
</tr>
<tr>
<td>$\tilde{u}$</td>
<td>Normalized velocity</td>
</tr>
<tr>
<td>$u_s$</td>
<td>Shear velocity</td>
</tr>
<tr>
<td>$v_s$</td>
<td>Shifting velocity</td>
</tr>
<tr>
<td>$V$</td>
<td>Particle volume</td>
</tr>
<tr>
<td>$W$</td>
<td>Kernel function</td>
</tr>
<tr>
<td>$We$</td>
<td>Weber number</td>
</tr>
<tr>
<td>$y^+$</td>
<td>Dimensionless distance to wall</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>Bingham-bilinear model constant</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>WCSPH equation of state exponent</td>
</tr>
<tr>
<td>$\delta r_s$</td>
<td>Shifting distance</td>
</tr>
<tr>
<td>$\Delta t$</td>
<td>Timestep size</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>Turbulent kinetic dissipation</td>
</tr>
<tr>
<td>$\epsilon_{L2}$</td>
<td>Normal $L2$ error</td>
</tr>
<tr>
<td>$\eta$</td>
<td>Small number to avoid singularity</td>
</tr>
<tr>
<td>$\kappa$</td>
<td>Von Karman constant</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>Relaxation time for visco-elastic flows or wave length</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Dynamic viscosity</td>
</tr>
<tr>
<td>$\mu_T$</td>
<td>Eddy dynamic viscosity</td>
</tr>
<tr>
<td>$\mu_{\text{eff}}$</td>
<td>Effective dynamic viscosity</td>
</tr>
<tr>
<td>$\mu_0$</td>
<td>Zero shear rate viscosity asymptote</td>
</tr>
<tr>
<td>$\mu_\infty$</td>
<td>Infinite shear rate viscosity asymptote</td>
</tr>
<tr>
<td>$\mu_a$</td>
<td>Viscosity of phase $a$</td>
</tr>
<tr>
<td>$\mu_b$</td>
<td>Viscosity of phase $b$</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity</td>
</tr>
<tr>
<td>$\nu_T$</td>
<td>Eddy kinematic viscosity</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
</tr>
<tr>
<td>$\rho_0$</td>
<td>Reference density</td>
</tr>
<tr>
<td>$\rho_a$</td>
<td>Density of phase $a$</td>
</tr>
<tr>
<td>$\rho_b$</td>
<td>Density of phase $b$</td>
</tr>
<tr>
<td>$\sigma_k, \sigma_\epsilon$</td>
<td>$k - \epsilon$ model constants</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Stress-tensor</td>
</tr>
<tr>
<td>$\dot{\tau}$</td>
<td>Time derivative of stress-tensor</td>
</tr>
<tr>
<td>$\dot{\tau}$</td>
<td>The upper-convected time derivative of the stress tensor</td>
</tr>
<tr>
<td>$\tau_{Ryx}$</td>
<td>The Reynolds or turbulent shear stress</td>
</tr>
<tr>
<td>$\tau_Y$</td>
<td>Yield stress</td>
</tr>
<tr>
<td>$\overline{\tau}$</td>
<td>Turbulent shear stress</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Angular velocity</td>
</tr>
</tbody>
</table>
# List of Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALE</td>
<td>Arbitrary Lagrangian-Eulerian</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CFL</td>
<td>Courant-Friedrichs-Lewy</td>
</tr>
<tr>
<td>CVFEM</td>
<td>Control Volume Finite Element Method</td>
</tr>
<tr>
<td>DEM</td>
<td>Diffuse Element Method</td>
</tr>
<tr>
<td>DLSM</td>
<td>Discrete Least Squares Meshless</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>FDM</td>
<td>Finite Difference Method</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite Element Method</td>
</tr>
<tr>
<td>FVM</td>
<td>Finite Volume Method</td>
</tr>
<tr>
<td>FVPM</td>
<td>Finite Volume Particle Method</td>
</tr>
<tr>
<td>GPU</td>
<td>Graphic Processor Unit</td>
</tr>
<tr>
<td>INNSPH</td>
<td>Incompressible non-Newtonian Smoothed Particle Hydrodynamics</td>
</tr>
<tr>
<td>ISPH</td>
<td>Incompressible Smoothed Particle Hydrodynamics</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>LS</td>
<td>Level Set</td>
</tr>
<tr>
<td>MAC</td>
<td>Marker and Cell</td>
</tr>
<tr>
<td>MPS</td>
<td>Moving Particle Semi-implicit</td>
</tr>
<tr>
<td>PIC</td>
<td>Particle-in-cell</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds averaged Navier-Stokes</td>
</tr>
<tr>
<td>SPH</td>
<td>Smoothed Particle Hydrodynamics</td>
</tr>
<tr>
<td>UCM</td>
<td>Upper-Convected Maxwell</td>
</tr>
<tr>
<td>VOF</td>
<td>Volume-of-Fluid</td>
</tr>
<tr>
<td>WCSPH</td>
<td>Weakly Compressible Smoothed Particle Hydrodynamics</td>
</tr>
</tbody>
</table>
Abstract

University of Manchester
Antonios Xenakis
Doctor of Philosophy
Modelling multi-phase non-Newtonian flows using incompressible SPH
December, 2015

Non-Newtonian fluids are of great scientific interest due to their range of physical properties, which arise from the characteristic shear stress-shear rate relation for each fluid. The applications of non-Newtonian fluids are widespread and occur in many industrial (e.g. lubricants, suspensions, paints, etc.) and environmental (e.g. mud, ice, blood, etc.) problems, often involving multiple fluids. In this study, the novel technique of Incompressible Smoothed Particle Hydrodynamics (ISPH) with shifting (Lind et al., J. Comput. Phys., 231(4):1499-1523, 2012), is extended beyond the state-of-the-art to model non-Newtonian and multi-phase flows. The method is used to investigate important problems of both environmental and industrial interest.

The proposed methodology is based on a recent ISPH algorithm with shifting with the introduction of an appropriate stress formulation. The new method is validated both for Newtonian and non-Newtonian fluids, in closed-channel and free-surface flows. Applications in complex moulding flows are conducted and compared to previously published results. Validation includes comparison with other computational techniques such as weakly compressible SPH (WCSPH) and the Control Volume Finite Element method. Importantly, the proposed method offers improved pressure results over state-of-the-art WCSPH methods, while retaining accurate prediction of the flow patterns.

Having validated the single-phase non-Newtonian ISPH algorithm, this develops a new extension to multi-phase flows. The method is applied to both Newtonian/Newtonian and Newtonian/ non-Newtonian problems. Validations against a novel semi-analytical solution of a two-phase Poiseuille Newtonian/ non-Newtonian flow, the Rayleigh-Taylor instability, and a submarine landslide are considered. It is shown that the proposed method can offer improvements in the description of interfaces and in the prediction of the flow fields of demanding multi-phase flows with both environmental and industrial application.

Finally, the Lituya Bay landslide and tsunami is examined. The problem is approached initially on the real length-scales and compared with state-of-the-art computational techniques. Moreover, a detailed investigation is carried out aiming at the full reproduction of the experimental findings. With the introduction of a $k - \epsilon$ turbulence model, a simple saturation model and correct experimental initial conditions, significant improvements over the state-of-the-art are shown, managing an accurate representation of both the landslide as well as the wave run-up.

The computational method proposed in this thesis is an entirely novel ISPH algorithm capable of modelling highly deforming non-Newtonian and multi-phase flows, and in many cases shows improved accuracy and experimental agreement compared with the current state-of-the-art WCSPH and ISPH methodologies. The variety of problems examined in this work show that the proposed method is robust and can be applied to a wide range of applications with potentially high societal and economical impact.
Declaration

No portion of the work referred to in this report has been submitted in support of an application for another degree or qualification of this or any other university or other institution of learning.
Copyright

1. The author of this thesis (including any appendices and/or schedules to this thesis) owns certain copyright or related rights in it (the Copyright) and s/he has given The University of Manchester certain rights to use such Copyright, including for administrative purposes.

2. Copies of this thesis, either in full or in extracts and whether in hard or electronic copy, may be made only in accordance with the Copyright, Designs and Patents Act 1988 (as amended) and regulations issued under it or, where appropriate, in accordance with licensing agreements which the University has from time to time. This page must form part of any such copies made.

3. The ownership of certain Copyright, patents, designs, trade marks and other intellectual property (the Intellectual Property) and any reproductions of copyright works in the thesis, (for example graphs and tables the Reproduction), which may be described in this thesis, may not be owned by the author and may be owned by third parties. Such Intellectual Property and Reproductions cannot and must not be made available for use without the prior written permission of the owner(s) of the relevant Intellectual Property and/or Reproductions.

4. Further information on the conditions under which disclosure, publication and commercialisation of this report, the Copyright and any Intellectual Property and/or Reproductions described in it may take place is available in the University IP Policy (see http://documents.manchester.ac.uk/display.aspx?DocID=487), in any relevant Thesis restriction declarations deposited in the University Library, The University Librarys regulations (see http://www.manchester.ac.uk/library/aboutus/regulations) and in The Universitys policy on presentation of Theses.
Acknowledgements

This section is dedicated to all those without whom this thesis would never have been realised. First of all I would like to thank my three supervisors Dr Steven J. Lind, Dr Benedict D. Rogers and Prof. Peter K. Stansby for their continuous support, their invaluable comments and their ideas which formed this work. I am very grateful for working with them and for everything they have taught me. Special thanks should go to Steve, with whom I started this journey at the Manchester Metropolitan University. As his first Ph.D. student I hope I made him proud.

I would also like to thank the SPH group of the University of Manchester and especially Dr Athanasios Mokos and Dr Georgios Fourtakas for their fruitful comments and their support during my Ph.D. studies. Moreover, I would like to thank the members of the Centre for Mathematical Modelling and Flow Analysis at the Manchester Metropolitan University and especially Prof. Clive Mingham, who welcomed and supported me in the early stages of this work.

I would like to thank the school of Mechanical, Aerospace and Civil engineering for funding this research project.

Given that the doctorate degree marks the end of my academic studies, I would like to take the opportunity and thank my teachers and my mentors through the years. Special thanks should go to Prof. Sokratis Tsangaris and Prof. Dimitris Christakis, two key figures in my involvement with research and particularly with computational fluid dynamics. Without them I would probably not have the chance to practise what I currently so much enjoy.

Finally, I would like to thank and dedicate this work to the people that are closest to my heart. Firstly, I would like to thank my parents Markos and Sophia and my sister Anastasia and her family Marmar and Marious for their support, care and encourage-
ment (Αρχικά θα ήθελα να ευχαριστήσω τους γονείς μου Μάρκο και Σοφία, την
αδελφή μου Αναστασία και την οικογένεια της Μαρμάρ και Μάριους για τη δι-κή τους υποστήριξη, φροντίδα και ενδυνάμωση). Moreover, my friends Thanos,
Theodora, Thanasis, Grigoris, Savvas and all the others who have been there for my
couragement, support and to share happy times. I would also like to thank Katerina
who although has been as far away as Crete she always managed to be mentally close
to me and put a big smile on my face. Last but not least, I would like to thank the
people of the University of Manchester TaeKwon-Do group for helping me release my
stresses and have fun times here in Manchester.
Chapter 1

Introduction

1.1 Background and Motivation

Free-surface and multi-phase non-Newtonian flows exist both in nature and industrial environments and are often associated with incidents of potentially high industrial and sociological impact. Examples of such flows in the environment include terrestrial and submarine landslides (often associated with tsunami generation (Tappin et al., 2001)), which can be an immediate threat to large numbers of people, or mud flows which can cause serious damage to infrastructures (bridges, roads, dwellings). Likewise, phenomena such as scouring and erosion pose considerable engineering challenges as significant amounts of soil or mud can relocate or change shape and can hamper the rigidity of structures like offshore wind turbines or pipelines mounted on the seabed.

In industrial activity, free-surface and multi-phase non-Newtonian flows are also found in many applications. Examples of such flows include lubricant and suspension flows, as well as moulding flows, which usually are involved in demanding processes, often exhibiting high deformations and pressure distributions. Furthermore, applications in the food industry (ketchup, syrups, mayonnaise (Rao, 2007)) as well as flows that include polymeric melts and solutions (including polystyrene, nylon, rubber (Tropea et al., 2007)) are other typical examples of industrial non-Newtonian flows.

Non-Newtonian applications involving free-surface and multi-phase physics can be very complex to study, since highly transient phenomena and substantial deformation of the interfaces are involved. Moreover, the consequences of the aforementioned events
on economic and industrial activity and societal welfare can be crucial (e.g. Agrawal et al., 2013; Aldrich and Sawada, 2015; Liu et al., 2015), making the research around such flows an important focus of the scientific community.

A common way to study such problems is through computational simulations and the science of Computational Fluid Dynamics (CFD). The main approaches to CFD analysis use computational grids or meshes, where the flow domain is discretized using tessellating cells, and mesh-free approaches, where the computational points (commonly referred to as particles) move and follow the fluid motion.

The finite volume method (Leveque, 2002), finite difference method (Mitchell and Griffiths, 1980) and finite element method (Zienkiewicz and Taylor, 2000) are traditional grid-based methods used to simulate flow problems. Nevertheless, it is widely acknowledged (e.g. D’Amato and Vnere, 2013; Tezduyar, 2006) that modelling free-surface and/or interface problems with such techniques can be quite challenging and computationally expensive, because of the necessarily complicated treatment of interfaces (e.g. re-meshing, volume fraction tracking).

On the other hand, particle-based methods like the moving particle semi implicit (MPS) method (Koshizuka and Oka, 1996) and Smoothed Particle Hydrodynamics (SPH) (Gingold and Monaghan, 1977; Lucy, 1977) can handle large deformations of the interfaces in a straightforward manner. SPH is a relatively new Lagrangian numerical method, first introduced in 1977 by Gingold and Monaghan (1977) and independently by Lucy (1977), originally to model and solve astrophysical problems. As presented herein, it was later applied to flow simulations due to the straightforward simulation of complex boundaries and interfaces. The advantages of this method over traditional Eulerian computational techniques become profound in cases of large deformations like free-surface flows, or multi-phase flows, where numerical diffusion at interfaces can be severe for grid-based approaches. SPH has shown very promising results for such cases (Ataie-Ashtiani and Shobeyri, 2008; Capone et al., 2010; Zanganeh et al., 2012; Zhu et al., 2010; Lind et al., 2012). Moreover, no additional coupling with a computational grid is needed like other particle methods (for example, marker and cell method (McKee et al., 2008)). The clear potential of the SPH method in simulating the challenging interface deformation seen in environmental flows motivates this research.
Nevertheless, traditional SPH is found to suffer from noisy pressure fields (Lee et al., 2008) due to the approximation of incompressible flows as “weakly compressible” using an equation of state. This weakness has been successfully addressed by the velocity divergence-free Incompressible SPH (ISPH) method of Cummins and Rudman (1999), which was, however, limited to relatively low Reynolds numbers, due to numerical instabilities (Lee et al., 2008).

One of the more recent SPH techniques, with great promise, is the velocity divergence-free ISPH technique employing shifting, firstly introduced by Xu et al. (2009) (see also Xu, 2009) and later developed by Lind et al. (2012) and Skillen et al. (2013), for free-surface Newtonian flows. This method was found to be efficient and stable (Xu, 2009; Xu et al., 2009), while providing a significant improvement in terms of accuracy (Skillen et al., 2013; Lind et al., 2012), especially in the pressure field, which has traditionally been a weak point of the SPH technique.

1.2 Aims and Objectives

The aim of this research project is to develop a novel non-Newtonian and multi-phase physics model within the existing framework of ISPH with shifting (Xu et al., 2009; Lind et al., 2012), and apply the method in complex environmental and industrial applications. To achieve this the following steps are to be undertaken:

- Implement inelastic non-Newtonian models (e.g. Bingham, power-law and others described in Section 2.3.1) and validate for internal (e.g. Poiseuille flow) and free-surface flows (e.g. dam-break).

- Apply the method in complex single-phase flows (like moulding flows) and compare against state-of-the-art SPH results.

- Extend the method for multi-phase flows, with attention given to Newtonian/non-Newtonian interactions.

- Validate the method with a range of appropriate reference solutions and applications (e.g. gravity driven waves, Newtonian/non-Newtonian two-phase Poiseuille flows).
• Apply the method to environmental cases (submarine and terrestrial landslides) considering water (Newtonian) and soil (non-Newtonian) interactions.

1.3 Thesis Structure

This thesis is divided into four key parts:

• The theoretical background (Chapters 2 and 3).

• The introduction of the non-Newtonian rheology in Incompressible SPH (ISPH) (Chapters 4 and 5).

• The extension of the method into multi-phase flows and relevant environmental applications (Chapters 6 and 7).

• The conclusion of this study (Chapter 8).

Specifically, Chapters 2 and 3 give the theoretical background of this work. In preparation for the analysis and discussion in later Chapters, Chapter 2 presents the basic theoretical background of fluid mechanics. The governing equations of incompressible fluid flows are introduced with a detailed explanation of non-Newtonian rheology. Multi-phase flows are also described along with a brief theoretical appraisal of turbulent flows. Finally, the basics of dimensional analysis and similarity are introduced.

In Chapter 3 the background of computational fluid dynamics (CFD) methods is presented. Focus is given on the justification for using ISPH with shifting to model non-Newtonian and multi-phase flows. The main grid-based methods and their mesh-free counterparts are outlined with focus on their deficiencies which motivate the introduction of the novel non-Newtonian and multi-phase method presented in this work.

In Chapter 4 the methodology used in this thesis is explained in detail. The implementation of the non-Newtonian formulation in ISPH with shifting is described. This is followed by a rigorous validation in Chapter 5 for non-Newtonian free-surface flows. Applications included a mud dam-break case and two shear-thinning moulding flows with complex geometries.
Multi-phase physics is then introduced into the model (Chapter 6). The mathematical formulation along with validation results are presented. Namely, novel non-Newtonian solutions to steady-state two-phase Poiseuille flow, the Rayleigh-Taylor instability, gravity driven waves, and a sub-marine landslides are considered. Comparisons are presented against experimental, analytical and computational results, where improvements offered in the prediction of the interface over the state-of-the-art are clearly illustrated.

In Chapter 7 the proposed methodology is applied to the complex case of the Lituya Bay landslide and tsunami. Comparisons are made with computational and experimental results. It is shown that crucial model extensions like the introduction of turbulence and saturation models are essential in fully describing this highly transient impact flow. The final results offer a clear improvement over state-of-the-art CFD methods.

In the final chapter (Chapter 8) the key findings of this thesis are summarised. This includes a critical appraisal of the findings with comment on the contributions of the proposed methodology to the general scientific field of CFD. Suggestions on the possible next steps of this research are also made.
Chapter 2

Fluid Dynamics Background

2.1 Introduction

This chapter presents the fundamental equations describing fluid dynamics. The material presented herein gives an outline of the theoretical background on which this thesis is based. The governing equations are introduced with which non-Newtonian flows can be described. Detailed explanation of non-Newtonian rheology is given along with the relevant mathematical models. Additionally, theoretical aspects of multi-phase flows are introduced, followed by remarks on turbulence and dimensional analysis.

2.2 Governing Equations

2.2.1 The continuum approximation

Fluid flows are commonly studied using the continuum approximation, where the fluid is approximated as a continuous medium rather than a set of molecules. The field values (i.e. the velocity, the pressure, the viscosity and others) are measured at each point in this continuum and their variation in space and time is defined by solving a set of equations in either differential or integral form, named the governing equations of the flow which satisfy the conservations of mass, momentum and energy.
2.2.2 Eulerian and Lagrangian Descriptions of Motion

The continuum approximation is typically used in most analytical and computational fluid dynamics methods. The formulation of the continuum method conservation laws can be presented in two different frameworks, the Eulerian and the Lagrangian.

In the Eulerian frame of reference, the basic conservation laws are derived by discretizing the flow domain using fixed fluid elements. The flow passes through these elements, which may contain different portions of the continuum at different times. In the Eulerian framework the independent variables are the position of the element \( \mathbf{r} \) and the time \( t \), with which the flow velocity can be expressed as \( \mathbf{u} = \mathbf{u}(\mathbf{r}, t) \). Moreover, the change of a variable \( A \) inside a space element can be expressed as

\[
\frac{DA}{Dt} = \frac{\partial}{\partial t} A + (\mathbf{u} \cdot \nabla)A ,
\]

(2.1)

where \( D/Dt = \partial/\partial t + \mathbf{u} \cdot \nabla \) is the material derivative (Emanuel, 1994).

On the contrary, in the Lagrangian framework, the continuous matter is discretized into control masses which are now allowed to flow as the time evolves. Therefore, in the Lagrangian approach the attention is focused on flowing masses of fluid, rather than fixed, undeformable areas of the flow domain as in the Eulerian framework. The conservation laws of mass, momentum and energy now need to be expressed in Lagrangian reference frame in order to accommodate the changing positions of the fluid elements. The independent variables in this framework are the initial positions of the fluid elements \( \mathbf{r}_0 = (x_0, y_0, z_0) \) and the time \( t \). The position at any time \( t \) is given by \( \mathbf{r} = \mathbf{r}(\mathbf{r}_0, t) \). The velocity field can then simply written as

\[
\mathbf{u} = \mathbf{u}(\mathbf{r}_0, t) = \frac{\partial \mathbf{r}}{\partial t} .
\]

(2.2)

It is also worth noting that the Eulerian and the Lagrangian frameworks can be related through the material derivative of equation (2.1), which is derived on the Eulerian framework using the Lagrangian concept of a flowing particle carrying fluid property \( A \) (for more details see Pao, 1967; Emanuel, 1994).

It is understandable that the Lagrangian framework naturally facilitates the exam-
CHAPTER 2 Fluid Dynamics Background

ination of highly transient flows with great deformations. Based on this characteristic of the Lagrangian coordinate system, some very interesting computational methods have been developed, with profound benefits over their Eulerian counterparts, which are discussed later in this thesis (see Section 3.4).

2.2.3 Conservation of Mass

If a flowing portion of a fluid is considered it is observed that regardless of the changes of its shape and size (for compressible flows, with variable fluid density), its mass is conserved throughout the flow. This principle leads to the conservation of mass equation, or continuity equation, which takes the form

$$\frac{D}{Dt} \int_V \rho dV = 0 \, ,$$

where $V$ is the volume of the fluid mass and $\rho$ its density. Using the Reynolds’ transport theorem (Reynolds, 1903) the Lagrangian derivative of $D/Dt$ can be converted to a volume integral containing only Eulerian derivatives, with

$$\int_V \left[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) \right] dV = 0 \, .$$

Since the volume $V$ was arbitrarily chosen the integral must be identically zero. Then the equation expressing the conservation of mass becomes

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \, ,$$

or

$$\frac{D\rho}{Dt} + \rho \nabla \cdot \mathbf{u} = 0 \, .$$

For an incompressible fluid, where the density remains constant during the evolution of the flow (i.e. $D\rho/Dt = 0$), equation (2.6) can be written as

$$\nabla \cdot \mathbf{u} = 0 \, .$$
The conservation of mass equation, expressed either in its general form (2.5) or the incompressible form (2.7) is the first conservation law that the density and the velocity field must satisfy.

2.2.4 Conservation of Momentum

The conservation of momentum equation for fluids, is in fact the application of Newton’s second law of motion on a fluid element. Therefore, a fluid element of a Lagrangian framework is changing its momentum at a rate equal to the total summation of the external forces acting on it. The external forces acting on a fluid mass can be either body forces, like the gravitational force, or surface forces like the viscous and the pressure forces. The net body force acting on the Lagrangian fluid element of volume $V$ can then be written as $\int_V \rho \mathbf{F} dV$, where $\mathbf{F}$ is the total vector of the body forces per unit mass. Similarly, if $\mathbf{P}$ is a vector representing the total surface forces on the surface $S$ of the fluid element, the net surface force on the fluid mass will be $\int_S \mathbf{P} dS$. The momentum contained in the volume $V$ can also be written as $\int_V \rho u dV$, with $\rho u$ representing the momentum per unit mass. Therefore the rate of the change of the momentum of the elementary mass contained in within volume $V$ can be written as,

$$\frac{D}{Dt} \int_V \rho u dV = \int_S \mathbf{P} dS + \int_V \rho \mathbf{F} dV . \quad (2.8)$$

Similarly with the conservation of mass equation, the Reynolds’ transport theorem is applied in conjunction with Gauss’ theorem giving the following representation of the conservation of momentum

$$\rho \left( \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \nabla \cdot \mathbf{\tau} + \rho \mathbf{F} . \quad (2.9)$$

The left hand side of equation (2.9) represents the rate of change of the momentum of a unit volume of the fluid, with the first term accounting for the temporal accelerations and the second representing local advective accelerations. On the right-hand side of equation (2.9) the forces causing the acceleration are represented with the first two terms being the gradient of the surface stresses $\mathbf{P}$ (i.e. pressure $p$ and viscous surface
stresses $\tau$ respectively) and the last term is due to body forces, such as gravity, which act on the mass of the fluid.

### 2.2.5 Conservation of Energy

The conservation of energy law is an application of the first law of thermodynamics in fluid flows. Therefore, the conservation of energy law expresses that the change in the internal energy of an elementary mass of fluid is equal to the total work done and the heat which has been added into the system. The total energy per unit mass contained in an arbitrary volume of fluid $V$ in a Lagrangian frame is $\rho e + \rho u^2/2$, with $e$ being the internal energy per unit mass and $u^2/2$ the kinetic energy per unit mass. The total energy contained in the volume $V$ will therefore be $\int_V (\rho e + \rho u^2/2) dV$. Similarly with the conservation of the momentum principle, the forces that may act on a volume of fluid $V$ may be either surface forces, represented by the vector $P$, or body forces denoted by the vector $F$, which can produce a total work of $\int_S \mathbf{u} \cdot \mathbf{P} dS + \int_V \mathbf{u} \cdot \rho \mathbf{F} dV$. Moreover, the amount of heat leaving the fluid per unit time is $\int_S \mathbf{q} \cdot \mathbf{n} dS$, where $\mathbf{q}$ represents the vector of the conductive heat flux leaving the elementary mass and $\mathbf{n}$ is the unit outward normal. Thus, the energy conservation becomes

$$\frac{D}{Dt} \int_V \left( \rho e + \frac{1}{2} \rho \mathbf{u}^2 \right) dV = \int_S \mathbf{u} \cdot \mathbf{P} dS + \int_V \mathbf{u} \cdot \rho \mathbf{F} dV - \int_S \mathbf{q} \cdot \mathbf{n} dS , \quad (2.10)$$

which finally after applying the Gauss’ and Reynolds’ transport theorems one gets

$$\rho \frac{\partial e}{\partial t} + \rho \nabla \cdot (e \mathbf{u}) = \nabla \cdot (\mathbf{\tau} \cdot \mathbf{u}) - \nabla \cdot \mathbf{q} . \quad (2.11)$$

The left-hand side of equation (2.11) represents the rate of change of internal energy, the first term being the temporal change at a fixed location while the second is due to local convective energy flux due to fluid flow. The right-hand side represents the cause of the change in internal energy, the first being the conversion of mechanical energy into thermal energy, while the second term represents the rate at which heat is being added or lost by conduction from outside.

Although the governing equations presented in this section cover the basic principles
of any problem in fluid mechanics, in this project, only isothermal incompressible flows are considered without any change of the internal energy. Thus, the incompressible continuity equation (2.7) and the momentum equation (2.9) are used in the following forms:

\[ \nabla \cdot u = 0, \quad (2.12) \]

\[ \frac{du}{dt} = -\frac{1}{\rho} \nabla p + \frac{1}{\rho} \nabla \cdot \tau + F, \quad (2.13) \]

where \( u \) is the velocity field, \( \rho \) is the density, \( p \) is the pressure, \( \tau \) is the extra stress tensor and \( F \) the body forces. Equations (2.12) and (2.13) are well known and effective expressions for the modelling of general non-relativistic fluid mechanics. In the next section the non-Newtonian rheology is presented.

**2.3 Non-Newtonian Fluids**

Two of the most important characteristics of fluids are the resistance to change of density, known as *compressibility*, and *viscosity* which is a measure of the diffusion of momentum, also interpreted as the resistance to deformation. Deformation of incompressible fluids occurs due to shear stresses. For most gasses and liquids the rate of their deformation when subject to shear stresses is related to their viscosity, thus,

\[ \text{Shear Stress} = \mu \times \text{Rate of deformation}, \quad (2.14) \]

where \( \mu \) is the viscosity of the fluid. The fluids that exhibit this behaviour for a constant viscosity are named Newtonian fluids. However, there is a selection of fluids both in environmental and industrial applications (like paint, lubricants, blood, mud, ice for example) which do not follow that rule and present a variable viscosity (Bird *et al.*, 1983). Those fluids are named non-Newtonian fluids, and are of great interest to the scientific community due to their unique properties and the variety of applications.

Non-Newtonian fluids can be categorized in the following general classes:

1. Fluids where the rate of shear depends only on the shear stress at any point.

These type of non-Newtonian fluids are usually known as “time-independent”,
“inelastic” or “generalized Newtonian fluids”.

2. Fluids with more complex rheological behaviour, for which their rate of shear depends on conditions like the shearing duration and the material history, in addition to the shear rate at any point. Such fluids are named “time-dependent fluids”. In this category belong the “visco-elastic fluids”, which combine characteristics of both ideal fluids and elastic solids.

It should be noted that although non-Newtonian characteristics are distinguished in the classification above, many non-Newtonian fluids exhibit a combination of these characteristics. Commonly, the dominant rheological characteristic is recognized and taken as a basis for the analysis process.

In the following paragraphs the main mathematical models of some of the most common non-Newtonian fluids will be presented.

### 2.3.1 Inelastic Fluid Behaviour

Inelastic fluid behaviour is very common in many non-Newtonian applications. The inelastic fluid behaviour under shear can be expressed as a function of the shear rate

\[ \tau_{yx} = f(D_{yx}) , \]  

(2.15)

where \( D_{yx} \) is the \( yx \) component of the shear rate tensor \( D \). This equation implies that for a given velocity field, the shear rate \( D \) has a one-to-one relationship with the shear stress and vice versa. Given a shear-rate shear-stress relationship, inelastic non-Newtonian fluids can be categorised as

- **Shear thinning**, or thixotropic when the viscosity decreases (thins) with increasing shear rate.

- **Shear thickening**, or rheopectic when the viscosity increases (thickens) with increasing shear rate.

- **Visco-plastic or Bingham plastics**, which behave as elastic solids for low shear stresses, but when the shear stress exceeds a yield stress \( \tau_Y \), the substance flows either with a constant or a variable viscosity.
CHAPTER 2
Fluid Dynamics Background

2.3.1.1 Shear-thinning or pseudoplastic fluids

One of the most common rheological behaviours are the shear-thinning or pseudoplastic fluids, where the effective viscosity $\mu_{\text{eff}}$ decreases with increasing shear rate (Papanastasiou, 1994). Often this is modelled as the effective viscosity for zero shear rate $\mu_0$ decreasing to a lower valued viscosity at infinite shear rate $\mu_\infty$. Moreover, it is common for such fluids to reach asymptotes both near the zero shear rate and near the infinite shear rate regions, resembling a Newtonian behaviour in those regions (Van Wazer et al., 1963).

As this type of rheological behaviour is common, several different mathematical models have been developed to empirically describe this type of behaviour, some of which are presented bellow.

(i) The power-law or Ostwald de Waele model

A simple mathematical model to express non-Newtonian rheological behaviour is the power-law or Ostwald de Waele model (Malkin, 1994; Macosko, 1994), in which the rate of deformation $D$ is exponentially connected with the shear stress. Thus,

$$\tau_{yx} = \mu(D_{yx})^N,$$ (2.16)
where $N$ is the power-law exponent and $\mu$ acts as a curve-fitting parameter. The effective viscosity of such a fluid will be given by

$$\mu_{\text{eff}} = \frac{\tau_{yx}}{D_{yx}} = \mu(D_{yx})^{(N-1)}, \quad (2.17)$$

For $N < 1$, the fluid exhibits shear-thinning properties
- $N = 1$, the fluid shows Newtonian behaviour
- $N > 1$, the fluid shows shear-thickening behaviour

For a shear-thinning fluid, the power-law exponent $N$ takes values in the region $N \in (0, 1)$, with more extreme shear-thinning behaviour for smaller power-law exponents. Similarly, for a shear-thickening fluid $N \in (1, \infty)$.

(ii) The Carreau viscosity equation

Often, the limiting values of viscosities $\mu_0$ and $\mu_\infty$ may be important to represent accurately the shear-rate shear-stress relationship of an inelastic non-Newtonian fluids. The Carreau viscosity model (Chhabra and Richardson, 2008) incorporates both limiting viscosities $\mu_0$ and $\mu_\infty$:

$$\frac{\mu - \mu_\infty}{\mu_0 - \mu_\infty} = \left\{1 + \left(\lambda D_{yx}\right)^{2}\right\}^{(N-1)/2}, \quad (2.18)$$

where $N(< 1)$ and $\lambda$ are two curve-fitting parameters.

(iii) The Cross viscosity equation

Another common rheological model often used for shear-thinning rheologies is the Cross model (Chhabra and Richardson, 2008; Papanastasiou, 1994):

$$\frac{\mu - \mu_\infty}{\mu_0 - \mu_\infty} = \frac{1}{1 + K (D_{yx})^N}. \quad (2.19)$$

Again, in equation (2.19), $N(< 1)$ and $K$ are two fitting parameters whereas $\mu_0$ and $\mu_\infty$ are the limiting values of the effective viscosity at low and high shear rates, respectively.
2.3.1.2 Shear-thickening or dilatant fluid behaviour

Shear-thickening fluids are similar to shear-thinning fluids in the sense that they do not present any yield stress. Shear-thickening rheological behaviour is characterised by the increase of the effective viscosity with increasing shear-rate. This behaviour in some cases is due to the expansion or dilation of the substance for increasing shear-rates and accordingly such fluids are sometimes termed dilatant fluids (Chhabra and Richardson, 2008). This sub-class of inelastic fluids is perhaps the least widespread, and they are found only in a few applications. Typical examples of materials exhibiting dilatant behaviour include concentrated suspensions of china clay, titanium dioxide and mixtures of corn flour in water (Malkin, 1994).

The most common mathematical expression of shear-thickening fluids is the power-law/Ostwald de Waele model of equation (2.17), with the power-law exponent $N$ taking values greater than unity, i.e. $N > 1$.

2.3.1.3 Visco-plastic fluid behaviour

Visco-plastic fluids are characterised by a yield stress $\tau_Y$, which marks the transition between two very distinctive behaviours of those substances. For shear stresses higher than the yield stress the substance flows, whereas for lower stresses an elastic-solid behaviour is exhibited (Currie, 1993; Fay, 1994).

Visco-plastic materials can also be regarded as shear-thinning in the sense that their effective viscosity is reduced for increasing shear rate. At very low shear rates, the substance exhibits a solid-like behaviour, which could otherwise be thought as a flow with infinite viscosity. For shear stresses above the yield stress the substance is flowing. Clearly $\mu_0 \gg \mu_{\infty}$.

Many models have been proposed over the years for modelling visco-plastic fluid behaviours. Some of the most common are presented here.
(i) The Bingham plastic model

The Bingham plastic model (Currie, 1993) is the simplest representation of visco-plastic behaviour. The mathematical form of the Bingham model is

\[ \tau_{yx} = \tau_Y + \mu D_{yx} \quad \text{for} \ |\tau_{yx}| > |\tau_Y| , \]
\[ D_{yx} = 0 \quad \text{for} \ |\tau_{yx}| < |\tau_Y| , \]

where, \( \tau_Y \) the yield stress and \( \mu \) the plastic viscosity.

(ii) The Herschel-Bulkley fluid model

The Herschel-Bulkley rheological model (Chhabra and Richardson, 2008) is a simple generalization of the Bingham plastic model, where for stresses above the yield stress a power-law effective viscosity is exhibited. The expression of the Herschel-Bulkley model takes the following form:

\[ \tau_{yx} = \tau_Y + \mu (D_{yx})^N \quad \text{for} \ |\tau_{yx}| > |\tau_Y| , \]
\[ D_{yx} = 0 \quad \text{for} \ |\tau_{yx}| < |\tau_Y| . \]

The physical meaning of \( \mu \) and \( N \) in equation (2.21) is similar to that of the power-law model in equation (2.16).

(iii) The Casson fluid model

A rheological model often applied to foodstuffs and biological materials, especially blood, is the Casson fluid model (Chhabra and Richardson, 2008) written as:

\[ (|\tau_{yx}|)^{1/2} = (|\tau_Y|)^{1/2} + (\mu |D_{yx}|)^{1/2} \quad \text{for} \ |\tau_{yx}| > |\tau_Y| , \]
\[ D_{yx} = 0 \quad \text{for} \ |\tau_{yx}| < |\tau_Y| . \]

The comparative performance of the aforementioned models as well as several other models for non-Newtonian behaviour has been thoroughly evaluated in an extensive review paper by Bird et al. (1983), and in other relevant literature (Currie, 1993; Fay, 1994; Van Wazer et al., 1963; Malkin, 1994; Macosko, 1994; Papanastasiou, 1994; Mitsoulis, 2007; Chhabra and Richardson, 2008). The overall conclusion drawn from the relevant literature is that an effective rheological representation of a non-Newtonian
flow can be achieved given that the non-Newtonian model used is appropriately chosen.

### 2.3.2 Elastic fluid behaviour

In many real life applications, mainly in industry, the kinematic history or the time of shearing might be crucial factors affecting the rheological behaviour of fluids (Chhabra and Richardson, 2008). In such cases, the inelastic rheological models are not adequate. Elastic fluid models have been developed for such cases. The elastic rheological behaviour can be divided in thixotropical and rheopexical (Chhabra and Richardson, 2008). The rheological behaviour of thixotropical and rheopexical fluids is subjected to the rate at which the “linkages” break down and reform relative to the shear to which they are subjected.

To understand visco-elastic behaviour one should consider two limiting cases. The first one is the viscous flow of a Newtonian fluid, where the shearing stress is proportional to the shear rate. The other extreme is the elastic solid body, which follows Hooke’s law,

\[
\tau_{yx} = -G \frac{dx}{dy},
\]

where \( G \) is the Young’s shear modulus and \( dx \) is the shear displacement of two elements separated by a distance \( dy \). This body will deform elastically and return into its original shape as long as the yield stress is not surpassed. A visco-elastic material can be considered to be between these two extreme cases, showing both elastic and viscous behaviour (Bird et al., 1983). Many materials of practical interest (such as polymer melts, polymer and soap solutions, synovial fluid) exhibit visco-elastic behaviour (Macosko, 1994).

Although elastic non-Newtonian behaviour is not considered in the context of this thesis, for completeness of the non-Newtonian behaviour overview, some remarks of the most popular elastic mathematical models are presented in the following paragraphs. Perhaps the most well known elastic mathematical model is the Maxwell (Christensen, 1971), in which the viscous and elastic properties are linearly combined. The shear-
stress/shear-rate relationship of a Maxwell fluid is given by:

\[ \tau + \lambda \dot{\tau} = \mu D, \tag{2.24} \]

where \( \dot{\tau} \) is the time derivative of \( \tau \), \( \mu \) is the viscosity of the Maxwell fluid and \( \lambda (= \mu / G) \) is the relaxation time, which is a characteristic of the fluid.

The Maxwell model is also the basis for more complex visco-elastic models such as the Upper-convected Maxwell (UCM) and the Oldroyd-B models (Macosko, 1994). In the Maxwell model and its derivatives, the fluid-like response is the more dominant (Van Wazer et al., 1963). A more solid-like behaviour is obtained by the Voigt model, which is represented by a parallel spring-dashpot arrangement.

Although the visco-elastic models of this section are found in some very interesting applications, they are not going to be the focus of this thesis, due to their complexity and relatively limited applicability in the applications considered. The focus of this work is going to be on the inelastic rheological models presented in Section 2.3.1, which are very commonly used for both environmental and industrial applications and are appropriate for the applications discussed.

### 2.4 Multi-phase flows

Flows incorporating more than one fluid are classified as multi-phase flows. The difference between the multiple phases can be either thermodynamic (e.g. gas, liquid, solid) or in the chemical components of the flow (e.g. flows with different substances) (Papanastasiou, 1994). Furthermore, the thermodynamic state or the chemical composition of the substances involved might change during the flow.

With multi-phase flows extremely complex phenomena can be analysed. In particular multi-phase flows can be found in many industrial and environmental applications such as propulsion systems, vapour-particle deposition, fluid-particle transport, sedimentation, nuclear reactor cooling, thermoclines in oceans, seabed erosion, or meteorological phenomena like rain, snow, cloud formation, avalanches, landslides.

Based on the morphologies of the different phases, two major categories can be
identified: the disperse multi-phase flows and the separated multi-phase flows. In a
two-phase disperse multi-phase flow discontinues particles, bubbles or droplets of the
dispersed phase are distributed in a continuous second phase. In a separated flow all
phases involved are continuous and separated by distinct interfaces (Brennen, 2005).

2.4.1 Multi-phase Flow Models

In order to study multi-phase flows with analytical and computational tools, it is often
useful to make the distinction between disperse and separated flows according to their
physical properties. Of the disperse and the separated multi-phase flows, the first one
is the more challenging for mathematical and computational analysis, since the size of
the disperse matter might be so small to prohibit the direct modelling of its particles.
In such cases trajectory models and two-phase models may be introduced (Brennen,
2005). In trajectory models, the movement of the disperse particles is tracked either by
following the actual particles or larger, representative particles. Drag, lift and moment
forces or even thermal history are calculated and taken into account for the estimation
of the disperse matter trajectories. In two-fluid models, the disperse phase is treated as
a continuous phase which interacts with the continuous matter in which it is dispersed.
Averaging processes are necessary to accurately determine the characteristics of the
disperse phase (Brennen, 2005). Thus, this approach neglects the discrete nature of
the disperse phase and the effects on the continuous phase are approximated.

In contrast, in separated flows the mathematical modelling and computational sim-
ulation is much more straightforward. One option is to solve the typical single-phase
kinematic equations (2.6, 2.9, 2.11) separately for each phase with a coupling at the
interface. Appropriate kinematic and dynamic conditions need to be taken into ac-
count on the interface (Crowe, 2005). Another option is to treat the two phases as
a single continuum with an appropriate smoothing treatment at the interfaces, where
the classic conservation equations (2.6, 2.9, 2.11) are solved once for all the phases
involved (Brennen, 2005).


## 2.5 Turbulence

It is well known that there are two main types of viscous flow, the laminar flow and the turbulent flow, which differ in terms of their internal structure. In the laminar flow, the fluid is moving smoothly and in distinctive layers (no diffusion across layers is observed). Thus, in the laminar flow there is no macroscopic mixing in the fluid’s neighbouring layers. The only mixing occurring on this type of flow is on a molecular level (Papanastasiou, 1994).

On the other hand, in turbulent flow the fluid particles move in every direction in a random fashion. The movement of the fluid particles in such a flow closely resembles the thermal movement of molecules. Therefore, in turbulent flow, in addition to the molecular mixing, there is macroscopic mixing (Pope, 2001). In general, turbulent flow appears in flows with viscous shear stresses resulting from velocity discontinuities and high velocity gradients. These types of turbulence might be seen either in the regions of boundary layers close to bodies or in the bulk of the fluid flow.

Turbulence can play a crucial role in many flows, especially in environmental applications, in which it is very rarely absent. Furthermore, due to the relatively high magnitude of stresses which appear in a turbulent flow, turbulence might be an important factor for non-Newtonian flows (e.g. in seabed erosion problems around offshore structures (Dosanjh and Humphrey, 1985)).

The characteristic parameter of a turbulent or a laminar flow is the Reynolds number, $Re$, which is defined with the general equation:

$$Re = \frac{ul}{\nu},$$

(2.25)

where $\nu$ is the kinematic viscosity of the fluid and $u$ and $l$ are the characteristic velocity and the characteristic length scale in the flow, respectively. The characteristic length and velocity are relevant to the geometry of the examined problem (Papanastasiou, 1994).

From a physical point of view, the Reynolds number represents the balance of the inertia forces over the viscous forces acting on the fluid. For low Reynolds numbers the flow is classified as a laminar flow, while for larger Reynolds numbers a turbulent flow
may occur. When a turbulent flow appears the inertia forces are increased to a level that viscous forces cannot damp any velocity disturbances, allowing for the increase of these disturbances. For very large Reynolds numbers these disturbances cover the flow domain in total, and the flow is categorized as a fully developed turbulent flow. The Reynolds number at which the flow has transitioned from laminar to turbulent flow is named the critical Reynolds Number \((Re_c)\). \(Re_c\) depends on the definition of the problem, but in general flows with \(Re > 5000\) are considered turbulent. For Poiseuille flows specifically, turbulence starts to appear at \(Re \approx 2000\) and the transition to turbulence is completed at \(Re \approx 4000\).

2.5.1 Turbulent stresses

It has been found experimentally, that the shear stresses generated in a viscous flow increase substantially from laminar to turbulent flow (Reynolds, 1894). This phenomenon is due to the turbulent velocity fluctuations \(u'\), where fluid particles move between neighbouring flow layers. Assuming a fluid particle that moves from layer 1, with average velocity\(^1\) on the \(x\) direction \(\bar{u}_{x,1}\), to layer 2 with average velocity \(\bar{u}_{x,2}\), so that the average velocity has only an \(x\) component, the velocity difference \(\bar{u}_{x,1} - \bar{u}_{x,2}\) is shown as a velocity fluctuation. This velocity fluctuation, and therefore the momentum fluctuation at layer 2, can be attributed to the act of the turbulent shear stress or Reynolds stress \(\tau_{Rxy}\) of one layer to the other. The magnitude of this stress is equal to (Reynolds, 1894)

\[
\tau_{Rxy} = -\rho u'_x u'_y .
\] (2.26)

2.6 Dimensional Analysis and Similarity

Using the governing equations of Section 2.2 or results from experiments and computations, almost any flow problem can be modelled. In terms of efficiency, however, it is worth optimizing the analysis process, so that the data and the results collected from a few calculations or experiments can be related to as many different problems as

\(^1\)In this case the velocity average denote the ensemble average, which is a statistical average taken upon a large number of independent occurrences (Lesieur, 2008).
possible. This can be achieved with the method of dimensional analysis. It should be noted that this method is not a solution method, but instead it enables general flow characterization.

Dimensional analysis aims to reduce the number of independent variables used to describe a phenomenon. This is achieved by using dimensionless parameters which are determined by the actual physical and geometrical parameters. The number of these dimensionless parameters is always smaller than the dimensional parameters, thus the parameters describing a problem are simplified.

### 2.6.1 Basic dimensionless parameters

The dimensionless parameters that can be used vary according to the problem that is examined. Many different sets of dimensionless parameters can be used and they are chosen based on suitability to the characteristics of the flow. These parameters are usually named dimensionless “numbers” and they are very important for the categorization of the flow. Some of the most common dimensionless numbers will be presented in the following paragraphs.

Perhaps the most common dimensionless parameter is the Reynolds number, $Re$ presented in equation (2.25) which characterises the flow regimes for laminar and turbulent flow.

Another very useful parameter is the Mach number, $M$:

$$M = \frac{u}{\sqrt{E/\rho}},$$

where $E$ is the modulus of bulk elasticity for fluids. The Mach number is very important in compressible flow problems. The denominator of equation (2.27) expresses the speed of sound. For Mach numbers $M \leq 0.3$ the compressibility effects are usually considered negligible.

Another important parameter especially in flows with interfaces is the Weber number, $We$:

$$We = \frac{\rho u^2 l}{\sigma},$$

where $\sigma$ is the surface tension of the fluid. Physically, the meaning of Weber number
is the balance of the inertia force over the surface tension force. Thus, the Weber number is very important in the development of a multi-phase flow. Note that the Weber number is small when surface tension is important.

For free-surface gravity-driven flows a particularly useful dimensionless parameter is the Froude number, $Fr$:

$$Fr = \frac{u}{\sqrt{gl}}, \quad (2.29)$$

where $g$ is the gravitational acceleration. It should be noted that despite the fact that gravitational acceleration appears in equation (2.29), the Froude number is not related to the hydrostatic pressure. Physically, the Froude number is a measure of the balance of the inertia force over the gravitational force.

The list of the dimensionless parameters presented above is not exhaustive and for specific problems different dimensionless parameters can be introduced. Nevertheless, the parameters presented above are some of the most common ones and they are usually used in the process of dimensional analysis, with most applied throughout this thesis. More details on dimensionless parameters can be found in Currie (1993), Fay (1994), Papanastasiou (1994) and other relevant literature.

In the context of the current work Mach and Weber numbers are not considered, since incompressible flows without surface-tension effects are examined. On the other hand, the Reynolds number of the flows examined varies vastly. Most of the cases examined exist in the laminar regime ($Re < 1000$) since highly-viscous fluids and relatively low velocities are examined. Nevertheless, in some multi-phase flows of environmental interest, higher Reynolds Numbers are calculated (see Chapters 6 and 7). Most of the cases examined herein are also well into the subcritical region with Froude numbers $Fr \ll 1$. However, in Chapter 7 a case with supercritical Froude number is examined with a shallow high velocity environmental flow.

### 2.6.2 Similarity

Similarity refers to the process of relating magnitudes and general datasets between models of different scales. This is particularly useful in the experimental process when simulations are performed in scaled-down or prototype models. There are three main
CHAPTER 2 Fluid Dynamics Background

categories of similarity:

- Geometric,
- Kinematic and
- Dynamic.

The geometric similarity is perhaps the most obvious demand of similarity, since the two related cases are presumed to be similar. For geometric similarity to be valid, both related cases need to have the same shape and every length to be linked with the same scale factor $k_l$:

$$k_l = \frac{l_1}{l_2}. \quad (2.30)$$

In addition to lengths, the scale factor $k_l$ can be used to relate areas and volumes as:

$$k_l = \sqrt{\frac{A_1}{A_2}} = \sqrt[3]{\frac{V_1}{V_2}}. \quad (2.31)$$

Kinematic similarity can be applied to relate flows between a model and a prototype. For the kinematic similarity to be valid, the velocities at relative points should have the same direction and the magnitude of the velocities to obey the scale factor:

$$k_u = \frac{u_1}{u_2}. \quad (2.32)$$

Kinematic similarity demands that the orientation of the flow relative to the boundaries be the same between the two related cases.

Dynamic similarity is the most constricting of the three similarity criteria, since it demands all forces acting at relative points to be parallel and their magnitudes to be connected with scale factor:

$$k_F = \frac{F_1}{F_2}. \quad (2.33)$$

With dynamic similarity it is pre-empted that geometric and kinematic similarities are also satisfied. It should be noted here that universal similarity cannot always be maintained.

In the context of this work the geometric similarity of equation (2.30) is used to relate computational and experimental results to a real scale environmental application
Moreover, a Froude similarity model (Hughes, 1993) is used to relate the kinematic parameters for the aforementioned application.

### 2.7 Summary

In this chapter the principles of fluid mechanics that will be used in this work have been introduced. The equations expressing conservation of mass (2.6), momentum (2.9) and energy (2.11) which characterize any fluid flow have been given. In this work incompressible flows without any change of the internal energy are examined, therefore the equations used are the incompressible continuity equation (2.12) and the momentum equation (2.13).

A thorough explanation of non-Newtonian rheology has been given. The distinction between inelastic and visco-elastic non-Newtonian models was made. Some of the most commonly used rheological models have been presented which cover shear-thinning, shear-thickening, pseudo-plastic and visco-elastic flows. In this work the main focus is on inelastic flows, therefore only rheological models from Section 2.3.1 will be implemented.

Additionally, a brief theoretical background on multi-phase flows was given, highlighting the differences between the main types of multi-phase flow and the different ways of modelling. A brief discussion on turbulent flows has also been presented, which can be relevant to environmental multi-phase non-Newtonian flows. Turbulent effects are introduced in the final stages of this thesis, when a complex environmental application with high Reynolds number has been examined (see Chapter 7). Finally, remarks on the theory of dimensional analysis and similarity have been made, with focus on common dimensional parameters which are going to be used in this work.

In the next chapter an introduction to computational methods for flow analysis is given. The characteristics of grid-based and meshless CFD methods are presented, followed by the advantages and justification for using Smoothed Particle Hydrodynamics (SPH). A detailed presentation of the SPH method is made, along with a literature review, with emphasis on works with non-Newtonian flow modelling and multi-phase flows.
Chapter 3

Computational Fluid Dynamics and the SPH method

3.1 Introduction

The main investigative tools of a fluid mechanics researcher are experimental, theoretical and computational methods. This thesis is based entirely on the computational approach commonly known as computational fluid dynamics (CFD). In this chapter the basics of CFD are introduced, which form the basis on which the work presented herein is developed. Grid-based methods are described, focusing on the deficiencies which have motivated the development of particle-based methods. Special attention is given to the smoothed particle hydrodynamics (SPH) method, including motivation for its use in this thesis. The fundamentals of SPH are explained, while key works from the SPH literature are summarized. Focus is given to SPH studies of non-Newtonian and multi-phase flows and the literature findings are comprehensively explained and discussed. It is shown that the introduction of a particle based method with good representation of the pressure field for non-Newtonian flows (both single-phase and multi-phase) can significantly improve the state-of-the-art. The developed method can then be used to understand important applications (such as highly pressurised moulding flows, violent landslides and tsunami events), which motivates the work carried out for this thesis.
3.2 Computational Fluid Dynamics

Computational fluid dynamics (CFD) is the study of fluid mechanics using computational methods (numerical methods and algorithms) to simulate fluid flows. In principle CFD replaces the conservation equations of fluid mechanics (see Section 2.2) with a set of numerical boundary conditions, which are then advanced in space and time in order to give numerical values to the field variables of the flow based on discrete forms of the conservation equations (2.6, 2.9, 2.11).

In recent decades the exponential growth of computational power has allowed the increased application of CFD in more complex problems. It is widely accepted that CFD can now substitute laboratory experiments in many applications and bring benefits such as increased cost efficiency, straightforward data collection and analysis. Examples in aeronautical (e.g. Obert, 2009; Leschziner, 2006), automotive (e.g. Drake and Haworth, 2007; Muyldermans et al., 2004) and marine (e.g. Ji et al., 2012; Tyagi and Sen, 2006) industry highlight the increasing applicability of CFD.

CFD applications have arguably started even before the invention of the digital computer. A notable example is the pioneering work of Richardson in the 1920s (Richardson, 1922) who introduced numerical methods for weather forecasting (Lynch, 2006). His models were solved by human-computers, who were performing the calculations by hand. Perhaps the first application of digital computers in fluid dynamic problems was made by Kopal as early as 1947 (Kopal, 1947), who studied supersonic flows around sharp cones and used primitive computers to solve the governing differential equations. Other pioneers in CFD included Fay and Riddell (1958), Blottner (1964) and Hall et al. (1962), who studied problems like inviscid flows and boundary layers in the 1950s and 1960s. In recent years the application of CFD methods covers problems from the whole spectrum of fluid mechanics.

As seen in Section 2.2.2 there are two reference frames in which a flow can be analysed, the Eulerian and the Lagrangian frame of reference. Based on these two frameworks two main methodological approaches of CFD techniques have been developed, the grid-based approach and the particle-based approach. In the first, usually a computational grid is constructed with fixed cells or elements. The field variables
are then calculated over each cell/element of the computational domain. On the other hand, particle-based methods are based on the Lagrangian approach where computational particles represent finite masses of the fluid, which are then allowed to flow with the motion of the fluid. In these methods, the field variables express properties of material elements of the continuous matter rather than fixed areas or points in the computational domain. A combination of the two is also possible (e.g. with arbitrary Lagrangian-Eulerian (ALE) methods (Hirt et al., 1974)). It should be noted that although grid-based methods and particle based methods are generally based on the Eulerian and the Lagrangian frames of reference respectively, there are exceptions like the grid-based Finite Element Method which is based on the Lagrangian frame of reference (Zienkiewicz and Taylor, 2000) or the particle based Eulerian SPH method (Lind and Stansby, 2015). Therefore, in the following chapters of this thesis, distinction is going to be made between grid-based and particle-based CFD methods, rather than Eulerian and Lagrangian methods.

In the following paragraphs some of the main grid-based and particle-based approaches are presented with focus on the SPH method.

### 3.3 Grid-based methods

In the grid-based approach, the Finite Element Method (FEM) (Zienkiewicz and Taylor, 2000), Finite Volume Method (FVM) (Leveque, 2002) and Finite Difference Method (FDM) (Mitchell and Griffiths, 1980) are the main computational techniques. All these methods solve the governing equations by dividing the continuum field into cells or elements with triangular, quadrilateral or other shapes (for two dimensional domains) to form a grid or mesh.

The FEM is one of the most used computational approaches in the field of CFD. It originates from the field of structural mechanics, where its characteristic ability to fit complex domains and undergo small deformation easily has given it increased applicability over FDM and FVM (Ferziger, 1981). Nevertheless, FEM is considered more complicated than FVM and FDM, which are usually preferred for complex applications with high number of cells or elements (Wesseling, 2001).
The FVM is another very common computational approach. Its main difference with FEM is that the integral form of the governing equations is discretized on a Eulerian frame of reference rather than the Lagrangian, used by FEM (Ferziger, 1981; Ferziger and Peric, 2002). Conservation of the governing equations is maintained, since the fluxes leaving one cell by default enter adjacent cells. Therefore, physics such as incompressibility can be easily implemented. Compared with the FEM moving boundaries and interfaces are more difficult to model given that the Eulerian approach is used. The computational grid as with FEM can be either structured or unstructured, meaning that the cells can be in irregularly positioned to fit complex boundaries.

The last of the grid-based methods is the FDM, which is also the oldest. Taylor series expansions are used to approximate the conservation equations with finite differences. For FDM unlike FEM and FVM a structured grid is usually preferred, since a high degree of regularity is needed in its simplest implementation. Higher-order differencing approximations are facilitated by FDM, allowing higher orders of accuracy. Nevertheless, it has been found that mass, momentum and energy are not fully conserved (Morinishi et al., 1998) and special treatment is used to enforce the conservation properties. Finally, the equations need to be transformed in terms of a body-fitted coordinate system to fit complex boundaries (Wesseling, 2001).

### 3.3.1 Interface treatment for grid-based methods

Tracking of free surfaces and interfaces is not a straightforward process with the aforementioned grid-based methods, therefore, special techniques are employed for such flows. One of the most common techniques is the adaptive update of the grid, also known as “remeshing”, where the grid is reconfigured to account for the deformations of the computational domain (e.g. Thornton and Vemaganti, 1990; Yue et al., 2006; Wicke et al., 2010). Nevertheless, remeshing can be associated with augmented computational cost as well as accuracy issues (e.g. Najm, 1992). Other mesh-based techniques for interface tracking are the overset grids (Meakin, 1999), the chimera grids (Steger et al., 1983) and the overlapping grids (Chesshire and Henshaw, 1990). In these methods, which are commonly used for interactions of fluids with solids (e.g. a propeller moving in water (Muscari et al., 2011)), multiple distinctive grids are implemented
CHAPTER 3

CFD and the SPH method

which overlap each other, while properties of interacting cells are exchanged by interpolation. A separate treatment is the Arbitrary Lagrangian-Eulerian (ALE) method in which the mesh (which is neither Eulerian or Lagrangian) is advected with a non-flow velocity allowing a more uniform mesh to be retained (e.g. McDaniel et al., 2006; Psihogios et al., 2015). This technique is advantageous in multi-phase or highly-deformable flows since the mesh element uniformity is better maintained.

Other techniques have also been developed focusing on the tracking of the phases, rather than adapting to the deformation of the mesh. A very popular method, also found on commercial packages (such as ANSYS, OpenFOAM and Star-CCM) is the Volume-of-Fluid method (VOF) (Hirt and Nicholls, 1981). With VOF the free-surface profile or the interface between two fluids can be tracked with the use of volume fraction and the advection equation, but the conservation equations of the flow are solved using one of the aforementioned CFD methods. Another CFD approach which is used to track interfaces, is the level-set (LS) method (Osher and Sethian, 1988). LS uses contours of scalar functions to track the interface. The interface is represented by a so-called level set function (Sethian, 1999). With LS or VOF numerical computations involving curves or highly transient phenomena, such as splitting or merging of flows are easily reproduced, nevertheless, mass-loss is a common phenomenon especially in the LS method (Rider and Kothe, 1995). Marker and cell (MAC) (Harlow and Welch, 1965; McKee et al., 2008) and particle-in-cell (PIC) (Harlow, 1955, 1964; Kelly et al., 2015) are other grid-based methods, which incorporate the use of virtual Lagrangian particles to track different phases. Although particles are used in MAC and PIC they are not considered pure particle based methods, since grid-based algorithms need to be implemented for the solution of a flow.

3.3.2 Limitations of grid-based methods

Grid-based methods have been applied with great success in the simulation of non-Newtonian flows. Indicative works include the work of Li and Kleinstreuer (Li and Kleinstreuer, 2005; Li et al., 2005; Li and Kleinstreuer, 2006) in biomechanical engineering, Mitsoulis et al. (1993) and Blackery and Mitsoulis (1997) in creeping flows and industrial applications and studies of flows in T-Shape geometries by Neofytou
et al. (2014). Furthermore, a number of commercial grid-based software packages such as ANSYS, Star-CCM and OpenFOAM offer several non-Newtonian rheological models, or allow the implementation of such models. In general, grid-based methods, like the aforementioned, can be very effective for flows which take place in closed domains (e.g. channels, tubes, vanes), but the implementation of multi-phase problems or free-surface single-phase problems can be problematic and computationally expensive. To study such flows with grid-based methods would require either “re-meshing” of the grid, where the grid is renewed every few timesteps so that the mesh deformations of the interfaces can be accommodated, or coupling with a phase tracking method (see Section 3.3.1). Both aforementioned approaches are often found to be inefficient, therefore hampering the applicability of grid-based methods. As shown in the following paragraphs (Section 3.4) Lagrangian mesh-free methods offer straightforward representation of the interfaces and accommodate even high deformations with relative ease while conserving key quantities, such as the mass in either phase.

### 3.4 Particle-based methods

The FEM, FVM and FDM presented in Section 3.3 are the most popular CFD techniques. However, the use of a computational mesh makes it inherently difficult to examine problems with highly nonlinear deformations like free-surface flows, or multi-phase flows. Numerical diffusion across interfaces tends to be severe, which is often associated with poorly conserved governing equations (mass-loss) or lack of interface resolution. On the other hand, particle-based methods use computational particles instead of a fixed mesh. Given that computational points in particle-based methods flow with the matter, they are exceptionally flexible at modelling coalescing or fragmented regions of a flow. Some of the main particle-based methods to be presented here are the moving particle semi implicit (MPS) method (Koshizuka and Oka, 1996), the discrete least squares meshless method (DLSM) (Arzani and Afshar, 2006), the diffuse element method (DEM) (Nayroles et al., 1992) and SPH (Gingold and Monaghan, 1977; Lucy, 1977).

MPS is one very common Lagrangian method to simulate incompressible free-
surface flows. This method calculates the field variables through integral interpolants over computational particles. In MPS a weighted averaging process is used to express differential operators, without the need of a gradient of a kernel function (Koshizuka and Oka, 1996). It is worth noting the recent work of Souto-Iglesias et al. (2013; 2014) who have shown the close mathematical resemblance between SPH and MPS.

DLSM method is a recent meshless method, based on the approach of least squares. In the DLSM, particles or nodal points are used to solve the differential form of the governing equations. A least-square function is used to minimise the weighted summation of the squared residual of the governing differential equations and its boundary conditions at the computational particles (Arzani and Afshar, 2006).

The DEM is another particle based method, having several similarities with SPH, which will be discussed later. Again this method uses a least squares approximation to solve partial differential equations (Nayroles et al., 1992) and it has been presented as a generalization of the grid-based FEM or a mesh-free FEM counterpart. This method should not be confused with the discrete element method (Cundall and Strack, 1979; Williams et al., 1985, also shortened to DEM) which is based on mass-spring-damper interactions and extensively used in granular flows (e.g. Cleary and Sawley, 2002; Teufelsbauer et al., 2011), and is often coupled with other CFD techniques (Amritkar et al., 2014; Robinson et al., 2014).

Of all the particle methods, SPH is one of the most popular. Similar to MPS, integral interpolants over computational particles are used to estimate the properties and solve the governing equations. A kernel weighted function is used to regulate the interpolation between the computational particles, but unlike MPS the gradient of the kernel is used to solve any differential operators.

Due to the straightforward prediction of interfaces and the ease of simulating high deformations, coupled with recent advances which allow the accurate prediction of pressure fields in Newtonian flows (Lind et al., 2012; Skillen et al., 2013), SPH is chosen for this project as the computational technique to simulate non-Newtonian and multi-phase incompressible flows. In the following section a detailed presentation of the SPH method and its fundamental characteristics is given.

It should be noted that the above overview on mesh-free methods is not exhaustive.
and further discussion can be found in the relevant literature (e.g. Fries and Matthies, 2004; Li and Liu, 2007; Liu and Liu, 2003; Chen et al., 2006).

3.5 The Smoothed Particle Hydrodynamics (SPH) Method

3.5.1 Introduction

SPH is a relatively new computational technique first introduced in 1977 by Gingold and Monaghan (1977) and separately by Lucy (1977) originally to model and solve non-axisymmetric astrophysical problems. However, its advantages of accurately and easily describing complex boundaries and interfaces for engineering flows became clear in the 1990s. SPH is a Lagrangian particle method, where the variables of a fluid particle are expressed through integral interpolations which are approximated as summed interpolants over neighbouring fluid particles.

The advantages of this approach in comparison with classical Eulerian methods, like Finite Volume Method (FVM), Finite Difference Method (FDM) and Finite Element Method (FEM), become apparent when examining problems with major deformations like free-surface flows, or multi-phase flows. Given that computational points in SPH flow with the matter, it is exceptionally flexible at modelling coalescing or fragmented regions of a flow. Thus, SPH is a suitable tool to examine complex cases like solid fractures, or fluid flows such as water spraying, wave impacts, scouring phenomena, moulding flows, high pressure injections etc. In addition, SPH has a significant advantage when including complex physics, due to the relative ease of code development.

3.5.2 SPH Fundamentals

Based on interpolation theory with a weighting or kernel function, any mathematical expression (e.g. fluid mechanics governing equations, thermodynamic equations, structural mechanics equations, chemical equations) can be expressed in terms of its values at a set of arbitrarily spaced computational particles. The kernel function is indeed a weighting function, which determines the contribution to a field variable, $A(r)$, at
position, \( \mathbf{r} \). The kernel estimate of \( A(\mathbf{r}) \) is defined as (Monaghan, 1992):

\[
\langle A_h(\mathbf{r}) \rangle = \int_V A(\mathbf{r}') W(\mathbf{r} - \mathbf{r}', h) d\mathbf{r}',
\]

(3.1)

where \( V \) represents the solution space, the smoothing length, \( h \), represents the effective width of the kernel, \( W \) is a weighting function, which in SPH is generally referred to as a smoothing kernel and \( \langle A_h \rangle \) denotes the approximation of variable \( A \) over a smoothing length, \( h \). In the SPH formalism equation 3.1 is discretised to give:

\[
\langle A_h(\mathbf{r}) \rangle = \sum_{j=1}^{N} \frac{m_j}{\rho_j} A_j W(\mathbf{r} - \mathbf{r}_j, h),
\]

(3.2)

where, \( m_j \) and \( \rho_j \) are the mass and density of particle \( j \) at position \( \mathbf{r}_j \). The summation is performed over particles which lie within a circle of a radius determined by the kernel \( W \) centred at \( \mathbf{r} \).

The kernels used in the SPH method approximate a \( \delta \) function (see Figure 3.1). Monaghan (1992) suggests that a suitable kernel must have a compact support in order to ensure zero interactions outside its computational range. Recent studies (Amicarelli et al., 2011; Dehnen and Aly, 2012; Quinlan et al., 2006) indicate that the stability of the SPH algorithm depends strongly upon the second derivative of the kernel, suggesting that suitable kernels must have a high degree of continuity. One of the most popular kernels is based on cubic spline functions (Monaghan, 1992) and is defined by:

\[
W(r, h) = \frac{\sigma}{h^m} \times \begin{cases} 
1 - \frac{3}{2}s^2 + \frac{3}{4}s^3 & 0 \leq s < 1 \\
\frac{1}{4}(2 - s)^3 & 1 \leq s < 2 \\
0 & 2 \leq s 
\end{cases}
\]

(3.3)

Here, \( s = |\mathbf{r}|/h \), \( m \) represents the number of dimensions and \( \sigma \) is a normalization constant which takes the values \( 2/3 \), \( 10/7\pi \), \( 1/\pi \) in one, two and three dimensions, respectively. This kernel has compact support so that its interactions are exactly zero for \( |\mathbf{r}|/ > 2h \).

Kernels are particularly important for the stability of the SPH method which
strongly depends on the second derivative of the kernel (Morris, 1997). The cubic kernel of equation (3.3) is considered to have relatively poor performance since it is a low-order spline approximation of the Gaussian function (Morris et al., 1997). Many higher-order approximations are available in the SPH literature, like the Wendland kernel (Lucy, 1977) or the quintic kernel (Morris et al., 1997) which is the one also used for this study (see Chapter 4).

### 3.5.3 Weakly Compressible SPH

The first approach of modelling incompressible flows with SPH was made with “Weakly Compressible” SPH (WCSPH), in which incompressibility is approximated by manipulating the speed of sound and consequently the Mach number of the flow. Thus, changes in density can be restrained in a certain threshold. Moreover, pressure is treated as a thermodynamic variable, which is then solved via an artificial equation of state (Batchelor, 1967)

\[
p = B \left[ \left( \frac{\rho}{\rho_0} \right)^\gamma - 1 \right],
\]

where \(\rho_0\) is the reference fluid density, \(\gamma\) is usually equal with \(\gamma = 7\) and \(B\) is a pressure constant which governs the relative density fluctuation \(|\Delta \rho|/\rho_0\).

WCSPH has been widely applied in applications with great engineering interest. In coastal engineering (Colagrossi and Landrini, 2003; Gomez-Gesteira and Dalrymple, 2004; Dalrymple and Rogers, 2006; Dominguez et al., 2013; Altomare et al., 2014; Mahmoudi et al., 2014; Canelas et al., 2014; Aureli et al., 2015), where both single-
phase and multi-phase models have been tested, mainly to study phenomena related to wave impacts and wave breaking. Turbulent flows have also been studied in great detail, even using direct numerical simulations (DNS) (Robinson and Monaghan, 2012; Mayrhofer et al., 2015). More industrially relevant applications were studied by Groenenboom et al. (2014) and Canelas et al. (2014a) where airplane ditching and waves impacting harbours respectively were studied with WCSPH. Similarly, Wang et al. (2014) have studied overtopping of floating bodies.

Another attraction of WCSPH is that it can be enhanced computationally with relative ease. Recent advances in parallel computing techniques using graphic processor units (GPUs) and multi-GPU programming made WCSPH simulations with larger number of particles possible, allowing the application of the method in demanding engineering problems (Mokos, 2014; Mokos et al., 2015; Crespo et al., 2009; Dominguez et al., 2013a; Szewc, 2014).

Despite the aforementioned successful WCSPH applications, there are certain deficiencies which make the method unattractive. Specifically, WCSPH comes with a significant computational penalty, since the maximum permissible timestep is given by the Courant-Friedrichs-Lewy (CFL) constraint, in which the speed of sound is inversely proportional to the timestep, thus only a very small timestep is permitted. Spurious sound waves also reflect on solid surfaces due to an artificially low speed of sound, so boundaries located at significant distances are needed for certain flows. The ultimate drawback though is the inaccurate, high-noise level in the pressure results which arise from the sensitivity to the density summation given the large exponent $\gamma$. This noisy pressure field has motivated relevant work in WCSPH (Bonet and Lok, 1999; Ferrand et al., 2013; Marrone et al., 2011; Oger et al., 2007; Quinlan et al., 2006; Violeau, 2012), which managed to improve the pressure profile at the expense of increased modelling and computational cost.

These deficiencies of WCSPH with incompressible flow simulations have given rise to the development of alternative SPH methodologies, commonly referred to as incompressible SPH (ISPH), which enforce incompressibility in a more direct manner. In the next section (see Section 3.5.4) ISPH is explained in detail and the improvements over WCSPH are highlighted.
3.5.4 Incompressible SPH (ISPH)

The main difference between ISPH and WCSPH is that the stiff equation traditionally used by the latter (3.4) to calculate pressure, is now replaced by the Poisson equation:

\[ \nabla \cdot \left( \frac{1}{\rho} \nabla p^{n+1} \right) = \frac{1}{\Delta t} \nabla \cdot \mathbf{u}^*_i, \]  

(3.5)

where, \( \mathbf{u}^*_i \) is the intermediate velocity of particle \( i \), obtained during the time integration process (a full mathematical description in provided in Section 4.2). ISPH shows many advantages compared with the WCSPH. Firstly, ISPH is more efficient than WCSPH, because instead of the speed of sound it is only the kinematic speed of the fluid that is taken into account in the CFL constraint for the evaluation of the timestep (Lind et al., 2012). Moreover, the flow domain does not need to be overscaled to minimize sound pressure reflections. On the other hand, pressure is no longer treated as an explicit thermodynamic variable because it is now solved through a Poisson equation, which incurs an increased computational cost per timestep and makes the parallelisation of the algorithm significantly more complicated. Nevertheless, with recent developments in ISPH computational efficiency (Guo et al., 2015; Leroy, 2015) and a larger timestep, the increased computational cost of ISPH compared with WCSPH can be minimised.

The basis for ISPH was set by Chorin (1968), some nine years earlier than the first presentation of SPH (Gingold and Monaghan, 1977; Lucy, 1977). Chorin (1968) proposed a finite-difference method for solving the time-dependent Navier-Stokes equations for an incompressible fluid. This method was based on the decoupling of momentum (2.13) and continuity equations (2.12). Based on this study many different ISPH methodologies were developed from the mid 1990s onwards. The most significant ones are:

1. The divergence-free velocity method (Cummins and Rudman, 1998, 1999), in which particle positions, are advected with velocity to intermediate positions. At these positions, an intermediate velocity field, \( \mathbf{u}^*_i \), is calculated by integrating the SPH momentum equation forward in time without the pressure gradient term, which is then obtained by the pressure Poisson equation. The intermediate velocity is then projected onto a divergence-free space using the gradient of the
pressure.

2. *The density-invariance method* (Shao and Lo, 2003; Lo and Shao, 2002), in which density invariance is enforced directly. This method uses prediction-correction fractional steps with the temporal velocity field integrated forward in time without enforcing incompressibility in the prediction step. The resulting deviation in density is then projected onto a divergence-free space to satisfy incompressibility.

3. *The compound method* (Hu and Adams, 2006, 2007), in which both the density invariance condition and divergence free velocity field are imposed at every timestep through iteration.

4. *The shifting method* (Xu, 2009; Xu *et al.*, 2009; Lind *et al.*, 2012; Skillen *et al.*, 2013), in which particles are shifted from areas where particles are more dense to areas that are more sparse. This is done in addition to applying the divergence-free methodology (Cummins and Rudman, 1998, 1999).

In his work, Xu (Xu, 2009; Xu *et al.*, 2009), compared and evaluated the first three methods and found that divergence-free ISPH method alone cannot maintain stability, especially at high Reynolds numbers, due to clumping and stretching of the particles, while error accumulation is also found to be significant. On the other hand, the density-invariant method shows stable results but is inaccurate, with noise disturbances in the pressure field. Finally, the method which compounds both aforementioned methodologies, has proved to be both accurate and stable, at the expense of increased computational cost since the computationally demanding pressure-Poisson equation needed to be solved twice per timestep. The shifting methodology, however, has the benefit of avoiding stretching and bunching of the particles, but without the prohibitive computational cost. As a result the method showed great advantages in comparison to the already known ISPH or WCSPH methods.

A variety of different test cases have been presented in the relevant literature (Xu, 2009; Xu *et al.*, 2009; Lind *et al.*, 2012; Skillen *et al.*, 2013; Rogers *et al.*, 2014), that shows that this method is accurate, stable and gives almost noise-free results in key flow variables, that is pressure. Lind *et al.* (2012) have examined ways to improve the application of the method in free-surface flows and showed that both the flow
field and the pressure results are accurate compared with analytical and experimental solutions. Skillen et al. (2013) studied more complex free-surface problems, which included body slamming into water and breaking waves around cylinders, showing that the pressure results remain virtually noise-free even for such demanding cases. More recently, Leroy et al. (2014) implemented the unified semi-analytical wall boundary conditions (Ferrand et al., 2013) in ISPH with shifting allowing the modelling of more complex geometries and extended the method to solve heat transfer problems and turbulent flows.

The benefits of the shifting algorithm are not restricted to only ISPH approaches. Recently, the shifting formulation has been adopted by some WCSPH works (Vacondio et al., 2013; Fourtakas, 2014; Mokos, 2014; Mokos et al., 2014) because of the benefits it brings in particle redistribution. It has been found that common WCSPH problems such as voids in the flow domain or extreme particle clumping can be dealt effectively with the shifting method.

3.5.5 SPH and Non-Newtonian flows

Many attempts have been made in the past to model non-Newtonian flows with both WCSPH and ISPH. Zhu et al. (2010) have used WCSPH to study the flow of Bingham fluids in coaxial cylinder and vane rheometers. Fan et al. (2010) studied piston driven moulding flows of shear-thinning Power-law liquids. Ren et al. (2012) also studied moulding flows with WCSPH in a variety of different circular geometries offering an improved pressure field, but their work still showed fluctuations and irregular contours. Minatti and Paris (2015) have studied granular flows using an inelastic non-Newtonian models with variable yield stresses. Their prediction of the free-surface is reasonable, although it is evident that particles were disordered, suffering from clumping in regions. Gao et al. (2014) also studied landslides with WCSPH incorporating non-Newtonian rheology, describing terrestrial landslides but at poor resolution.

Fang et al. (2006) has also simulated transient visco-elastic free-surface flows, using an Oldroyd-B fluid. Fang et al. (2006) specifically deal with the problem of visco-elastic free-surface flows by applying both an artificial viscosity and an artificial stress parameter. Chui and Heng (2010) have also used a WCSPH visco-elastic model to
study fluid-structure interaction problems in bioengineering applications. Considerable work has been also done by Ellero (Ellero et al., 2002; Ellero and Tanner, 2005) to simulate visco-elastic flows using Oldroyd-B and the upper convected Maxwell (UCM) models for different Weissenberg numbers. The results they presented for the problem of transient plane Poiseuille flow achieved good agreement with the analytical solution. Similar work has been done by Xu et al. (2013, 2014) who studied two and three dimensional visco-elastic flows, including Couette and Poiseuille flows as well as three dimensional falling drops and jets.

Considerable work has been done on saturated soils and sediment. Bui and co-workers (Bui and Fukagawa, 2013; Bui et al., 2008, 2007) has modelled saturated soil and the interaction with both high velocity water-jets and sediment/standing water long period interactions. Fourtakas (Fourtakas et al., 2013; Fourtakas, 2014) has also studied saturated soils with application to nuclear industry. They also used a shifting methodology to improve their results and the particle distribution.

ISPH techniques have been widely applied in non-Newtonian flows also. The density invariance method of Shao and Lo (2003) has been the first ISPH methodology applied in non-Newtonian flows, and it has proven to be popular with other researchers, who adopted it in later works (Ataie-Ashtiani and Shobeyri, 2008; Hosseini et al., 2007; Morimoto et al., 2013; Vakilha and Manzari, 2008; Mirmohammadi and Ketabdari, 2011; Liang and He, 2014). Hosseini et al. (2007) used the density invariance method of Shao and Lo (2003) to study mud dam-breaks. Rafiee et al. (2007) extended this approach to visco-elastic flows, using Oldroyd-B rheological model for falling visco-elastic drops and jets. Another ISPH application in non-Newtonian flows was by Vakilha and Manzari (2008) for small scale flow, looking at flow through porous media using a Bingham fluid. Mirmohammadi and Ketabdari (2011) used the density invariant method to model scouring effects beneath marine pipelines, without giving conclusive results for the erosion depth. Morimoto et al. (2013; 2014) employed a similar approach to study scouring of sediment, but achieved relatively poor agreement with the experiment. More recently Liang and He (2014) have looked into the Mohr-Coulomb model and its application to landslide events using the density invariance method of Shao and Lo (2003) and managing good agreement with experiments.
The divergence-free velocity ISPH technique (Cummins and Rudman, 1999, 1998) is much less applied in non-Newtonian applications than the density invariant ISPH method (Shao and Lo, 2003). Kulasegaram et al. (2011) and later Deeb et al. (2014) and Badry et al. (2014) adopted the divergence free velocity field method of Cummins and Rudman (1999, 1998), to model the flow of self-compacting concrete. These are applications with very slow velocities and creeping flows, where the shifting formulation is not necessary.

In general, the existing non-Newtonian WCSPH and ISPH methods presented above have validated velocity fields and interface/free-surface profiles, but have not shown adequate performance in the estimation of the pressure field, which is an important flow characteristic for many non-Newtonian flows. It is clear that the state-of-the-art is lacking a method which can successfully handle free-surface non-Newtonian flows and provide an accurate pressure field. This motivates the work of Chapters 4 and 5 of this thesis, in which a novel ISPH algorithm with validated pressure fields is presented and applied in complex engineering and environmental applications involving inelastic non-Newtonian flows.

### 3.5.6 SPH and Multi-phase flows

Another field where SPH shows great advantages and has been commonly applied is multi-phase flows. Similarly to free-surface flows, multi-phase flows involve highly transient phenomena, where the separate phases can exhibit highly nonlinear deformations. SPH allows easy modelling of such problems and can track with accuracy the changes of the interface between the two phases. It is noted here that in this review more emphasis is given to research studies which involve a non-Newtonian phase, in many cases applied in environmental problems.

Considerable work has been done with WCSPH for the modelling of sediment or soil suspension. As mentioned, Bui et al. (2007) have studied the interaction of water and soil using WCSPH, where a water jet is blasted into a mass of soil. Fourtakas et al. (2013) have also studied the suspension of soil with the use of WCSPH, for nuclear sludge applications. Schwaiger and Higman (Schwaiger, 2008, 2007) used a WCSPH model to study the terrestrial landslide and tsunami generation of the Lituya Bay.
CHAPTER 3

CFD and the SPH method

incident (Fritz et al., 2001; Miller, 1960). Although their method showed relatively good agreement with the experimental results (Fritz et al., 2001) it was accompanied by many simplifications, assuming the rock-slide phase as a Newtonian fluid and the water phase as an inviscid fluid. Zanganeh et al. (2012) have studied scouring around pipes by coupling the two phases with an extra force term in the momentum equation. Their results matched the experimental data (Mao and Lyngby, 1986) well. Capone et al. (2010) have simulated a submarine landslide, where a mass of sand is collapsing inside water. The formulation implemented a Bingham fluid to model the collapsing sand, managing to get closer agreement with the original experiment (Assier-Rzadkiewicz et al., 1997) over pre-existing computational methods. Another WCSPH Newtonian/ non-Newtonian application was performed by Razavitoosi et al. (2014) examining erosional effects between water and a sedimentary bed. Although in their work the Bingham rheological model is shown to perform better than other artificial viscosity models, the pressure field is again found to be noisy.

Another interesting WCSPH approach for dispersed multi-phase flows has been recently suggested by Laibe and Price (2014), where the dispersed phase and the continuous phase are computationally modelled as a single-phase. To allow this a drag term needs to be introduced in the governing equations. This approach has been found to simplify the modelling of dispersed two-phase flows, such as dusty air (Laibe and Price, 2014).

In ISPH, Cummins and Rudman (1999) in their introduction of the divergence-free ISPH method, modelled the Rayleigh-Taylor instability problem. The results presented were compared with a WCSPH method, showing significant advantages in the representation of the interface profile. Hu and Adams (2007, 2009) have studied multi-phase phenomena with their divergence-free and density-invariant compound ISPH methodology which was based on an original work done for WCSPH (Hu and Adams, 2006). Amongst the applications covered in their study were the problems of oscillating droplets, and the Rayleigh-Taylor instability. Their method handled the oscillating droplet problem and provided improved results over Cummins and Rudman (1999) for the case of the Rayleigh-Taylor instability. Nevertheless, the interface of the Rayleigh-Taylor case appeared to suffer from instabilities, with several particles
being dispersed in alternate phases. Shadloo and co-workers (Shadloo et al., 2013; Shadloo and Yildiz, 2011), who used a version of the ISPH with shifting of Xu et al. (2009) also simulated the Rayleigh-Taylor instability, showing reasonable results, but predicted an over-developed flow and apparent particle dispersion into either of the two phases although a surface tension model was used. Zainali et al. (2013) have presented an extension of the ISPH with shifting algorithm to multi-phase and visco-elastic flows. Their method incorporates the shifting algorithm of Shadloo et al. (2011) and Xu et al. (2009) and a multi-phase model similar to (Shadloo et al., 2013) to study Newtonian/visco-elastic interactions with application to bubbles. Their results showed close agreement with benchmark computational results. The Hu and Adams (2007, 2006) methodology was also used by Tong and Browne (2014) to study multi-phase flows and heat transfer, and a close agreement for the surface forces was obtained, despite a low convergence rate.

Ataie-Ashtiani and Shobeyri (2008) using the density-invariant method (Shao and Lo, 2003), studied submarine landslides implementing both rigid bodies and a Bingham fluid/water multi-phase model. Hosseini et al. (2007) used the density-invariant method (Shao and Lo, 2003) to study the collapse of a mud column (modelled as a Bingham fluid) inside a water tank, but with relatively poor resolution.

It is clearly shown that the state-of-the-art incompressible multi-phase SPH methods often face problems with artificial mixing of the phases (particle dispersion across the interface), while the results of the pressure fields are generally omitted. Evidently, a method which can accurately represent multi-phase flows involving inelastic non-Newtonian phenomena, while having the capacity of accurate prediction of the pressure results can improve significantly on the state-of-the-art. This motivates the work of Chapter 6 of this thesis, where the ISPH with shifting method (which provides good quality pressure fields as shown in Lind et al., 2012; Skillen et al., 2013) is applied to incompressible multi-phase flows, with attention given to improving the quality of the interface representation and optimising particle distribution.
3.6 Summary

In this chapter the basics of CFD methods have been presented. A brief introduction to the main grid-based methods has been given. As mentioned these techniques may be in many cases the computational tool of choice for many research studies (e.g. Jenny et al., 2006; Gada and Sharma, 2009) but they often suffer from a poorly resolved interface and problems with mass conservation may occur as well (Rider and Kothe, 1995). These deficiencies can hamper applicability of these techniques in free-surface or multi-phase applications.

A description of particle-based CFD approaches was then given. The emphasis was on the SPH method and its different formulations. The WCSPH approach and ISPH variants have been introduced and it is shown that the ISPH method with shifting (Xu et al., 2009; Lind et al., 2012) is one of the most promising of the proposed SPH techniques managing to predict accurately both the flow fields and the pressure of demanding flows with Newtonian rheology (Xu, 2009; Xu et al., 2009; Lind et al., 2012; Skillen et al., 2013; Leroy et al., 2014; Leroy, 2015).

A review of non-Newtonian flows and multi-phase flows shows that although most approaches, either WCSPH or ISPH, can successfully predict the development of the flow profile, pressure fields can either be poorly predicted or omitted altogether from presented results.

To address the challenge of accurate pressure field predictions for such flows, it is evident that a novel incompressible non-Newtonian and multi-phase SPH technique is required, which may be achieved by extending the ISPH with shifting approach.

In the next chapters the new incompressible non-Newtonian SPH (INNSPH) method is introduced, which extends the work of Lind et al. (2012) to allow simulation of problems with non-Newtonian rheology. Validation cases are also presented both for Newtonian and non-Newtonian flows, showing the advantages that the new method has to offer.
Chapter 4

The Incompressible non-Newtonian SPH (INNSPH) method

4.1 Introduction

In the first chapters of this thesis (see Chapters 2 and 3) an overview of the CFD theoretical background is given. Moreover, it is highlighted that amongst the already existing SPH computational techniques, there is no single method offering an accurate and robust representation of the pressure field for free-surface non-Newtonian flows. On the other hand, it was shown that for Newtonian flows the ISPH with shifting methodology offers accurate predictions of both the pressure field and the velocity profiles of the flow (Xu et al., 2009; Lind et al., 2012; Skillen et al., 2013). The aim of this thesis is to extend the ISPH method with Fick’s shifting to model non-Newtonian and multi-phase flows.

In the current chapter the first steps towards the incompressible non-Newtonian SPH (INNSPH) method are introduced. As indicated previously (see Chapter 3) the formulation is based on the projection method proposed by Cummins and Rudman (1998, 1999), with the improvements by Xu et al. (2009) and later Lind et al. (2012), who introduced a shifting technique based on Fick’s law of diffusion. In the context of the current research study, the aforementioned formulation is expanded by adding Vola’s formulation for the calculation of the effective viscosity (Vola et al., 2004, 2003), which is readily applicable to a selection of non-Newtonian fluids, and the formulation
of shear stress divergence as proposed by Shao and Lo (2003). In the following paragraphs the numerical formulation of the INNSPH method is thoroughly explained.

### 4.2 The divergence free method

Cummins and Rudman (1998, 1999) were the first to apply Chorin’s approach (Chorin, 1968) of a divergence free velocity field in SPH. In this technique the particle positions, \( r^n_i \), are advected with velocity \( u^n_i \) to positions \( \mathbf{r}^\ast_i \)

\[
\mathbf{r}^\ast_i = \mathbf{r}^n_i + \Delta t (\mathbf{u}^n_i) .
\]  

(4.1)

At these positions, an intermediate velocity field, \( \mathbf{u}^\ast_i \), is calculated by integrating the SPH momentum equation (2.13) forward in time without the pressure gradient term:

\[
\mathbf{u}^\ast_i = \mathbf{u}^n_i + \Delta t \left( \frac{1}{\rho} \nabla \cdot \mathbf{\tau} + \mathbf{F} \right) .
\]  

(4.2)

where, \( \mathbf{\tau} \) is the shear stress tensor and \( \mathbf{F} \) is the body force. The following pressure Poisson equation is then solved to obtain the pressure needed to enforce incompressibility:

\[
\nabla \cdot \left( \frac{1}{\rho} \nabla P \right) = \frac{\nabla \cdot \mathbf{u}^\ast_i}{\Delta t} .
\]  

(4.3)

The pressure gradient term is next added to obtain a divergent-free velocity field:

\[
\mathbf{u}_{i}^{n+1} = \mathbf{u}^\ast_i + \Delta t \sum_j \frac{m_j}{\rho_i \rho_j} (p_i - p_j) \nabla W_{ij} .
\]  

(4.4)

Finally, the particle positions are centred in time,

\[
\mathbf{r}_{i}^{n+1} = \mathbf{r}^n_i + \Delta t \left( \frac{\mathbf{u}_{i}^{n+1} + \mathbf{u}^n_i}{2} \right) .
\]  

(4.5)

This type of projection is termed a full pressure projection.
4.3 The shifting methodology

As mentioned previously, although Cummins and Rudman’s projection method (Cummins and Rudman, 1998, 1999) was promising, it was found to be very unstable, especially in flows with moderate to high Reynolds numbers (Hu and Adams, 2007; Xu et al., 2009; Shao and Lo, 2003). This problem has been addressed by the shifting approach by Xu et al. (2009), which, in its most recent version as presented by Lind et al. (2012) and Skillen et al. (2013), uses Fick’s law of diffusion to control particle distribution. The original formulation is

\[ J = -D' \nabla C, \quad (4.6) \]

where \( J \) is the particle flux, \( C \) is the particle concentration and \( D' \) is the diffusion coefficient. Assuming that the flux is proportional to the velocity of the particles, a particle shifting velocity \( v_s \), and subsequently a particle shifting distance, \( \delta r_s = v_s \Delta t \), can be found. Consequently,

\[ \delta r_s \propto -D' \nabla C \Delta t. \quad (4.7) \]

Special attention needs to be given to the value of diffusion coefficient, \( D' \), which should be large enough to provide effective particle shifting and redistribution, but small enough to avoid error by excessive diffusion. An upper limit on the diffusion coefficient can be found through a Von Neumann stability analysis of the advection-diffusion equation, to give:

\[ D' \leq \frac{1}{2} \frac{h^2}{\Delta t_i'}, \quad (4.8) \]

where \( \Delta t_i' \) is the maximum timestep, locally determined from the CFL condition for a specific velocity and particle spacing. If we choose, as Skillen et al. (2013) suggested, \( \Delta t_i' = 0.5 h/\|u\|_i \), where \( \|u\|_i \) the velocity magnitude of particle \( i \), we get:

\[ D' \leq h \|u\|_i, \quad (4.9) \]
which substituted into (4.7), gives:

$$\delta r_s = -Ah\|u\|, \nabla C_i \Delta t,$$  

(4.10)

where $A$ is a problem-independent dimensionless constant, with a value within an order of magnitude of unity.

### 4.4 Gradient and divergence operators

The normalization explained by Bonet and Lok (1999) and Oger et al. (2007) is applied to improve the accuracy of the gradient and divergence operators. The expression for the kernel normalization can be read as

$$\nabla \tilde{W}_{ij} = \mathbf{L}(r) \nabla W_{ij},$$  

(4.11)

where $\nabla \tilde{W}_{ij}$ is substituted into

$$\nabla \phi_i \simeq - \sum_j V_j (\phi_i - \phi_j) \nabla W_{ij},$$  

(4.12)

for $\nabla W_{ij}$ as the normalized kernel first derivative. Matrix $\mathbf{L}$ is defined as:

$$\mathbf{L}(r) = \left( \begin{array}{c} \sum_j V_j (x_j - x) \frac{\partial W_{ij}}{\partial x} \sum_j V_j (x_j - x) \frac{\partial W_{ij}}{\partial y} - 1 \end{array} \right).$$  

(4.13)

This correction is applied everywhere in this work, apart from the viscous term (see equation 4.16). In this work a quintic spline kernel (Morris et al., 1997)

$$W(r_i - r_j, h) = \frac{1}{h^2 478\pi} \left\{ \begin{array}{ll} (3-s)^5 - 6(2-s)^5 + 15(1-s)^5, & \text{if } 0 \leq s < 1; \\
(3-s)^5 - 6(2-s)^5, & \text{if } 1 \leq s < 2; \\
(3-s)^5, & \text{if } 2 \leq s < 3; \\
0, & \text{if } s \geq 3 \end{array} \right.$$

(4.14)
is used, where $s = |\mathbf{r}_i - \mathbf{r}_j|/h$. The smoothing length $h$ is chosen to be $h = 1.3dx$, where $dx$ is the initial particle spacing. This $h/dx$ ratio was chosen as a good compromise between computational efficiency and accuracy, since reasonable accuracy convergence rates are achieved (see Chapters 5 and 6).

4.5 Viscous terms

The methodologies of Xu et al. (2009) and Lind et al. (2012) were developed to solve Newtonian flows. Thus, the particle viscosity $\mu$ was constant, so the following viscous term, first used by Morris et al. (1997), could be adopted in the momentum equation.

$$
(\mu\Delta \mathbf{u})_i = \sum_j m_j (\mu_i + \mu_j) \mathbf{r}_{ij} \cdot \nabla W_{ij} \rho_j (r_{ij}^2 + \eta^2) \mathbf{u}_{ij},
$$

(4.15)

where $m$ is the mass of the particle, $\rho$ is the fluid density, $\mu$ is the dynamic viscosity, $\mathbf{u}_i$ is the velocity of the particle $i$, $\mathbf{u}_{ij} = \mathbf{u}_i - \mathbf{u}_j$, $\mathbf{r}_i$ is the position of particle $i$, $\eta$ is a small value to avoid singular denominator. In the current study, this form is not adequate, because in non-Newtonian rheology the value of the effective viscosity depends on rate of deformation. Although, the viscous formulation of equation (4.15) allows for a non-constant viscosity, it is found to be inadequate for non-Newtonian flows since the non-diagonal terms of the shear-rate tensor are not included. Thus, the divergence of the stress tensor as presented by Shao and Lo (2003) is used.

$$
\left(\frac{1}{\rho} \nabla \cdot \mathbf{\tau}\right)_i = \sum_j m_j \left(\frac{\tau_{ij}}{\rho_i^2} + \frac{\tau_{ji}}{\rho_j^2}\right) \nabla_i W(\mathbf{r}_{ij}, h),
$$

(4.16)

where the stress tensor $\mathbf{\tau}$ is given by the following constitutive law (Vola et al., 2004):

$$
\mathbf{\tau} = \mu_{\text{eff}}(|\mathbf{D}|) \mathbf{D},
$$

(4.17)

where $|\mathbf{D}|$ is the the second principal invariant of the shear strain rate $\mathbf{D} = \nabla \mathbf{u} + \nabla \mathbf{u}^T$, (Vola et al., 2004):

$$
|\mathbf{D}| = \sqrt{\frac{1}{2} \sum_{i,j} D_{ij} D_{ij}}.
$$

(4.18)
In two dimensions, one gets:

\[ D = \nabla v + \nabla v^T = \begin{bmatrix} 2\frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & 2\frac{\partial v}{\partial y} \end{bmatrix} \]. \tag{4.19}

The contents, \( D_{ij} \), of tensor \( D \) are obtained using finite difference approximations, before decomposing into \( x \) and \( y \) directions. Thus,

\[
\left( \frac{\partial u}{\partial x} \right)_i = \left( \frac{\partial u}{\partial r_{ab}} \right) \left( \frac{\partial r_{ab}}{\partial x} \right)^{-1} \frac{(u_i - u_j)(x_i - x_j)}{r_{ij}^2},
\]

\[
\left( \frac{\partial u}{\partial y} \right)_i = \left( \frac{\partial u}{\partial r_{ab}} \right) \left( \frac{\partial r_{ab}}{\partial y} \right)^{-1} \frac{(u_i - u_j)(y_i - y_j)}{r_{ij}^2}.
\]

If we combine equations (4.16)-(4.21), the following is obtained:

\[
\left( \frac{1}{\rho} \nabla \cdot \tau \right)_i = \sum_j m_j(\mu_i + \mu_j) r_{ij}^2(\nabla \cdot W_{ij}) (u_{ij} \cdot \nabla_i W_{ij} u_{ij} + u_{ij} \cdot \nabla_i W_{ij} r_{ij}) .
\]

Similar equations to (4.22) were presented by Fan \textit{et al.} (2010), and Morris \textit{et al.} (1997).

With the aforementioned formulation, non-Newtonian rheology can be facilitated in ISPH with Fickian shifting method. The non-Newtonian models that are used in this project are presented below (Section 4.5.1) in forms that can be accommodated in computational analysis.

### 4.5.1 Non-Newtonian models

As shown in Section 2.3 there are many different approaches in modelling non-Newtonian flows, depending on their rheological behaviour. The main categories of non-Newtonian behaviours are the inelastic and the elastic fluid behaviours. In this project emphasis is given to inelastic fluid behaviour, with which many real-life non-Newtonian fluids are described.

In this section some of the inelastic non-Newtonian models of Section 2.3.1 are presented, with focus on their numerical implementation. The forms in which they are presented here are as proposed by Vola \textit{et al.} (2004), who studied the computational
application of inelastic non-Newtonian models.

*Power-law model:* The effective viscosity in a power law model is:

\[
\mu_{\text{eff}}(|D|) = \mu |D|^{N-1}, \tag{4.23}
\]

where \( \mu \) and \( N \) are the fluid consistency coefficient and the flow behaviour index respectively. For shear-thinning fluids \( (N < 1) \) equation (4.23) should be written as

\[
\mu_{\text{eff}}(|D|) = \mu(|D| + \eta)^{N-1}, \tag{4.24}
\]

where \( \eta \) is a small number to avoid singularity.

*Bingham model:* The Bingham model as shown in equation (2.20) represents a multi-valued function, where the fluid behaves as a solid when the shear stress is below the yield stress \( \tau_Y \), and as a Newtonian fluid when the shear stress exceeds that limit. So the constitutive equation of a Bingham fluid effectively becomes:

\[
|\tau| \leq \tau_Y \rightarrow D = 0, \tag{4.25}
\]

\[
|\tau| > \tau_Y \rightarrow \mu_{\text{eff}}(|D|) = \frac{\tau_Y}{|D|} + \mu . \tag{4.26}
\]

It is understandable, that the discontinuity that occurs in the constitutive equation of such a fluid may cause problems in the solution procedure. To overcome this issue, many different approaches have been used, where the solid zone of the Bingham fluid is approximated by a highly viscous fluid. Some examples are presented below.

1. *The bilinear model* (as presented by Hosseini et al. (2007))

\[
|D| \leq \frac{\tau_Y}{\alpha \mu} \rightarrow \mu_{\text{eff}}(|D|) = \alpha \mu, \tag{4.27}
\]

\[
|D| > \frac{\tau_Y}{\alpha \mu} \rightarrow \mu_{\text{eff}}(|D|) = \frac{\tau_Y}{|D|} + \mu . \tag{4.28}
\]

where \( \alpha \) is a large number (order of \( 10^2 \)).
2. The exponential model (as presented by Zhu et al. (2010))

\[
\mu_{\text{eff}}(|D|) = \tau_Y \left[ 1 - e^{m|D|} \right] + \mu, \quad (4.29)
\]

where \( m \) is a parameter related to the transition between the solid and the fluid zones. The higher the value of \( m \), the sharper the shape of the transition.

3. The Cross model (as presented by Shao and Lo (2003))

\[
\mu_{\text{eff}}(|D|) = \mu_0 + K\mu_{\infty}|D| \left[ 1 + K|D| \right], \quad (4.30)
\]

where \( K = \mu_0/\tau_Y \), \( \mu_{\infty} = \mu \) and \( \mu_0 = m\mu_{\infty} \), with \( m \) playing the same role as in the exponential model (see equation (4.29)).

In this study both the bilinear model (equations 4.27, 4.28) and the original formulation for Bingham fluids (equations 4.25, 4.26) were used. In the latter case, as equation 4.25 implies, the rate of deformation in the solid zone is zero when the stress rate is below the yield stress (equation 4.25). Thus, using equation (4.17) \( \tau = 0 \) is imposed, when \(|\tau| \leq \tau_Y\).

Herschel-Bulkley model: The Herschel-Bulkley model, similar to the Bingham model, represents a multi-valued function, where the fluid ideally behaves as a solid when the stress is not exceeding the yield stress \( \tau_Y \). Once yielded, a Herschel-Bulkley fluid resembles a power-law rheological behaviour. The Herschel-Bulkley model is expressed by:

\[
|\tau| \leq \tau_Y \rightarrow D = 0, \quad (4.31)
\]

\[
|\tau| > \tau_Y \rightarrow \mu_{\text{eff}}(|D|) = \frac{\tau_Y}{|D|} + \mu|D|^{N-1}, \quad (4.32)
\]

where \( N \) is a power-law exponent. Again in this model the solid zone could be replaced by a highly viscous fluid, similar to the formulations described for the Bingham fluid.

4.6 Solution algorithm

The algorithm of the INNSPH method can be described as follows:
(i) Convect particle $i$ to an intermediate position $\mathbf{r}^*_i$ (equation 4.1)

(ii) Calculate rate of deformation, $\mathbf{D}$ (equations 4.18 and 4.19), and effective viscosities, $\mu_{\text{eff}}$ (Section 4.5.1)

(iii) Calculate intermediate velocity field $\mathbf{u}^*_i$, without the $\nabla p$, using the stress term of equation 4.16

(iv) Calculate pressure from Poisson Equation (equation 4.3)

(v) Correct $\mathbf{u}^*_i$ by pressure gradient to obtain $\mathbf{u}^{n+1}_i$ (equation 4.4)

(vi) Obtain particle position centered in time (equation 4.5)

(vii) Shift particles into new positions (equation 4.10)

(viii) Correct velocity field

(ix) Begin next time step and return to (i)

4.7 Boundary conditions

In this work, several techniques are utilized to treat the wall boundaries for the different cases simulated. In most cases, the multiple boundary tangent method (Yildiz et al., 2009) is used, as it allows the modelling of a range of wall geometries in a straightforward manner. In other cases, which usually involve high-curvature, convex regions (e.g. the case of the Cross model complex filling of Section 5.6), the dummy particle method (Koshizuka et al., 1998) was chosen over the multiple boundary tangent method (Yildiz et al., 2009), because the latter was found to cause some local pressure disturbances. It is worth mentioning that boundary conditions are not the focus of this work, so the most appropriate were chosen for each case in order to minimise error.

Free-surface particles are recognised by the $\nabla \cdot \mathbf{r} \leq 1.5$ criterion (Lind et al., 2012, for two dimensional flows). Moreover, a Dirichlet boundary condition is enforced on the free-surface particles, setting their pressure to zero.
4.8 Timestep

Two timestep criteria were used in this study to ensure stability and accuracy of the method. First the Courant (CFL) condition (Hosseini et al., 2007; Morris et al., 1997; Shao and Lo, 2003) which states that:

$$\Delta t \leq 0.1 \frac{h}{\|u_{\text{max}}\|},$$  \hspace{1cm} (4.33)

where $\|u_{\text{max}}\|$ is the estimated maximum velocity during the simulation. Moreover, and perhaps more importantly for the non-Newtonian simulations, a timestep constraint which takes into account the viscous diffusion is employed. As suggested by Morris et al. (1997) and later by Shao and Lo (2003), the constraint takes the form:

$$\Delta t \leq \beta \frac{h^2}{\nu_{\text{max}}},$$  \hspace{1cm} (4.34)

where $\nu_{\text{max}}$ is the maximum estimated value of the kinematic viscosity for the specific problem, and $\beta$ a constant of the order of 0.1, which is chosen through computational experimentation (Hosseini et al., 2007; Morris et al., 1997; Shao and Lo, 2003). For the bilinear model (equations (4.27) and (4.28)) which is used in both Bingham and Herschel-Bulkley models, the value of $\nu_{\text{max}}$ is determined by the sub-yield stress value of the effective viscosity expression (equation (4.27)). For the power-law model numerical experiments are needed to determine a suitably small timestep because $\nu_{\text{max}}$ can take prohibitively large values.

4.9 Validation of the INNSPH formulation for Newtonian flows

In the previous sections of this chapter (see Sections 4.2-4.6) the mathematical model of INNSPH has been presented, which will allow the implementation of this method to free-surface single-phase non-Newtonian flows. However, before the method is applied to non-Newtonian problems, some benchmark Newtonian cases are considered to validate the implementation of the viscous term formulation, described in Section 4.5.
In the following sections some internal and free-surface Newtonian flows are presented, including plane Couette and Poiseuille flows, and free-surface wet and dry dam-break cases. It should be noted that these test cases are part of the preliminary validation process of the method, aiming to verify that the newly introduced formulation has not affected the accuracy of the original method (Lind et al., 2012). It is not a validation of the ISPH with shifting method, which has been thoroughly validated in the relevant literature (Xu et al., 2009; Lind et al., 2012; Skillen et al., 2013). Further rigorous tests and validations are performed in the next chapter (see Chapter 5) where the INNSPH algorithm is applied to non-Newtonian flows.

4.9.1 Internal flows

Initially the INNSPH formulation is applied in internal flows. Specifically, the Couette and the Poiseuille flows are considered. The configuration for both these test cases consists of a rectangular channel of height \( L = 1 \) m. Solid walls confine the computational domain at \( y = 0 \) m and \( y = L \), while periodic boundary conditions are imposed in the \( x \) direction. For both Couette and Poiseuille flows the computational domain was descretized with 50 particles across the height of the channel, using initial particle distance \( dx = 0.02 \) m.

4.9.1.1 Couette flow.

To validate the new formulation of the INNSPH method the analytical solution given in Morris et al. (1997) was used. Figure 4.1 shows the configuration for the Couette test case. As shown, the flow is driven by the movement of the wall at \( y = L \), which has a velocity magnitude \( V_0 = 1 \) m/s. The series solution for a Couette flow is given by:

\[
 u_x(y, t) = \frac{V_0}{L} y + \sum_{n=1}^{\infty} \left( \frac{2V_0}{n\pi} (-1)^n \sin \left( \frac{n\pi}{L} y \right) \exp \left( -\nu \frac{n^2\pi^2}{L^2} t \right) \right),
\]

where \( V_0 \) the velocity of the plate at \( y = L \). The flow was simulated for \( V_0 = 1 \) m/s, \( \nu = 1 \) m\(^2\)s\(^{-1}\) and \( L = 1 \) m giving \( Re = 1 \) at the steady state. Using equation (4.35) analytical results can be acquired from the initialization of the flow at \( t = 0 \) s until steady-state is reached.
Figure 4.1: Computational configuration for Couette flow: Arrows represent the expected velocity profile at steady state.

Figure 4.2: Comparative results of transient Couette flow with a steady state Reynolds number $Re = 1$: • Current study results, — series solution results

Figure 4.2 shows the comparison between velocity profiles obtained by equation (4.35) and the INNSPH method. As shown, the results calculated by INNSPH show close agreement with the theoretical results for all times of the flow simulation. Only some minor oscillations of the velocity field are observed for $t = 0.022$ s near the stationary wall at $y = 0$ m.
Figure 4.3: Computational configuration for Poiseuille flow: Arrows represent the expected velocity profile.

Figure 4.4: Comparative results of transient Poiseuille flow with a steady state Reynolds number $Re = 1$: • Current study results, – series solution results

4.9.1.2 Poiseuille flow.

Figure 4.3 shows the configuration for Poiseuille test case. In this case a body force $F = F_x$ is acting on $x$ direction, which accelerates the flow. Similar to the Couette flow, Morris et al. (1997) used an analytical solution which describes the acceleration of the flow from rest to steady-state. The series solution for this flow is given by:

$$u_x(y,t) = \frac{F}{2\nu}y(y - L) + \sum_{n=0}^{\infty} \frac{4FL^2}{\nu\pi(2n + 1)^3} \sin \left( \frac{\pi y}{L} (2n + 1) \right) \exp \left( -\frac{(2n + 1)^2\pi^2\nu}{L^2}t \right).$$

(4.36)
The problem was simulated for the following set of characteristic values: \( L = 1 \text{ m} \), \( F = 8 \text{ m s}^{-2} \) and \( \nu = 1 \text{ m}^2\text{s}^{-1} \), for a particle spacing of 0.02 m. These values correspond to a peak fluid velocity of \( V_0 = 1 \text{ m/s} \) and a Reynolds number of \( Re = 1 \), achieved when the flow has reached steady state conditions.

Figure 4.4 shows the comparisons between the series solution of equation 4.36 and the results obtained by INNSPH. It is clearly illustrated that, like the Couette flow, INNSPH manages to predict the development of the Poiseuille flow for all times of the simulation.

With these test cases it is shown that INNSPH can predict accurately the transient phenomena of an internal Newtonian flow. In the next section (4.9.2) some free-surface Newtonian flows are considered and comparisons with previously published results (Lind et al., 2012) are performed.

### 4.9.2 Free-surface flows

Dam-break simulations are an excellent way to examine the behaviour of the algorithm in high acceleration free-surface flows. Moreover, it is a useful way to assess the abilities of the method in pressure calculation, due to the high pressure contours near impact sites that take place in such flows. In this section both cases of wet and dry bed dam-break problems are presented. Qualitative comparisons between the INNSPH methodology and the original ISPH code with shifting of Lind et al. (2012) are given.
4.9.2.1 Dry bed dam-break.

Dry bed dam-breaks are a common free-surface validation test-case for SPH since they involve highly transient phenomena. The computational configuration of the dry dam-
break problem is illustrated in Figure 4.5. As shown, a water column of height 0.2 m and width 0.1 m, is let free to flow at time \( t = 0 \) s in a reservoir of length 0.4 m.

Figure 4.6 illustrates that the proposed method retains a smooth free-surface profile, which is closely comparable to the one produced by the original ISPH method with Fickian shifting as presented in (Lind et al., 2012). Furthermore, the contours of the pressure distribution are almost identical between the control and the evaluated results. Some problems observed with particles detaching from the free-surface (figure 4.6c) resemble results shown in Lind et al. (2012) for different values of the diffusion coefficient \( D' \). Therefore, numerical optimization with different values of \( D' \) would improve the small discrepancies observed in this study, but this is not necessary at this stage.

### 4.9.2.2 Wet bed dam-break.

Similar to the dry bed dam-break, wet bed dam-breaks are a common test case for free-surface computational methods, since they involve highly deformable free-surfaces. Figure 4.7 shows the set-up of a wet dam-break as proposed by Lind et al. (2012). A water column of height and width of 1 m and 2 m respectively, is released at \( t = 0 \) s in a wet reservoir with total length of 4 m and wet depth of 0.1 m.

Comparisons of the free-surface profiles are made in Figures 4.8, where the INNSPH and the ISPH with shifting results of Lind et al. (2012) are overlaid. The comparison shows that the free-surfaces of both methods is in very good agreement. Small differences are only observed at the latter stages of the simulation (detail in Figure 4.9),

![Figure 4.7: Wet dam-break configuration as presented in (Lind et al., 2012), with \( h_1 = 1 \) m, \( x_1 = x_2 = 2 \) m and \( h_2 = 0.1 \) m.](image-url)
CHAPTER 4

The INNSPH method

Figure 4.8: Comparative results of wet bed dam-break case: Overlay INNSPH free-surface particles (purple colour) with results from Lind et al. (2012)

Figure 4.9: Detail of the free-surface particles at an evolved stage of the wet dam-break case: Purple colour: INNSPH free-surface particles.

where the dispersed particles predicted by INNSPH seem to be greater in number than those from Lind et al. (2012). However, with manipulation of the diffusion coefficient $D'$ these discrepancies can be reduced (Lind et al., 2012).
4.10 Summary

In this chapter the numerical model of incompressible non-Newtonian SPH (INNSPH) has been comprehensively described. Detailed presentation of the divergence-free ISPH method of Cummins and Rudman (1998, 1999) was given, along with the shifting method as shown by Lind et al. (2012) and Skillen et al. (2013) which are the basis of the INNSPH method.

The shear-stress formulation of Shao and Lo (2003) was introduced, with which non-Newtonian rheological models can now been implemented. Moreover, some inelastic rheological models (Vola et al., 2004, 2003) were presented, in a form that allows computational analysis. With these models most types of inelastic non-Newtonian behaviour can be simulated.

Finally, some preliminary validation test-cases for Newtonian flow were shown. The intention of this validation process was to show that the accuracy and efficiency of the well validated ISPH with shifting method (Xu, 2009; Xu et al., 2009; Lind et al., 2012; Skillen et al., 2013) was not compromised by the introduction of the new stress formulation. Indeed, the method was found to be accurate when compared with analytical solutions for internal flows, while the comparisons with the original ISPH shifting algorithm (Lind et al., 2012; Skillen et al., 2013) for free-surface flows showed very good agreement between the two methods for Newtonian flows.

In the next chapter the application of the INNSPH method to internal and free-surface inelastic non-Newtonian flows is presented. A rigorous validation procedure is followed, comparing the INNSPH with analytical, experimental and state-of-the-art computational results highlighting the improvements over the state-of-the-art by the INNSPH algorithm.
Chapter 5

Non-Newtonian Results of the INNSPH method & Discussion

5.1 Introduction

In the previous chapter the INNSPH method has been introduced, based on the ISPH methodology with Fickian shifting of Lind et al. (2012) and Skillen et al. (2013). Moreover, some preliminary validation test cases for Newtonian flows have been presented (see Section 4.9) showing that the proposed method retains the benefits of ISPH with shifting for Newtonian flows.

In the current chapter, the INNSPH method is applied in non-Newtonian flows with detailed validation. For this purpose, the following cases are considered: a variety of non-Newtonian Poiseuille and circular Couette flows using Bingham, power-law and Herschel-Bulkley rheological models, where analytical solutions are readily available. Dam-break cases are presented for Bingham visco-plastic model and compared with both experimental (Komatina and Jovanovic, 1997) and computational results (Shao and Lo, 2003; Hosseini et al., 2007). A similar dam-break case for shear-thinning power-law fluid is also compared with related computational results (Hosseini et al., 2007). Some more demanding moulding flow problems are considered and compared with computational results, both for a power-law shear-thinning fluid (Fan et al., 2010) and for a Cross-model pseudoplastic fluid (Ren et al., 2012). Traditionally in SPH, pressure results are rarely presented due to the very noisy pressure fields, making it
difficult to establish the accuracy of the pressure results given by INNSPH. In order to bridge this gap in the validation process of the proposed method, some of the aforementioned test cases were also compared with results from a control volume finite element method (CVFEM) (Baliga and Patankar, 1980) available through ANSYS-CFX CFD commercial software. Moreover, a standard WCSPH technique is implemented, from which pressure results are taken as reference for comparison with INNSPH for a still fluid test case. Finally, the importance of the shifting algorithm is highlighted, showing its necessity in realistic non-Newtonian simulations.

The INNSPH method as shown herein, manages to retain the benefits of the SPH methodology in accurately calculating free-surface profiles, while providing an improved estimation of the pressure field, compared with other SPH methods.

5.2 Poiseuille flows

Closed-channel plane Poiseuille flows are chosen as a test case because the shear phenomena of non-Newtonian fluid can be easily tested. Poiseuille flows are commonly used for the validation of computational methods, since the analytical steady state solution, can be easily acquired from:

\[
\frac{\partial u}{\partial t} = \frac{\partial \tau_{yx}}{\partial x} + F_x, \tag{5.1}
\]

where \(u\) and \(F_x\) are the velocity and body force in the \(x\) direction respectively, and \(\tau_{yx} = \mu_{\text{eff}} \frac{\partial u}{\partial y}\) since \(\partial v/\partial x = 0\) for this case. The steady state solution is derived from equation (5.1), by substituting the relevant rheological model of Section 4.5.1 to determine \(\tau_{yx}\), and setting \(\partial u/\partial t = 0\). Detailed derivation of equation (5.1) is given in Appendix A.

For the simulation of Poiseuille flows, a 2-D computational domain was constructed, which consisted of a rectangular box, with height \(L = 1\) m. Figure 5.1 illustrates the configuration of the computational experiment. The domain was discretized with 50 SPH particles across the \(y\) direction, while periodic boundary conditions were set in the \(x\) direction. For simplicity, the parameters \(\rho = 1\) kg/m\(^3\), \(\mu = 1\) Pa·s and \(F_x = 8\) ms\(^{-2}\),
CHAPTER 5

INNPSPH Results & Discussion

Figure 5.1: Configuration of Poiseuille flow computational domain: Arrows show the expected velocity profile.

<table>
<thead>
<tr>
<th>Case</th>
<th>Approximate computational times [h:mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bingham model ($\Delta t = 5.0 \times 10^{-6}$ s)</td>
<td>4:30</td>
</tr>
<tr>
<td>Power-Law ($\Delta t = 2.5 \times 10^{-4}$ s)</td>
<td>0:20</td>
</tr>
<tr>
<td>Herschel-Bulkley ($\Delta t = 5.0 \times 10^{-6}$ s)</td>
<td>4:40</td>
</tr>
</tbody>
</table>

Table 5.1: Approximate computational times for the different non-Newtonian models with particle spacing of $dx = 0.02$ m and for a physical time period of 1s.

were chosen which corresponds to a maximum velocity $u_{\text{max}} = 1$ m/s for a Newtonian flow ($Re \approx 0.7$).

The test cases were simulated on a desktop computer with a CPU specifications of 3.00GHz and 4 GiB RAM memory. Approximate computational times are presented in Table 5.1.

All three inelastic rheological models, presented in Section 4.9, have been tested and validated against the analytical solutions derived from equation (5.1). The comparisons were made with the normalized velocity $\tilde{u} = u/u_0$, where $u_0$ is the average velocity in the computational domain. The results are presented between the plane of symmetry at $y = 0$ and the upper wall at $y = L/2$, and are shown in Figure 5.1, for the various parameters of the different rheological models.
5.2.1 Bingham model

The Bingham rheological model represents a visco-plastic flow, where below the threshold value of yield stress $\tau_Y$ the rate of deformation is zero. Bingham flows are usually determined by the Bingham number:

$$Bn = \frac{\tau_Y L}{\mu V},$$

where $V$ is the characteristic velocity (in this case the maximum velocity), and it gives a measurement of the yield stress relative to viscous stress. In Figure 5.2 results of the normalized velocity profiles $\tilde{u}$, at a range of Bingham numbers between 0 to 2.1 are presented. As shown, the INNSPH results shows exceptionally good agreement with the analytical solution.

Timestep convergence. During the study of the Bingham fluids, the significance of the timestep size was also investigated. As shown in Figure 5.3, a significantly small timestep length is needed so that the solution can converge in the plug flow region of the flow field. This is because, during the acceleration phase of the flow the components of the deformation rate tensor evolve rapidly, with the effect of over-predicting the stress of the particles, making them artificially exceed the yield stress criterion.
CHAPTER 5
INNSPH Results & Discussion

Figure 5.3: Time convergence results, conducted for Bingham model ($Bn = 3$). The solid line indicates an order of convergence of approximately 1.2.

The maximum deviation of the INNSPH results from the analytical solution at steady state

$$\epsilon_{\text{max}} = \frac{||u_{\text{analytical}} - u_{\text{INNSPH}}||}{||u_{\text{analytical}}||},$$

(5.3)
is plotted in Figure 5.3 for different timestep values. In this case the rate of time convergence was found to be approximately 1.2.

Comparison of original Bingham model with the Bingham bilinear model. The comparison between the effectiveness of the bilinear model as proposed by Hosseini et al. (2007) as well as the original formulation for Bingham fluids (equations 4.25-4.28), is also presented. As it is shown in Figure 5.4, for the same time and space resolution ($dx = 0.02$ m, $\Delta t = 5 \times 10^{-6}$ s) the Hosseini bilinear model, where the solid-zone is modelled with a highly viscous fluid, shows only minor differences compared with the analytical solution. Nevertheless, due to the easier implementation in the algorithm and the simplicity of determining the timestep size the bilinear model is used for the visco-plastic simulations in this thesis.

5.2.2 Power-Law model

With the use of the power-law model both shear-thinning ($N < 1$) and shear-thickening ($N > 1$) rheological behaviours can be tested. In Figures 5.5a and 5.5b comparisons
of velocity profiles between the INNSPH and the analytical solutions, for different values of shear-thinning and shear-thickening materials are presented. In both cases the agreement with the analytical solutions is very satisfactory for most values of the exponent parameter $N$, while signs of inaccuracies start to appear only at more extreme shear-thinning behaviours (e.g. $N = 0.1$ in Figure 5.5a).

The spatial convergence of the method has also been investigated. As shown in Figure 5.6, the $\epsilon_{L2}$ relative errors in steady-state velocity

$$
\epsilon_{L2}(u) = \sqrt{\frac{\sum (|u_{\text{analytical}}|^2 - |u_{\text{calculated}}|^2)}{\sum |u_{\text{analytical}}|^2}},
$$

for both a shear-thinning and a shear-thickening power-law fluid, with $N = 0.5$ and $N = 3$ respectively show close to linear convergence, with a calculated convergence rate of approximately 0.9, for both cases. These results show an obvious discrepancy from the theoretical convergence rate suggested in the literature (Quinlan et al., 2006; Noutcheuwa and Owens, 2012), which should be second order in $h$. Nevertheless, when different $dx/h$ ratios were tested the convergence rate did not show substantial improvement. Thus, the smoothing length of $h = 1.3dx$ is maintained as a reasonable compromise between accuracy and computational efficiency. It should also be noted that for Newtonian flows the convergence rate calculated by Lind et al. (2012) is not higher than 1.2 for the case of Taylor-Green vortices, which shows that the addition
5.2.3 Herschel-Bulkley model

The Herschel-Bulkley model enables visco-plastic behaviours to be tested. The difference with the Bingham model is that in the yielded region the fluid behaves like a Power-Law fluid, rather than a Newtonian one, with constant viscosity. Test cases for different set of yield stress $\tau_Y$ and power-law exponent $N$ parameters were conducted and shown in Figure 5.5c. As illustrated the agreement of the INNSPH solutions with the analytical results is very good for the Herschel-Bulkley model as well.

In Table 5.2 the relative $L_2$ errors in steady-state velocity (see equation 5.4) for all different rheological models are presented for select rheological parameter values. The
(a) Spatial convergence $N = 0.5$

(b) Spatial convergence $N = 3$

Figure 5.6: Convergence results of INNSPH, conducted for a power-law model ($N = 0.5$ and $N = 3$). The solid line indicates an order of spatial convergence of approximately 0.9 in both cases.

<table>
<thead>
<tr>
<th>Case Parameters</th>
<th>$\epsilon_L^2$ error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bingham fluid $Bn = 2.1$</td>
<td>$6.41 \times 10^{-2}$</td>
</tr>
<tr>
<td>Power-Law shear-thinning $N = 0.5$</td>
<td>$4.46 \times 10^{-2}$</td>
</tr>
<tr>
<td>Power-Law shear-thickening $N = 3$</td>
<td>$4.66 \times 10^{-2}$</td>
</tr>
<tr>
<td>Herschel-Bulkley $N = 0.5, Bn = 2.1$</td>
<td>$6.56 \times 10^{-2}$</td>
</tr>
</tbody>
</table>

Table 5.2: $\epsilon_L^2$ errors in steady-state velocity of all different non-Newtonian models for particle spacing of $dx = 0.02$ m

relative $L_2$ error for all cases presented here was calculated for a particle spacing of $dx = 0.02$ m, and found to be of the order of $10^{-2}$.

Evidently, for all three rheological models INNSPH results and analytical solutions appear to be in very close agreement. As already mentioned, for the shear-thinning cases (Bingham model (Figure 5.2) and power-law with $N < 1$ (Figure 5.5a)) errors appear to increase relative to the severity of the rheological parameters (an increase of yield stress $\tau_Y$ or decrease of power-law exponent $N$), hence minor oscillations in the velocity profile appear in extreme cases (see Figure 5.5c). Table 5.3 shows the increase of the $L_2$ relative error in steady-state velocity of equation (5.4) calculated over all SPH particles, for a decreasing power-law exponent ($N < 1$). This phenomenon is expected, and is most likely due to the relative decrease in the particle resolution in the high shear region of the flow. In more shear-thinning fluids a greater solid-zone
appears, with an ever decreasing shear zone thickness (the area between the solid-zone and the solid walls, where the fluid is most deformed). Consequently, the ability to capture and resolve the shear-zone is going to be reduced with an increasing solid-zone, given that the number of particles in the different cases is constant and a relatively uniform particle distribution is retained. The aforementioned particle distribution is more clearly illustrated in Figures 5.2 and 5.5a, for increasingly shear-thinning flows.

As can be seen, all the plane Poiseuille flow cases suggest that this method is suitably accurate for a wide range of inelastic non-Newtonian flows. Moreover, suitable convergence has been demonstrated so greater accuracy can be achieved with a sufficient increase of the particle resolution.

### 5.3 Circular Couette flow

Having validated the method with the test case of the plane Poiseuille flow, the more complex internal problem of an annular viscometer is examined. In this case the computational domain (shown in Figure 5.7) consists of a fixed outer cylinder with radius $r_o = 1$ m and an internal cylinder with radius $r_i = 0.5$ m, rotating with a constant angular velocity $\omega = 1$ rad/s. The value of the initial particle spacing chosen was $dx = 0.025$ m corresponding to 20 particles discretizing the region between the inner and the outer cylinder walls.

Simulations were conducted for a Power-law shear-thinning fluid with $N = 0.5$ and a Bingham fluid with yield stress $\tau_Y = 10$ Pa, while for both fluids viscosity and density values were chosen as $\mu = 1$ Pa·s and $\rho = 1000$ kg/m³ respectively. The resulting flow has relatively low Reynolds number ($Re < 250$). The steady state analytical solution

<table>
<thead>
<tr>
<th>Power-law index $N$</th>
<th>$\epsilon_{L2}$ relative error</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>$4.46 \times 10^{-2}$</td>
</tr>
<tr>
<td>0.2</td>
<td>$5.91 \times 10^{-2}$</td>
</tr>
<tr>
<td>0.1</td>
<td>$8.28 \times 10^{-2}$</td>
</tr>
<tr>
<td>0.05</td>
<td>$8.78 \times 10^{-2}$</td>
</tr>
</tbody>
</table>

Table 5.3: $\epsilon_{L2}$ relative error in steady-state velocity for increasing shear-thinning behaviour for a power-law fluid.
for these problems is well documented in the literature (Bird et al., 1987; Capone et al., 2010; Hosseini et al., 2007). Figure 5.8 shows the comparisons between INNSPH and the analytical results. As shown the agreement between the analytical and the INNSPH results is very good despite the coarse domain discretization (20 particles spanning radially between the two cylinder walls).

Comparisons were also performed against the WCSPH results presented by Capone et al. (2010) for the Bingham fluid, who used a variable particle resolution discretization scheme to extract the best results near the region of rotating inner cylinder. Figure 5.9 shows the tangential velocity at each particle position for the two methods and the comparison with the analytical solution. As shown the two techniques produce very
similar results and achieve a good agreement with the analytical solution. Nevertheless, INNSPH is able to match the accuracy of the WCSPH results (Capone et al., 2010) using significantly fewer particles and without the need of a variable resolution near the rotating cylinder, illustrating the increased efficiency and accuracy of INNSPH over the WCSPH method.

### 5.4 Dam-break case

Having successfully applied the ISPH method with Fickian shifting (Xu, 2009; Xu et al., 2009; Lind et al., 2012; Skillen et al., 2013) to closed channel non-Newtonian flows, the INNSPH method was consequently extended to free-surface flows.

Dam-break simulations are a useful way to examine the behaviour of a computational algorithm for free-surface flows. Moreover, due to the highly anisotropic shear-stress field during the collapse of the fluid column, dam-breaks are quite interesting from a rheological point of view.

In this section comparisons with experimental data (Komatina and Jovanovic, 1997), previously published SPH results (Hosseini et al., 2007) and CVFEM computational results will be presented. Both Bingham and Power-law rheological models are considered for these simulations. Figure 5.10 shows the set-up which is used for all
the cases in this section and which was utilised by Komatina and Jovanovic in their experiment (Komatina and Jovanovic, 1997). The problem consists of a rectangular column of fluid, of length $L = 2$ m and height $H = 0.1$ m, which rests on a mildly sloped bed, inclined at $S_0 = 0.1\%$.

The computational domain has been discretized using 2000 fluid particles with initial spacing of $dx = 0.01$ m.

### 5.4.1 Bingham fluid dam-break

The Bingham fluid modelled in this case corresponds to a water-clay mixture with a volume concentration of $C_V = 27.4\%$ (Komatina and Jovanovic, 1997) and density of $\rho = 1200$ kg/m$^3$. Komatina and Jovanovic (1997) used $C_V$ to estimate the rheological characteristics of the Bingham fluid from:

$$\mu = 0.621 \cdot \exp(0.173 \cdot C_V) \quad (5.5)$$

and

$$\tau_Y = 0.002 \cdot \exp(0.342 \cdot C_V) \quad , \quad (5.6)$$

which, for the assigned value of $C_V$, give $\mu \approx 0.071$Pa · s and $\tau_Y \approx 25$Pa. The peak Reynolds number of the flow was calculated to be $Re \approx 250$ and the Froude number was small with $Fr < 0.5$. 

Comparison against experimental data (Komatina and Jovanovic, 1997): In Figure 5.11 the computed non-dimensional leading edge propagation $X = (x - H)/H$ versus the non-dimensional time $T = t \sqrt{g/H}$ is plotted. There is a reasonably good agreement between INNSPH and the experimental results. Moreover, with increasing time, it is found that the flow tends to the “freeze-point”, where the shear stress is below the yield stress $\tau_Y$ everywhere in the flow domain. This behaviour was also observed in (Komatina and Jovanovic, 1997) and (Shao and Lo, 2003).

Comparison with previous SPH results (Hosseini et al., 2007) Further comparisons are made in Figure 5.12, where it can be seen that the free-surface shapes of INNSPH are in close agreement with the profiles presented in the work of Hosseini et al. (2007), who used the density invariant ISPH method. The rheological model used in their work (Hosseini et al., 2007) was the bilinear Bingham model (equations (4.27) and (4.28)) to describe the rheological properties of the flow. As shown the predicted free-surface profiles are very close to the ones presented by Hosseini et al. (2007) even though the initial particle spacing used in Hosseini et al. (2007) was half of that used in INNSPH. Pressure contours of the INNSPH method are also presented in Figure 5.12. As illustrated, the pressure contours of the INNSPH method show a very smooth distribution, even though the simulation was conducted at a comparatively low resolution (2000 particles compared to 8000 particles in Hosseini et al. (2007)).
CHAPTER 5
INNSPH Results & Discussion

Figure 5.12: Pressure contours of INNSPH method for a Bingham dam-break case (Komatina and Jovanovic, 1997), and comparisons with the free-surface profiles of (Hosseini et al., 2007): a) INNSPH method, b) computational results of (Hosseini et al., 2007).

Figure 5.13: Pressure contours and free-surface profiles comparison between INNSPH and CVFEM: a) INNSPH method, b) CVFEM.
Figure 5.14: Dam-break propagation comparisons with computational results (Hosseini et al., 2007): ■ computational results (Hosseini et al., 2007), solid line – INNSPH results.

Only some very minor pressure noise appears in particles close to the wall and near the leading edge of the flow at times lower than $t = 0.6s$.

**Comparison with CVFEM:** To validate the accuracy of the INNSPH pressure results, the dam-break case of Komatina and Jovanovic (1997) was compared with results from CVFEM. The same time step was used for both numerical methods, while for increased accuracy the size of the finite element mesh spacing used was half the particle distance of INNSPH simulation. Figure 5.13 shows the comparisons between the two algorithms. The agreement of both the free-surface profiles and the pressure contours is close. As illustrated, INNSPH shows some advantages over CVFEM, since the latter shows very small fluctuations in the pressure field in the region $x \in (1.4 \text{ m}, 2 \text{ m})$, at $t = 2 \text{ s}$.

### 5.4.2 Power-law dam-break case

Using the same configuration as Figure 5.10, Hosseini et al. (2007) undertook a dam-break simulation for a power-law fluid, with $\rho = 1200 \text{ kg/m}^3$, $\mu = 1.74 \text{ Pa} \cdot \text{s}$ and $N = 0.15$. The comparison between the INNSPH method and the density invariant ISPH results presented by Hosseini et al. (2007) for the propagation of the leading edge over time show an excellent agreement (Figure 5.14).

With the Bingham and the power-law dam-break simulations yielding favourable
comparisons with experimental (Komatina and Jovanovic, 1997) and computational (Hosseini et al., 2007) results, the INNSPH technique developed here clearly performs well when extended to free-surface flows. More complex flows can now be examined to assess the performance of INNSPH against other computational techniques for industrially relevant problems.

### 5.5 Power-law moulding flow

Fan et al. (2010) have presented in their work a WCSPH method for the simulation of non-Newtonian moulding flows. This case is particularly interesting because of the very high pressures that are observed in the flow domain. In Fan et al. (2010) a highly viscous power-law fluid was used, with

\[
\mu_{\text{eff}}(|D|) = \begin{cases} 
\mu |D_0|^{N-1} & |D| \leq |D_0| \\
\mu |D|^{N-1} & |D| > |D_0| 
\end{cases},
\]

where \(|D_0|\) is the critical shear rate, below which the effective viscosity \(\mu_{\text{eff}}\) is constant.

The flow parameters used for this case were the same as those used by Fan et al. (2010) with \(|D_0| = 10^{-4} \text{ s}^{-1}\), \(\mu = 6200 \text{ Pa} \cdot \text{s}\), \(N = 0.294\) and density \(\rho = 753.012 \text{ kg/m}^3\).

In addition, the flow was driven by a piston with constant speed of \(u_{\text{piston}} = 0.5 \text{ m/s}\), through a triangular shaped geometry with a neck width of 2 mm into a rectangular mould of width 10 mm (Figure 5.15). Moreover, a gravity body force \((g = 9.81 \text{ m/s}^2)\), with the same direction as the negative \(y\)-axis, acts upon all fluid particles. With the high viscosity, the Reynolds number calculated in the neck region was relatively small with \(Re \approx 0.25\). For this simulation a particle spacing of \(dx = 0.4 \text{ mm}\) was used.

The comparison between the INNSPH and the results of Fan et al. (2010) is presented in Figure 5.17. As illustrated, the free-surface profiles calculated by INNSPH
Figure 5.16: Detail of the pressure field predicted by the INNSPH methodology for the case of the power-law fluid moulding flow, at time $t = 0.05$ s.

Figure 5.17: Representation of flow profiles and pressure contours of INNSPH method for a power-law moulding flow case (Fan et al., 2010) and comparisons with computational results (Fan et al., 2010): a) INNSPH method, b) computational results of Fan et al. (2010).

Figure 5.18: Free-surface and pressure contour comparison between INNSPH and CVFEM for the case of power-law moulding flow at 1) $t = 0.01$ s and 2) $t = 0.03$ s: a) INNSPH method, b) CVFEM.
have a very good agreement with those of the WCSPH method (Fan et al., 2010). Smooth pressure results for INNSPH are also illustrated both in Figure 5.16 and Figure 5.17, whereas pressure results are omitted from the work of Fan et al. (2010).

**Comparison with CVFEM:** It is clearly shown that INNSPH can predict a very smooth pressure field, despite a highly stressed flow and high pressures of order $10^6$ Pa. To validate the accuracy of the pressure predictions, comparisons are again made against CVFEM. Time step and mesh size parameters are the same as used in the INNSPH simulation. Times $t = 0.01$ s and $t = 0.03$ s were chosen for comparison, since CVFEM free-surface profiles were found to deviate significantly from both INNSPH and WCSPH results (Fan et al., 2010) at times subsequent to the entrance of the flow into the rectangular cavity. As illustrated in Figure 5.18, the agreement between the two methods is very satisfactory, with INNSPH showing similar pressure contours to CVFEM. Such results highlight the wide applicability of INNSPH, since it is evident that the method not only simulates free-surface profiles efficiently but produces accurate pressure results that can be used to inform many industrial applications (e.g. the design of a mould for advanced manufacturing).

### 5.6 Cross model complex fillings

Ren et al. (2012) proposed in their work a new WCSPH method which produces better quality pressure results than the traditional SPH method, and with increased accuracy. Their method was based on the introduction of a density diffusive term (Ren et al., 2012), so that the pressure oscillations could be reduced. Moreover, a corrected kernel gradient was implemented for improved accuracy. Indeed, the comparisons that they presented against the traditional SPH method showed some very promising improvements regarding the pressure field. Generalized Newtonian fluid moulding flow was also considered in this work (Ren et al., 2012), but without any validation against experimental results. Nevertheless, this is useful to assess the quality of INNSPH results against a relevant state-of-the-art WCSPH technique.

A Cross rheological model, as described in equation (4.30), was chosen by Ren
et al. (2012), for their non-Newtonian simulation. The rheological parameters chosen in this work, included a zero shear-rate viscosity $\mu_0 = 100 \text{ Pa} \cdot \text{s}$, while $\mu_\infty = 5 \text{ Pa} \cdot \text{s}$, $K = 1$ and $\rho = 1000 \text{ kg/m}^2$. Furthermore, the flow was driven by a uniform inlet velocity of 2 m/s at $y = -115 \text{ mm}$. In this case no gravity force is acting on the fluid particles and the Reynolds number of the flow is estimated at $Re \approx 18$. Figure 5.19 shows the configuration of the flow domain as presented in Ren et al. (2012). Two different configurations of the inner cylinder are considered: the first one places the inner cylinder at the centre of the mould, while in the second one the cylinder is offset by a distance $\Delta x = -22.5 \text{ mm}$ (see Figure 5.21). The flow domain was discretized with an initial particle spacing of $dx = 1.5 \text{ mm}$, matching the discretization of Ren et al. (2012). Moreover, dummy boundary particles (Koshizuka et al., 1998) were utilized.

The results comparing the two SPH methods for the two different domain configurations are shown in Figures 5.20 and 5.21. In general, small differences are observed between the two different methods during the moulding flow. Regarding the shape of the free-surfaces, the voids that appear on either side of the entrance to the circular cavity are predicted to be larger with INNSPH than the method in Ren et al. (2012) (see Figures 5.20, 5.21). Furthermore, there are small differences in the predicted times of flow propagation. Specifically, the first frame depicted in Figure 5.20 was predicted with INNSPH to be at time $t = 0.044 \text{ s}$, and the last frame at $t = 0.156 \text{ s}$.
CHAPTER 5

INNSPH Results & Discussion

Figure 5.20: Comparison of the pressure contours of the WCSPH (Ren et al., 2012) and the INNSPH methods during the filling process of a symmetric mould with a Cross model shear-thinning liquid: a) INNSPH method, b) computational results of Ren et al. (2012).

Figure 5.21: Comparison of the pressure contours of the WCSPH (Ren et al., 2012) and the INNSPH methods during the filling process of an asymmetric mould with a Cross model shear-thinning liquid: a) INNSPH method, b) computational results of Ren et al. (2012).
(Figure 5.20a1 and Figure 5.20a6, respectively). Ren et al. (2012) have marked these frames at timesteps $t = 0.04$ s and $t = 0.17$ s (Figure 5.20b1 and Figure 5.20b6, respectively). Similar behaviour is observed for the asymmetric cavity (Figure 5.21), with almost the same discrepancies in time as stated for the symmetrical problem (Figure 5.20). Nevertheless, since there are no experimental results to provide clear evidence of the real non-Newtonian flow, it is uncertain which of the two different methods bears the closest relationship to reality. It should be noted though, that the agreement between the two techniques is still acceptable.

Figures 5.20 and 5.21 also offer the opportunity for a comparison between the pressure results given by the WCSPH of Ren et al. (2012) and the pressures calculated with INNSPH. As shown, the INNSPH method can predict a much smoother pressure distribution. The noise in the pressure field is almost absent, unlike that observed in the WCSPH method (Ren et al., 2012). A small amount of noise in INNSPH appears only at Figure 5.20a6, when the two fluid streams collide and the free-surface particles transition to fluid-bulk particles. Similar, but more severe behaviour appears in the WCSPH approach. Additionally, the particles of the INNSPH method are more evenly distributed, with no voids appearing in the fluid bulk—unlike the WCSPH technique which seems to suffer from local clumping and stretching of the particles.

Overall, the agreement between the two SPH methods is acceptable. Although there are no experimental results to provide further quantitative validation of this flow, it should be emphasised that INNSPH clearly provides smoother pressure profiles and more uniform particle distributions. Numerical analyses of SPH (Quinlan et al., 2006) demonstrate a first-order error arising due to particle non-uniformity. Therefore, in the absence of a validating third method, one can reasonably postulate that the more uniform distributions provided by INNSPH mean that it is the more accurate approach.
5.7 Quantitative pressure comparisons with WC-SPH

In order to show clearly and quantitatively the improvements of the INNSPH method over the traditional non-Newtonian SPH methodologies, a standard WCSPH method (Monaghan, 2005; Capone et al., 2010) is implemented for comparison. This approach was necessary as reference pressure solutions are very rarely presented in the literature when using WCSPH.

The WCSPH method used here is a standard approach extensively presented in the literature (Morris et al., 1997; Zhu et al., 2010; Monaghan, 2005; Capone et al., 2010): it uses a second-order accurate predictor-corrector time integration scheme to solve the momentum equation (2.13) and the continuity equation for WCSPH takes the form

\[
\frac{d}{dt} \rho = -\rho \nabla \cdot \mathbf{u}. \tag{5.8}
\]

The density time derivative \( \frac{d\rho}{dt} \) of equation (5.8) is approximated as

\[
\left( \frac{d\rho}{dt} \right)_i = \sum m_j \mathbf{u}_{ij} \cdot \nabla W_{ij}. \tag{5.9}
\]

The equation of state used is of the common form:

\[
p_i = B \left( \left( \frac{p_i}{p_0} \right)^\gamma - 1 \right), \tag{5.10}
\]

where \( p_0 \) is the reference density, \( \gamma = 7 \) and the reference pressure \( B \) is given by \( B = \rho_0 c_0^2 / \gamma \), with \( c_0 \) being the speed of sound. For consistency the same kernel equation (4.14) and viscous term calculation (see Section 4.5) as the INNSPH method are used. Capone et al. (2010) used the same viscous formulation in their non-Newtonian WCSPH simulations. Although the shifting methodology has very recently been applied in the context of WCSPH (Shadloo et al., 2011, 2012; Vacondio et al., 2013), it is omitted for the WCSPH simulations here to align with the popular state-of-the-art WCSPH methods in the literature. Density is also filtered with the implementation of a Shepard-filter (Shepard, 1968) every 20 time steps. The cases considered for
comparison with the WCSPH methodology are a still tank case filled with power-law fluids (with both shear-thinning and shear-thickening rheological behaviour), for which analytical results for the pressure are easily acquired, as well as the experimental dam-break case of Komatina and Jovanovic (1997), where both quantitative and qualitative comparisons can be obtained.

5.7.1 Still power-law fluid

The still water case is a common validation experiment of CFD particle methods, since it tests the stability of the method, and the pressure results can be easily validated as the pressure field is purely hydrostatic. In this study a still power-law liquid is considered, for which both a shear-thinning and a shear-thickening case were examined. Obviously, in a still tank rheological phenomena are non-existent, since the velocity field has zero magnitude. Nevertheless, the rheological formulation is included as it will affect the temporal evolution of the error. The computational domain consists of a tank with depth $H = 0.6$ m, while the particle spacing is $d_x = 0.02$ m both for WCSPH and INNSPH. The analytical pressure for such a case is given by

$$p(y) = \rho_0 g (H - y),$$

where $y$ is the height at which the measurement is taken. The density of the liquid is $\rho_0 = 1000$ kg/m$^3$, the speed of sound for the WCSPH is $c = 48.5$ m/s and the viscosity of the power-law model is $\mu = 1.0$ Pa · s. As mentioned previously both a shear-thinning and a shear-thickening fluid are considered with power-law exponents of $N = 0.5$ and $N = 2$ respectively. The simulations are undertaken for a physical time of 10 s.

Figures 5.22a and 5.22b illustrate the evolution of the pressure $\epsilon_{L2}(p)$ relative error, computed using equation (5.4) for the INNSPH method and the WCSPH approach. As illustrated, the relative error of the INNSPH pressure field is consistently at least one order of magnitude smaller than the relative error of the WCSPH method. Moreover, at the beginning of the simulation WCSPH has a substantial discrepancy in the pressure relative to the analytical solution, and after some initial oscillation, the relative error
CHAPTER 5
INNSPH Results & Discussion

(a) Shear-thinning liquid with $N = 0.5$. (b) Shear-thickening liquid with $N = 2$.

Figure 5.22: Comparison of the relative $L_2$ error evolution for a period of 10 s between INNSPH and WCSPH techniques for still power-law liquids.

increases to around $10^{-1}$ after approximately $t = 8$ s. On the other hand, INNSPH predicts a very accurate pressure field from the start of the simulation with initial relative error close to $10^{-3}$, which increases to the order of $10^{-2}$ after 10s. It is worth noting that in the case of the shear-thickening fluid the $\epsilon_{L_2}$ relative error in pressure appears to be dropping for the INNSPH, for increasing simulation times. Furthermore, the rate of increase of the error over time for the INNSPH shear-thinning case, appears to be smaller than the rate of increase of the WCSPH error.

These results show that the INNSPH method does not only produce accurate pressure results, but also offers a notable improvement over the standard weakly compressible SPH methodology in the prediction of the pressure field, which is an important flow characteristic for many engineering and environmental problems.

5.7.2 Dam-break case

In this section we present the performance of the INNSPH and the WCSPH techniques in predicting pressure results for a free-surface non-Newtonian flow based on the dam-break case of Komatina and Jovanovic (1997). The flow parameters as well as the computational domain characteristics are the same as the ones presented in Section 5.4. For the WCSPH method, a reference density $\rho_0 = 1200 \text{ kg/m}^3$ and speed of sound $c_0 =$
CHAPTER 5
INNSPH Results & Discussion

Figure 5.23: Comparison of free-surface profiles by INNSPH and WCSPH for the Komatina and Jovanovic dam-break case (Komatina and Jovanovic, 1997).

Figure 5.24: Detail of calculated pressures by INNSPH and WCSPH at bottom left corner of dam-break flow presented in Figure 5.23: a) INNSPH, b) WCSPH.

Figure 5.25: Detail of calculated pressures by INNSPH and WCSPH as in Figure 5.24 for the pressure range $800 \text{ Pa} \leq P \leq 1000 \text{ Pa}$: a) INNSPH, b) WCSPH.
20 m/s are used. Figures 5.23, 5.24 and 5.25 present the comparisons of the pressure fields and the free-surface profiles between the INNSPH and the WCSPH methods. Specifically, Figure 5.23 shows a comparison of the free-surface profiles between the two methods, while Figure 5.24 presents a closer view of the particle distribution and the calculated pressure field in the proximity of the left wall. Figure 5.25 presents the same results as Figure 5.24, but in a limited pressure range ($800 \text{ Pa} \leq p \leq 1000 \text{ Pa}$).

As shown, the agreement between the two methods is good in terms of the free-surface flow profiles, although WCSPH has a slightly more developed leading edge of the collapsing fluid column. In Figures 5.24 and 5.25 the particle and pressure distributions of the two techniques can be examined in greater detail. The main differences are that WCSPH shows some clumping of the particles around the left-bottom corner of the computational domain, while INNSPH shows some bunching of the free-surface particles. Importantly, although in this case WCSPH manages to present a relatively smooth pressure distribution, there are still fluctuations in the pressure field close to the bottom wall which are clearly seen in Figure 5.25. These fluctuations are not present in the INNSPH method. Moreover, the pressure on the free-surface of the WCSPH method appears to have increased values relative to the theoretical value of zero expected at the free-surface (see Figure 5.24).

### 5.8 The importance of shifting for non-Newtonian flows

Non-Newtonian flows in many cases involve low-velocity or creeping flows where particle instabilities might be negligible. Therefore, in many cases the shifting algorithm might supposedly be an unnecessary measure.

In this section some of the cases presented in the current chapter are revisited and examined without the use of the shifting algorithm. As it is shown in the following paragraphs the shifting algorithm is essential for the stability of the non-Newtonian implementation of the divergence-free method (Cummins and Rudman, 1998, 1999), even for flows with relatively low Reynolds numbers.
Poiseuille flow. The shifting approach as presented by Xu et al. (2009) and Lind et al. (2012) has been shown to improve the applicability of the method to cases which are prone to instabilities due to particle stretching or clumping. The Poiseuille flows presented in this thesis involve low velocities, which, for the Newtonian case \((N = 1\) and \(\tau_Y = 0 \text{ Pa} \cdot \text{s}\)), correspond to a Reynolds number of unity. For low Reynolds numbers the shifting approach is not needed for a plane-Poiseuille flow since the original divergence-free method of Cummins and Rudman (1998, 1999) remains stable. Nevertheless, increasing the body force \(F\) by an order of magnitude (for a Newtonian flow this would correspond to a Reynolds number of 10) the shifting approach becomes necessary, as instability begins to develop near the boundaries. It is worth noting that even for such Reynolds numbers, non-Newtonian effects are still significant. In Figure 5.26 the particle distributions of a power-law Poiseuille flow with \(N = 0.1\) and \(F = 80 \text{ m/s}^2\) are presented with and without the shifting formulation. As shown, the particles near the boundaries become increasingly irregular without the use of shifting, which eventually results in the failure of the simulation after 0.17s before the flow reaches a steady state. Similar behaviour has been observed by Basa et al. (2008) for Newtonian Poiseuille flows with WCSPH. On the other hand, with the addition of the shifting algorithm, the
simulation becomes stable and steady non-Newtonian Poiseuille flows can be attained.

**Power-law moulding flow.** The power-law moulding flow presented by Fan *et al.* (2010) (see Section 5.5) represents a realistic industrial application in which higher pressures take place (order of $10^6$ Pa). The importance of shifting is tested for this type of flow as well. In Figure 5.27 the comparison of the particle distributions for this case with and without a shifting approach are presented at time $t = 0.034$ s. As shown, if the shifting algorithm is excluded (Figure 5.27b) the original velocity divergence-free method (Cummins and Rudman, 1998, 1999) suffers from increased particle clumping near the paddle region. The clumping of the fluid particles is so severe in this case, that numerical instabilities quickly follow, resulting in the failure of the computational experiment before the completion of the simulation. With shifting (Figure 5.27a), however, the particles are well-distributed and the simulation is stable.

**Cross model complex fillings.** As with the power-law moulding flow presented by Fan *et al.* (2010), the Ren *et al.* (2012) case (see Section 5.6) represents a non-Newtonian moulding flow with real industrial application. The importance of the shifting methodology is tested here also, by excluding the shifting step from the INNSPH algorithm and comparing with the original (shifted) INNSPH results. In Figure 5.28 the comparison of the particle distributions for the case of a Cross complex filling with and without the shifting algorithm is presented. As can be seen, without the shifting algorithm severe instabilities start to develop near the point of impact on the central
cylinder, which quickly results in the failure of the numerical simulation.

It is clear that the shifting formulation is important for the stability of the method when applied to non-Newtonian flows. Remarkably, for industrially relevant cases it appears that the shifting approach is essential within ISPH to maintain a stable simulation and thus to acquire computational results.

5.9 Summary

In this chapter the INNSPH method introduced in Chapter 4, has been applied to non-Newtonian flows. Validation of the methodology against analytical solutions for internal flows, as well as experimental and computational results for free-surface flows of inelastic non-Newtonian fluids was performed. Some of the test cases considered included highly pressurised, highly viscous, paddle driven moulding flows, as well as complex filling processes of circular moulds of different geometries. It has been shown that the proposed methodology accurately estimates the flow profiles of a variety of inelastic non-Newtonian fluid models (Bingham, Cross, power-law, Herschel-Bulkley).

Comparisons of the proposed method were also held against other computational techniques. A finite volume method available with the commercial software ANSYS-
CFX was implemented for the case of the Bingham dam-break and the power-law moulding flow. Comparisons showed that INNSPH matches closely the pressure results estimated by the finite volume method. Moreover, when comparisons with a WCSPH algorithm were performed, the improved quality and accuracy of the pressure results from INNSPH was clearly illustrated.

Finally, comparisons of the INNSPH method with the original divergence-free ISPH method (Cummins and Rudman, 1999, 1998) (non-shifted INNSPH) were shown. It was clearly illustrated that the shifting formulation is very important for the application of the divergence-free method of Cummins and Rudman (1998, 1999) in general non-Newtonian flows. Even for relatively low Reynolds numbers the shifting algorithm is required to maintain the stability of the non-Newtonian simulations. Specifically, when industrially relevant applications were tested, it was found that the shifting algorithm plays a crucial role in the stability of the method.

In the next chapter, the INNSPH method is further extended to multi-phase flows. The mathematical model that allows the modelling of multiple phases is presented, along with applications of the INNSPH method, which include both Newtonian/Newtonian and Newtonian/non-Newtonian two-phase problems. Comparisons are held against experimental, analytical and other computational data and show that the INNSPH method offers further improvements compared with other state-of-the-art multi-phase computational methods.
Chapter 6

Extension of INNSPH to Multi-phase Flows

6.1 Introduction

In the previous chapters the INNSPH method has been introduced (Chapter 4) and successfully applied in single-phase, internal and free-surface non-Newtonian flows (Chapter 5). Comparisons with analytical, computational and experimental results showed that INNSPH is accurately predicting important flow field variables such as the velocity and pressure fields. Moreover, it clearly illustrated that INNSPH performs better than previously published non-Newtonian SPH techniques. Shifting has also proven to be an important computational aspect of the method, allowing the application of the divergence-free method (Cummins and Rudman, 1998, 1999) in demanding non-Newtonian simulations.

In the current chapter the INNSPH method is extended to multi-phase flows. A standard multi-phase SPH formulation is introduced (Hu and Adams, 2007; Shadloo et al., 2013), which allows the modelling of two-phase problems. The multi-phase INNSPH method is then applied to a selection of two-phase benchmark cases. Firstly, comparisons with analytical solutions are conducted for both Newtonian/Newtonian and Newtonian/non-Newtonian two-phase interactions, with a two-phase Poiseuille flow and low-amplitude gravity driven two-phase waves. Then the more complex appli-
cation of the Rayleigh-Taylor instability is considered, for which comparisons are made with computational results. Furthermore, a Newtonian/ non-Newtonian multi-phase problem of environmental interest is considered with the submarine landslide of Assier-Rzadkiewicz et al. (1997), which is validated against experimental and computational results. In the next paragraphs it is clearly shown that INNSPH can accurately predict complex multi-phase flows and offer improved results compared with state-of-the-art CFD techniques.

6.2 Multi-phase formulation

The INNSPH method as explained in Chapter 4 is based on the projection ISPH method (Cummins and Rudman, 1998, 1999) with the addition of shifting, first suggested by Xu et al. (2009) and further improved by Lind et al. (2012) and Skillen et al. (2013). To allow the implementation of multiple flows it is necessary to introduce a colour function:

\[ c_i = \begin{cases} 0 & \text{for phase } a \\ 1 & \text{for phase } b \end{cases} \]  

(6.1)

for each particle, \( i \) (Hu and Adams, 2007; Shadloo et al., 2013), enabling the different fluid phases to be identified. As it can be seen, the values that are given to \( c_i \) could represent the volume concentration of phase \( b \). According to the values of the colour function \( c \) the flow parameters of each phase are assigned.

By solving the governing equations (2.12) and (2.13) combined with a colour function (equation 6.1), multi-phase flows can be modelled with SPH in a straightforward manner. Nevertheless, because of the sharp change of the transport parameters at the interface, instabilities might occur, especially when solving the pressure-Poisson equation, which is solved once per timestep for the entire domain (rather than separately for the different phases). Thus, a smoothed colour function

\[ \tilde{c}_i = \frac{\sum_j c_j W_{ij}}{\sum_j W_{ij}} \]  

(6.2)

is used (Hu and Adams, 2007; Shadloo et al., 2013). The density \( \rho \) and the viscosity
μ at particle $i$ is then updated with

$$\rho_i = (1 - \tilde{c}_i)\rho_a + \tilde{c}_i\rho_b ,$$  \hspace{1cm} (6.3)

and

$$\mu_i = (1 - \tilde{c}_i)\mu_a + \tilde{c}_i\mu_b ,$$  \hspace{1cm} (6.4)

where $\rho_a$ and $\mu_a$ is the physical density and viscosity of phase $a$, which remains constant through the simulation (Shadloo and Yildiz, 2011; Shadloo et al., 2013).

For the case of the Rayleigh-Taylor instability (discussed in detail in Section 6.5.4), Figure 6.1 shows the effects of the smoothed density and viscosity around the interface. In Figure 6.1b it is clear that without any smoothing of the density near the interface, non-physical results are occurring, with apparent instabilities of the interface and excessive diffusion of the two phases. On the other hand, when viscosity is not smoothed (Figure 6.1c) only small differences are found comparing with the smoothed viscosity case (Figure 6.1a). Nevertheless, by smoothing the viscosity on the interface a continuity of stresses is satisfied, since on the interface we have:

$$\tau_a = \tau_b = (0.5\mu_a + 0.5\mu_b)D .$$ \hspace{1cm} (6.5)

It should be noted that the interface for the current method is tracked with the $\tilde{c}_i = 0.5$ criterion. Thus, the continuity of stresses on the interface, as shown in equation (6.5), is always maintained.

As is shown in the previous sections of the single-phase INNSPH method, density $\rho$ is treated as a constant throughout the computational domain, since incompressibility is fully maintained. Although this principle remains in the multi-phase extension of the method, different values of $\rho$ need to be allowed for each particle, in order to accommodate the different density values for the different phases as well as the smoothed values of the density near the interfaces. Thus, standard SPH equations with variable density have to be restored. In this work we use the following:
CHAPTER 6
Extension of INNSPH to Multi-phase flows

Figure 6.1: Comparison of the effect of density and viscosity smoothing (equations 6.3 and 6.4) in the evolution of the flow for a Rayleigh-Taylor instability case (described in Section 6.5.4).

- **Laplacian** by Shao and Lo (2003):

\[
\nabla \cdot \left( \frac{1}{\rho} \nabla A \right)_i = \sum_j m_j \frac{8}{(\rho_i + \rho_j)^2} A_{ij} \mathbf{r}_{ij} \cdot \nabla_i W_{ij} \left/ \frac{v_{ij}^2}{\eta^2} \right. . \quad (6.6)
\]

- **Gradient and divergence** by Oger et al. (2007):

\[
\nabla A_i = \sum_j \frac{m_j}{\rho_i \rho_j} (-A_i + A_j) \nabla_i W_{ij} . \quad (6.7)
\]

- **Viscous term** by Shao and Lo (2003):

\[
\left( \frac{1}{\rho} \nabla \cdot \tau \right)_i = \sum_j m_j \left( \frac{\tau_i}{\rho_i} + \frac{\tau_j}{\rho_j} \right) \cdot \nabla_i W_{ij} . \quad (6.8)
\]

The aforementioned operators are well validated (Shao and Lo, 2003; Oger et al., 2007; Hosseini et al., 2007; Ataie-Ashtiani and Shobeyri, 2008) and fit the purpose of the study, to model non-Newtonian multi-phase flows.
CHAPTER 6  

Extension of INNSPH to Multi-phase flows

6.3  Shifting treatment for multi-phase flows

6.3.1 Multi-phase shifting formulation

The ISPH with Fickian shifting method has initially been introduced by Lind et al. (2012) and later developed by Skillen et al. (2013). These two shifting approaches which read as:

$$\delta r_s = -\mathcal{D} \nabla C \quad \text{with} \quad \mathcal{D} = 0.5h^2$$  \hspace{1cm} (6.9)

and

$$\delta r_s = -Ah\|u\|_i \Delta t \nabla C$$  \hspace{1cm} (6.10)

by Lind et al. (2012) and Skillen et al. (2013) respectively, have been tested for the Rayleigh-Taylor instability test-case (see Section 6.5.4 for the definition of the problem) and although they were found to be stable throughout the simulation, small deficiencies were recognised for both these techniques. Namely the method of Lind et al. (2012) was found to cause increased diffusion of the particles between the two phases (see circled areas of Figure 6.2a), while in the shifting formulation of Skillen et al. (2013), particles that belonged to low velocity regions of the flow field were found to clump (see circled areas of Figure 6.2b). To improve these deficiencies a new shifting formulation is proposed which combines the formulations of Lind et al. (2012) and Skillen et al. (2013) as:

$$\delta r_s = \begin{cases} 
-Ah\|u\|_i \Delta t \nabla C, & \mathcal{D} \geq \mathcal{D}_0 \\
-\mathcal{D}_0 \nabla C, & \mathcal{D} < \mathcal{D}_0 
\end{cases}$$  \hspace{1cm} (6.11)

where $\mathcal{D}_0$ is the lower threshold of the diffusion coefficient. For the simulations presented in this study $\mathcal{D}_0$ is chosen as $\mathcal{D}_0 = 0.01h^2$, which was found to give more uniform particle distribution. As shown in Figure 6.2c the proposed methodology manages to minimise the deficiencies of the pre-existing methods, by maintaining a good particle distribution throughout the computational domain and minimising the diffusion of different phases’ particles. It should again be noted that the proposed methodology is meant to optimise the particle distribution, rather than solve any stability problems, since both pre-existing shifting algorithms by Lind et al. (2012) and Skillen et al. (2013)
Figure 6.2: Comparison of three different shifting approaches for the Rayleigh-Taylor instability case (discussed in detail in Section 6.5.4).

Figure 6.3: Shifting treatment near and on the interface.

have proven to be stable alternatives.

6.3.2 Treatment of the interface

With the above equations, the divergence-free ISPH method with shifting can be modelled for multi-phase flows. Attention must be given to the incorporation of the shifting algorithm, since uniform shifting, with no consideration of the different phases, might lead to artificial mixing of the different phase particles. Therefore, a strategy similar to the treatment of free-surface particles is followed (Skillen et al., 2013; Lind et al., 2012), where the particles of one of the two phases (say phase $b$) that lie within the interface are not shifted in a direction normal to the interface. To recognise the interface
Figure 6.4: Comparison of the proposed multi-phase method with and without the interface shifting treatment.

particles a criterion of

\[ \nabla \cdot \mathbf{r}_b \leq 1.6 \]

is adopted for two-dimensional simulations, where \( \mathbf{r}_b \) refers to the positions of particles belonging to phase \( b \). As shown in Figure 6.3, this treatment is given only to the interface particles of one phase. The particles of the other phase are shifted uniformly, so that a regular particle distribution can be retained. For the simulations of this thesis the shifting treatment on the interface is applied only to the phase with the largest density, although the method is found to have equal effectiveness when applied to the alternate phase instead.

Figure 6.4 clearly demonstrates the improvements that this treatment offers to the method. In Figure 6.4a is shown that uniform shifting with no treatment of the interface has a diffusion effect between the particles of different phases. On the other hand, when normal-to-the-interface shifting is restricted, a much sharper interface profile is achieved. In the application section of this chapter (see Section 6.5) it is clearly illustrated that this approach offers significant improvements in the quality of the interface description, even for complicated flows. It should also be noted that similar treatment of the interface has been separately introduced in WCSPH by Mokos (Mokos, 2014; Mokos et al., 2014) which is applicable to large density ratios of 1 : 1000.
6.4 Limitations

The proposed multi-phase formulation has been developed to calculate efficiently two-phase problems relevant to environmental flows with small density ratios ($\rho_a/\rho_b < 3$, where $a$ and $b$ denote separate phases). This limitation arises from the pressure-Poisson solver which is meant to handle single-phase constant density flows. Although, with the introduction of the smoothed color function of equation (6.2) fluids with different densities can be handled, nevertheless, the stability of the method is limited to lower density ratios. As is discussed in detail in Section 8.3, different approaches can be followed in order to accommodate larger density ratios. Nevertheless, they are not considered in this study, since the time frame of this thesis could not allow for the extra programming and computational time needed. It should be noted, that the density ratio limitation discussed here only slightly limits the applicability of the INNSPH method, since most Newtonian/ non-Newtonian incompressible multi-phase problems of environmental interest fall into the aforementioned density ratio range of $\rho_a/\rho_b < 3$.

The method as presented here is also limited to two-phase flows, although more phases can be easily implemented by adopting a different colour function. Turbulence is also not considered at this stage of the study, since the modelling of such phenomena was not part of the initial outline of this project. A turbulent $k-\epsilon$ model is introduced in the latter stages of this work (see Section 7.4) but is not applied to the simulations showed in the current chapter, which also aligns with the methods with which comparisons are held.

6.5 INNSPH multi-phase validation and applications

6.5.1 Introduction

As presented in this chapter, appropriate steps have been taken to expand the INNSPH method to multi-phase flows. In the following subsections two-phase problems are examined including analytical solutions of two-phase Poiseuille flows (with both New-
tonian and non-Newtonian rheological characteristics) and a low-amplitude gravity driven wave flow. The benchmark case of the Rayleigh-Taylor instability is also investigated with thorough comparisons with other computational methods (Shadloo et al., 2013; Quinlan et al., 2014; Hu and Adams, 2007; Szewc et al., 2015). Although most of the aforementioned problems involve only Newtonian phases (with the exception of the Newtonian/ non-Newtonian Poiseuille flows), it is essential that these benchmark cases have been considered for a thorough validation of the proposed numerical method.

Having validated the multi-phase formulation for Newtonian and non-Newtonian flows, a case with environmental interest is considered, involving Newtonian/ non-Newtonian interactions. Specifically, the submarine landslide case is presented for the experiment of Assier-Rzadkiewicz et al. (1997) which is modelled and compared against other SPH methods (Ataie-Ashtiani and Shobeyri, 2008; Capone et al., 2010). This enables the assessment of the effectiveness and the deficiencies of the proposed technique for both Newtonian/ Newtonian and Newtonian/ non-Newtonian multi-phase flows and to establish the applicability of this method in complex, realistic environmental applications.

### 6.5.2 Two-phase Poiseuille flows

Poiseuille flows are a standard benchmark case, most commonly used for the validation of single-phase algorithms (e.g. Vakilha and Manzari, 2008; Zhu et al., 2010; Ren et al., 2012). Nevertheless, this test case is equally useful for the validation of multi-phase methods although it is very rarely found in the literature (e.g. Ghaitanellis et al., 2015). Closed-channel plane Poiseuille flows are useful because the shear phenomena between multiple phases can be easily tested. The continuity of the stresses at the interface between two phases is a particularly important characteristic that needs to be maintained and can be tested with this problem. In the following, comparisons for different sets of Newtonian/ Newtonian and Newtonian/ non-Newtonian flows are presented.
CHAPTER 6

Extension of INNSPH to Multi-phase flows

Figure 6.5: Configuration of the Poiseuille flow of two immiscible fluids with \( \mu_b > \mu_a \) as presented in Bird et al. (2007): Arrows represent the expected velocity profiles.

6.5.2.1 Newtonian flows

Bird et al. (2007) have derived analytical expressions for the steady-state flow profile of two immiscible Newtonian fluids. As shown in Figure 6.5 the interface between the two fluids is located in the middle of the channel at \( y = 0 \), so that the channel is half filled with fluid \( a \) and half filled with fluid \( b \).

In the simulations presented herein, the channel is discretised with 40 fluid particles distributed across the channel of width \( 2L = 2 \) m, while periodic boundary conditions were used along the \( x \) direction. Since the steady state solution is independent of the density ratio of the two phases, density values of \( \rho_a = \rho_b = 1 \) kg/m\(^3\) were chosen\(^1\). The viscosity of phase \( a \) was constant and equal with \( \mu_a = 1 \) Pa \( \cdot \) s, while different viscosity values were assigned to phase \( b \) to examine different viscosity ratios between the two phases. The flow is driven by a body force \( F_x = 2 \) m/s\(^2\) which for \( \mu_b/\mu_a = 1 \) corresponds to a peak velocity of \( u = 1 \) m/s and a Reynolds number of \( Re = 2 \).

Figure 6.6 shows comparisons between the analytical solution and the ISPH results for the case of two-phase Poiseuille flows with different viscosity ratios. The comparisons are presented for the normalized velocities \( \tilde{u} = u/u_0 \), where \( u_0 \) is the average channel velocity. As shown, the proposed method manages a close agreement with the analytical solutions, with the errors increasing for larger viscosity ratios \( \mu_b/\mu_a \). Moreover, a convergence study has been conducted for a viscosity ratio of \( \mu_b/\mu_a = 4 \).

---

\(^1\)When higher density ratios were tested (within the margin described in the limitations, see Section 6.4) the same results were acquired.
CHAPTER 6  Extension of INNSPH to Multi-phase flows

Figure 6.6: Velocity comparisons between ISPH and analytical solutions for two-phase Poiseuille flows: ● ISPH, — analytical results.

Figure 6.7: Convergence results of ISPH, conducted for a viscosity ratio of $\mu_b/\mu_a = 4$. The solid line indicates an order of spatial convergence of approximately 1.0.

As shown in Figure 6.7, the velocity $\epsilon_{L2}$ relative error

$$
\epsilon_{L2} = \sqrt{\frac{\sum (\tilde{u}_{\text{calculated}} - \tilde{u}_{\text{analytical}})^2}{\sum \tilde{u}_{\text{analytical}}^2}}
$$

(6.13)

indicates linear convergence of the method, with a calculated convergence rate of 1.0. Evidently the accuracy of the method improves for increasing spatial resolution.
6.5.2.2 Newtonian/ Non-Newtonian interaction

Having validated the method against two Newtonian phases with different viscosities, the method is then applied to Newtonian/ Non-Newtonian interactions. Similar with the Newtonian case, the flow domain is described by a channel of height \(2L = 2\) m, discretised with 40 fluid particles along the height, while periodic boundary conditions were used along the \(x\) direction. The interface is located in the middle of the channel \(y = 0\), with the two phases covering exactly half of the channel each (Figure 6.5).

In this case, opposite to the Newtonian case, analytical solutions could not be found in the literature. Thus, an analytical expression for interaction between a Newtonian and a Bingham fluid and a semi-analytical expression for the interaction between a Newtonian and a power-law non-Newtonian fluid were derived as described in detail in Appendix B. Both solutions refer to the steady-state of the flow. Again, density ratios are taken equal with \(\rho_a/\rho_b = 1\). The viscosity of the Newtonian phase is fixed at \(\mu_a = 1\) Pa\(\cdot\)s while for the non-Newtonian phase \(b\) the parameter changing between different test-cases is the yield stress \(\tau_Y\) for the Bingham fluid and the power-law exponent \(N\) for the power-law fluid. Again, the flow is driven by a body force \(F_x = 2\) m/s\(^2\) which for \(\tau_Y = 0\) Pa and \(N = 1\) corresponds to a peak velocity of \(u = 1\) m/s and a Reynolds number of \(Re = 2\).

Figure 6.8 presents the comparison between the INNSPH results and the analytical solution for a Newtonian/ Bingham visco-plastic fluid interaction. As shown the comparison between the two solutions is very close, with the INNSPH matching closely the analytical solution. As shown the INNSPH manages to capture with reasonable accuracy both the transition from the yielded zone of the fluid (where \(|\tau| > \tau_Y\)) to the solid-zone of the flow (\(|\tau| \leq \tau_Y\)) and the transition from the solid zone of the Bingham fluid to the Newtonian fluid of phase \(a\).

In Figure 6.9 the INNSPH method is compared against semi-analytical solutions for Newtonian/ power-law two-phase interactions. For this case both shear-thinning (6.9a and 6.9b) and shear-thickening (6.9c and 6.9d) are considered with \(N < 1\) and \(N > 1\) respectively. Similarly to the Poiseuille cases already shown, the Newtonian/ power-law cases present a very good agreement. It is shown that the transition from the non-Newtonian phase \(b\) to the Newtonian phase \(a\) is captured with great accuracy, both
CHAPTER 6
Extension of INNSPH to Multi-phase flows

(a) $\tau_Y = 1 \text{ Pa}$   
(b) $\tau_Y = 1.5 \text{ Pa}$

Figure 6.8: Velocity comparisons between ISPH and analytical solutions for two-phase Newtonian/Bingham Poiseuille flows: • ISPH, – analytical results.

(a) $N = 0.25$   
(b) $N = 0.125$   
(c) $N = 2$   
(d) $N = 8$

Figure 6.9: Velocity comparisons between ISPH and semi-analytical solutions for two-phase Newtonian/power-law Poiseuille flows: • ISPH, – semi-analytical results.
for shear-thinning (Figures 6.9a and 6.9b) as well as for shear-thickening phenomena (Figures 6.9c and 6.9d).

From the above it is clear that the proposed method manages to maintain the continuity of the stresses across the separate phases both for Newtonian/Newtonian as well as for Newtonian/non-Newtonian interactions. Specifically from Figures 6.8 and 6.9 it is clearly observed that the INNSPH method manages to predict both the non-Newtonian phenomena taking place in phase \( b \) (including the plug zone of visco-plastic materials) as well as their interaction with the Newtonian fluids of phase \( a \).

The test-cases conducted for the Poiseuille flow, are a good first assessment of the method which is to be applied in more complex two-phase cases of both Newtonian and non-Newtonian rheology in the following sections of this thesis.

### 6.5.3 Two-phase gravity waves

The problem of two-phase gravity waves was studied by Crapper (1984) for thermoclines in oceans, where rapid changes in density are observed. In such instances, two regions with considerably different densities might interact, and cause a standing wave driven by the acceleration of gravity. To study this problem Crapper proposed a simple model of two inviscid fluids, with \( \rho_a \) in the region \( 0 < y < L \) and \( \rho_b > \rho_a \) in the region \( -L < y < 0 \). At \( y = L \) and \( y = -L \) rigid walls are assumed, while surface tension, currents, or mixing of the two phases are omitted. For such flows Crapper (1984), derived that the waves are oscillating with an angular frequency

\[
\omega = \left( Agk \tanh kL \right)^{-1/2},
\]

where \( g \) is the gravitational acceleration, \( k \) is expressed in terms of the wavelength \( \lambda \) as \( k = 2\pi/\lambda \) and \( A \) is the Atwood number

\[
A = \frac{\rho_b - \rho_a}{\rho_b + \rho_a},
\]

which is important for penetrating flows (e.g. the Rayleigh-Taylor instability presented in Section 6.5.4). From equation (6.14) we can derive the period of each oscillation, as
CHAPTER 6
Extension of INNSPH to Multi-phase flows

Figure 6.10: Gravity waves configuration of the computational domain

Figure 6.11: Deviation of the calculated five periods $T_5$ from the analytical solution. Solid line represents rate of convergence, which in this case is approximately 1.0.

\[ T = \frac{2\pi}{\omega}. \]

Figure 6.10 shows the initial configuration of the flow domain. The parameters of the phases $a$ and $b$, chosen for this simulation are $\rho_a = 0.5 \times 10^3 \text{ kg/m}^3$ and $\rho_b = 1 \times 10^3 \text{ kg/m}^3$, corresponding to an Atwood number of $A = 0.333$. The viscosities are chosen as $\mu_a = \mu_b = 0.0 \text{ Pa} \cdot \text{s}$, so that an inviscid flow can be simulated. The flow is accelerated by a gravity force $g = 9.81 \text{ m/s}^2$, while the height of each phase $L$ and the wavelength $\lambda$ are chosen as $L = 0.5 \text{ m}$ and $\lambda = 1 \text{ m}$. The amplitude $\alpha$ of the waves needs to be small relative to the height $L$ to derive the analytical solution of equation (6.14), so an amplitude $\alpha = 0.075 \text{ m}$ is chosen. Periodic boundary conditions are implemented on the $x$ direction, while no-slip boundary conditions are chosen for the solid walls at $y = L$ and $y = -L$.

This problem is simulated for a number of different particle resolutions and comparisons were made for the time length of five periods $T_5$ calculated by INNSPH and the analytical solution, which for the values of the flow characteristics of the current study gives $T_5 = 6.94 \text{ s}$. Table 6.1 and Figure 6.11 show the deviation of the calculated $T_5$ from the analytical solution relative to the particle spacing. It is shown that the convergence of the method is again close to linear with an estimated convergence
Table 6.1: Deviation of the calculated five periods $T_5$ from the analytical solution for the standing gravity waves. Theoretical five periods duration $T_5 = 6.94$ s.

<table>
<thead>
<tr>
<th>Particle spacing $dx$ [m]</th>
<th>Calculated five period duration $T_5$ [s]</th>
<th>Deviation %</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.04</td>
<td>8.39</td>
<td>20.8</td>
</tr>
<tr>
<td>0.02</td>
<td>7.65</td>
<td>10.2</td>
</tr>
<tr>
<td>0.01</td>
<td>7.37</td>
<td>6.14</td>
</tr>
<tr>
<td>0.005</td>
<td>7.29</td>
<td>4.98</td>
</tr>
</tbody>
</table>

It should be noted that this oscillating waves test case is a very demanding case, sensitive to small changes of the parameters. It was found that for the method to be stable, sufficiently small particle resolutions should be used. In fact, the wave amplitude $\alpha$ should be noticeably larger than the particle spacing $dx$, since it was found that for $\alpha/dx \leq 1.5$ the numerical error is significant enough to limit the stability of the method to small computational times.

### 6.5.4 Rayleigh-Taylor instability

The problem of the Rayleigh-Taylor instability consists of two Newtonian fluids with densities $\rho_a$ and $\rho_b$, where $\rho_a > \rho_b$. A gravity force is then accelerating the upper heavier fluid into the lower lighter one causing a mixing of the two phases. This problem is quite interesting to examine, since the phenomena that take place in the Rayleigh-Taylor instability are quite rapid and violent making it a good measure for the performance of the method. Moreover, the phenomena studied in the Rayleigh-Taylor instability are also found in numerous industrial and physical applications (supernova explosions (Wang and Chevalier, 2001), instabilities in plasma fuel reactors (Chen et al., 1994), in rotating gas clouds with an external gravitational potential (Sharp, 1984)). Also, the Rayleigh-Taylor instability is an appropriate test case for the proposed method, since it has been previously examined widely in the literature (Cummins and Rudman, 1999; Shadloo et al., 2013; Quinlan et al., 2014; Hu and Adams, 2007; Szewc et al., 2015; Kruisbrink et al., 2012; Grenier et al., 2012; Monaghan and Rafiee, 2013; Leduc et al., 2009; Chen et al., 2015), offering plenty of material for conduct-
CHAPTER 6
Extension of INNSPH to Multi-phase flows

ing comparisons. Since there are no directly comparable experimental results it is uncertain which of the literature results should be considered as a reference solution, so comparisons with different Lagrangian techniques (Hu and Adams, 2007; Shadloo et al., 2013; Quinlan et al., 2014; Szewc et al., 2015) are considered.

The configuration of the computational domain of this problem is presented in Figure 6.12. As shown, the simulation is initialised with a sinusoidal shape of the interface between the two phases (shown in Figure 6.12), while the flow domain is confined by no-slip boundary conditions at all four sides. The Reynolds number of the flow was chosen as $Re = 420$, which was found to be the most common case in the literature. Moreover the density ratio is $\rho_b/\rho_a = 1.8$ and the viscosity ratio is $\mu_b/\mu_a = 1.8$. The gravitational force is calculated by $Re = \rho_b/\mu_b\sqrt{gh^3}$, where for a characteristic length $H = 1$ m and $Re = 420$ we get $g = 176.4 \times 10^3 \text{ m/s}^2$.

An important non-dimensional parameter for the Rayleigh-Taylor instability is the Atwood number (see Equation 6.15) which relates to the growth rate of the penetration distance and the shape of the interface. It is reported in the relevant literature (Andrews and Dalziel, 2010; Glimm and Majda, 1991) that for high Atwood numbers the interface tends to be bubbly rather than continuous. For this particular case a relatively low Atwood number of $A \approx 0.3$ is measured which accounts for a low growth rate and similarly-shaped interpenetrating fluids with continuous interfaces (instead of bubbly) (Andrews and Dalziel, 2010; Glimm and Majda, 1991).

The nature of the Rayleigh-Taylor instability, where the two phases are interacting violently, makes it an interesting test case for multi-phase methods. Moreover, due to the highly transient phenomena that take place during the evolution of this problem, the capabilities and deficiencies of a multi-phase numerical method can be illustrated. Therefore, it has been chosen as a validation case for several SPH computational studies. The multi-phase method proposed in this thesis will now be compared with the ISPH method of Hu and Adams (2007), the WCSPH technique of Szewc et al. (2015), who used a correction term for the interface, the method of Shadloo and Yildiz (2011), who used an ISPH with shifting approach similar to Xu et al. (2009), but with the addition of surface tension and with the more recent finite volume particle method (FVPM) of Quinlan et al. (2014), which is based on the finite volume method and SPH, having
CHAPTER 6  
Extension of INNSPH to Multi-phase flows

overlapping particles and kernels for the discretization of the flow domain.

Before comparing with previously published results, it is worth conducting a convergence study, in order to establish that the particle resolution used for the simulation, is adequate. Figures 6.13 and 6.14 illustrate the results at the dimensionless time $t = \sqrt{g/H} = 5.0$, by the proposed methodology for $50 \times 100$, $100 \times 200$ and $200 \times 400$ particles, with corresponding particle spacings of $dx = 0.02$ m, $dx = 0.01$ m and $dx = 0.005$ m respectively. As illustrated the method is clearly converging, with the interface differences for the particle spacings of $dx = 0.01$ m and $dx = 0.005$ m being very small. Specifically, in Figure 6.14 it is clearly shown that for increasing particle resolution, the interface description is consistently converging to a very sharp representation of the interface profile. Thus, the particle resolution of $dx = 0.005$ m is considered adequate.

In Figure 6.15 a longer evolution of the flow is also presented for a particle spacing of $dx = 0.005$ m. It is clear that during the rapid evolution of a Rayleigh-Taylor instability, very intense phenomena take place. The two phases are rapidly mixed, and a very irregular interface profile is formed. It is also shown that the proposed method can reproduce a very smooth interface profile.

Having justified the resolution used for the simulation of the Rayleigh-Taylor instability we compare the results with previously published Lagrangian techniques. Figure 6.16 shows the calculated flow profiles of the proposed method, the ISPH method
of Shadloo and Yildiz (2011) and the FVPM of Quinlan et al. (2014) at the dimensionless time $\bar{t} = 5.0$. The agreement between the current work and the other techniques is very close, with perhaps larger differences shown with the ISPH method of Shadloo and Yildiz (2011). In fact the method of Shadloo and Yildiz seems to over-predict the development of the flow, since similar results are predicted by the proposed technique.
CHAPTER 6

Extension of INNSPH to Multi-phase flows

Figure 6.15: Evolution of the Rayleigh-Taylor instability of $Re = 420$, $dx = 0.005$ m.

at later timesteps. Figure 6.16c which represent the results of Quinlan et al. (2014) shows a much better agreement with the multi-phase ISPH technique proposed here. There are some minor differences between the methods, in particular in the smaller scale structures, such as the inner core region of the roll-up (see the boxed area of Figure 6.16c). For these minor differences, resolution discrepancies may have an effect, with Quinlan et al. (2014) using three times smaller particle spacing ($1.66 \times 10^{-3} \text{m}$ comparing to $5.0 \times 10^{-3} \text{m}$ of ISPH).

The proposed method also shows a sharper interface profile comparing with other SPH techniques. In Figure 6.17 which represents a comparison between the method of Hu and Adams (2007) and the current work for a similar number of particles, the proposed method shows a much smoother distribution of the particles and a sharper interface. The approach of Hu and Adams (2007), on this occasion seems to suffer from the dispersion of particles into the different phases, whereas the multi-phase ISPH method proposed herein shows a much more regular distribution of the particles, with clear surface representation of the two phases, even for this low resolution. It is worth noting that in the simulation of the image 6.17a, 5000 fluid particles were used, whereas Hu and Adams (2007) used 7200 particles, which illustrates the advantageous performance of the proposed technique. Similarly in Figure 6.18 the proposed method is compared against the WCSPH results of Szewc et al. (2015). Both for lower and higher
Figure 6.16: Comparison of different methodologies for Rayleigh-Taylor of \( Re = 420 \) at \( \bar{t} = 5.0 \)

Figure 6.17: Comparison between current study and Hu and Adams (2007) results for similar number of particles at \( \bar{t} = 5.0 \)

 resolutions (Figures 6.18a and 6.18b) the proposed method shows a better quality representation of the interface against the WCSPH results (Szewc et al., 2014, 2015),
where the interface was enhanced with a correction term. It should also be noted that the proposed methodology matches almost identically the evolution of the interface as predicted by Hu and Adams (2007) and Szewc et al. (2015) (Figures 6.17 and 6.18, respectively) for the dimensionless time $\bar{t} = 5.0$.

### 6.5.5 Submarine landslide

The phenomenon of submarine landslides is very common in nature, and usually is connected to earthquake activities. Moreover, the occurrence of a submarine landslide can subsequently cause tsunami waves, with devastating consequences. An example of such a case is the 1998 Sissano tsunami, later proven to be caused by a submarine landslide (Tappin et al., 2001). Therefore, submarine landslides have attracted the attention of the scientific community and have been the target of numerous experimental (e.g. Assier-Rzadkiewicz et al., 1997) and computational (e.g. Assier-Rzadkiewicz et al., 1997; Capone et al., 2010; Ataie-Ashtiani and Shobeyri, 2008) studies.

Additionally, the collapsing mass of mud or sand, which causes the landslide, exhibits a visco-plastic rheological behaviour (Komatina and Jovanovic, 1997), often modelled with a Bingham rheological model (see equation 2.20). The interaction between the mud-phase and the displaced water masses also depends on the continuity of the stresses along the interface. Thus, the submarine landslide is a good test case for the performance of the proposed method in Newtonian/non-Newtonian multi-phase problems, and specifically in realistic environmental applications.
For this study the experimental model of Assier-Rzadkiewicz et al. (1997) is examined. The configuration of the computational domain (see Figure 6.19) consists of a tank with a 45° inclined wall. The overall length and height of the tank are 4 m and 2 m respectively. On the top of the inclined surface lies a mass of sand with density $\rho_a = 1950 \text{ kg/m}^3$, while the rest of the tank is filled with water of density $\rho_b = 1000 \text{ kg/m}^3$ and viscosity $\mu_b = 0.001 \text{ Pa} \cdot \text{s}$, up to a depth of 1.6 m (measured from the lower wall). The Bingham fluid is modelled with a bilinear rheological model (equations (4.27) and (4.28)). The parameter $\alpha$ of equation (4.27) is chosen as $\alpha = 10^3$, which was found to give the best results. The Bingham rheological parameters were the same as the ones suggested by Capone et al. (2010) with the viscosity being $\mu_a = 1.0 \text{ Pa} \cdot \text{s}$ and the yield stress $\tau_Y = 1000 \text{ N/m}^2$. It is worth noting here, that all the rheological parameters in this study were chosen after conducting computational experiments, since the original experiment of Assier-Rzadkiewicz et al. (1997) provided only the density of the sand, omitting to give information about the Bingham viscosity and the yield stress. The same approach was used in the previous published work in the literature (Assier-Rzadkiewicz et al., 1997; Capone et al., 2010; Ataie-Ashtiani and Shobeyri, 2008), where the viscosity and the yield stress were determined computationally. Table 6.2 shows the set of viscosity and yield stress values used in the relevant literature (Assier-Rzadkiewicz et al., 1997; Capone et al., 2010; Ataie-Ashtiani and Shobeyri, 2008). The computational domain was discretized with 9673 particles, with particle spacing $dx = 0.02 \text{ m}$.

This application is the first one presented in this study where the estimated Reynolds number belongs in the turbulent region with $Re \approx 2 \times 10^4$ for the water
CHAPTER 6

Extension of INNSPH to Multi-phase flows

<table>
<thead>
<tr>
<th>Paper reference</th>
<th>Yield stress $\tau_Y$ [N/m$^2$]</th>
<th>Bingham viscosity $\mu$ [Pa·s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Assier-Rzadkiewicz et al. (1997)</td>
<td>1000</td>
<td>0.0</td>
</tr>
<tr>
<td>Ataie-Ashtiani and Shobeyri (2008)</td>
<td>750</td>
<td>0.15</td>
</tr>
<tr>
<td>Capone et al. (2010)</td>
<td>1000</td>
<td>1.0</td>
</tr>
<tr>
<td>INNSPH</td>
<td>1000</td>
<td>1.0</td>
</tr>
</tbody>
</table>

Table 6.2: The set of viscosity and yield stress values used in the relevant literature

phase. However, turbulent modelling is not considered for this test-case, which might cause some deficiencies in the estimated results. Nevertheless, it is worth noting that the cases against which comparisons are held (Assier-Rzadkiewicz et al., 1997; Capone et al., 2010; Ataie-Ashtiani and Shobeyri, 2008), have also omitted the modelling of turbulent effects. A very low Froude number of $Fr \approx 0.2$ has also been calculated, characterising the flow as sub-critical.

The results of the INNSPH method are compared with the experimental results of Assier-Rzadkiewicz et al. (1997), the WCSPH computational results of Capone et al. (2010) and the density invariant ISPH method of Ataie-Ashtiani and Shobeyri (2008). The experimental results presented by Assier-Rzadkiewicz et al. (1997) are at times $t_1 = 0.4$ s and $t_2 = 0.8$ s and they refer only to the water free-surface. Figures 6.20 and 6.21 illustrate the comparison of the INNSPH results with the experimental results of Assier-Rzadkiewicz et al. (1997), the WCSPH results of Capone et al. (2010) and the density invariant ISPH method of Ataie-Ashtiani and Shobeyri (2008) for the water free-surface at times $t_1 = 0.4$ s and $t_2 = 0.8$ s. As shown the INNSPH method shows close agreement with the experimental results, predicting the water free-surface. Moreover, it is illustrated that in some regions of the computational domain the INNSPH method performs better than the WCSPH (Capone et al., 2010) and the density invariant (Ataie-Ashtiani and Shobeyri, 2008) approaches, e.g. at the second half of the computational domain ($x > 2$ m) at $t_2 = 0.8$ s (Figures 6.20b and 6.21b), where the INNSPH predicts exactly the free-surface profile, unlike the overpredicted WCSPH results.

In order to quantify any improvements of the proposed methodology, the discrep-
CHAPTER 6  
Extension of INNSPH to Multi-phase flows

(a) $t_1 = 0.4$ s.

(b) $t_2 = 0.8$ s.

Figure 6.20: Comparison between INNSPH, experimental results (Assier-Rzadkiewicz et al., 1997), the WCSPH (Capone et al., 2010) and the density invariant ISPH (Ataie-Ashtiani and Shobeyri, 2008) results.

Figure 6.21: Water free-surface comparison between INNSPH, experimental results (Assier-Rzadkiewicz et al., 1997), the WCSPH (Capone et al., 2010) and the density invariant ISPH (Ataie-Ashtiani and Shobeyri, 2008) results.
Table 6.3: Estimated deviation of the water free-surface height at $t_1 = 0.4$ s, $t_2 = 0.8$ s and total error for both timesteps.

\[
\epsilon_{L2} = \sqrt{\frac{\sum (y(x)_{\text{exp.}} - y(x)_{\text{calc.}})^2}{\sum (y(x)_{\text{exp.}})^2}},
\]

where $y(x) = y(x) - H$, with $H = 1.6$ m. Table 6.3 presents the deviation of the calculated results of the three SPH methods (current work, Capone et al. (2010) and Ataie-Ashtiani and Shobeyri (2008)) from the experimental results. As shown the proposed method manages a slightly better agreement with the experimental results, predicting very well the water free surface, with the improvement being more noticeable at time $t_1 = 0.4$ s.

Comparisons are also made for the collapsing sand phase between INNSPH, the WCSPH computational results of Capone et al. (2010) and the NASA-VOF2D method used by Assier-Rzadkiewicz et al. (1997). Figure 6.22 presents the comparison between the three computational techniques. For these simulations, INNSPH and WCPSH (Capone et al., 2010) used a viscosity of $\mu = 1.0$ Pa·s, while the viscosity used by the NASA-VOF2D method (Assier-Rzadkiewicz et al., 1997) was $\mu = 0.0$ Pa·s. As shown the differences between the three techniques are considerable, especially at time $t_2 = 0.8$ s. It can be seen though, that the results of the WCSPH (Capone et al., 2010) and the NASA-VOF2D (Assier-Rzadkiewicz et al., 1997) are exaggerated relative to the INNSPH technique, predicting a more deformed mass of sand. The interface profile also seems more developed for time $t_2 = 0.8$ s with the leading edge of the landslide of Assier-Rzadkiewicz et al. (1997) and Capone et al. (2010) being further ahead than the INNSPH one. This could mean that the INNSPH method predicts...
CHAPTER 6  Extension of INNSPH to Multi-phase flows

Figure 6.22: Mud phase comparison between INNSPH, Nasa-Vof2D results (Assier-Rzadkiewicz et al., 1997) and WCSPH results (Capone et al., 2010).

(a) $t_1 = 0.4$ s.  
(b) $t_2 = 0.8$ s.

6.6 Summary

In this chapter the INNSPH method has been successfully developed to model multi-phase flows. In order to model multi-phase physics a colour function has been introduced which identifies the phase in which the particles belong. It is also used to smooth the sharp density change on the interface. Special shifting treatment is considered for the flow interface, so that artificial mixing of the two phases can be avoided. Moreover, a new shifting formulation has been proposed which optimizes the particle distribution for multi-phase flows.

As discussed in Section 6.4 the proposed method is limited to multi-phase flows with low density ratios. This limitation arises from the Poisson solver, which cannot handle large density variations. Nevertheless, the method is still capable of simulating a great range of incompressible multi-phase flows, without any extra computational

a larger non-yielded area in the Bingham fluid, as opposed to the largely deformed WCSPH (Capone et al., 2010) and NASA-VOF2D (Assier-Rzadkiewicz et al., 1997) methods. The differences may be due to the different Bingham parameters chosen by the different methods (see Table 6.2).
cost coming with SPH methods which allow the modelling of higher density ratios (Hu and Adams, 2009; Lind et al., 2015) (discussed in more detail in Section 8.3).

The multi-phase INNSPH method has then been applied to a selection of demanding two-phase flows. Newtonian/Newtonian interactions were considered with the cases of the Rayleigh-Taylor instability and the gravity driven waves, for which comparisons were held against computational and analytical results respectively. Comparisons with analytical and semi-analytical Newtonian/Newtonian and Newtonian/non-Newtonian interactions were also conducted, with the case of the two-phase Poiseuille flow. It was shown that INNSPH can accurately predict the evolution of the flow fields of challenging flows, with either highly transient phenomena (e.g. the Rayleigh-Taylor instability) or slowly evolving interfaces (e.g. the gravity driven waves). Agreement with Newtonian/non-Newtonian results was also close, showing that the method can predict accurately flows involving both Newtonian and non-Newtonian rheological characteristics.

A more realistic Newtonian/non-Newtonian multi-phase flow with great environmental interest was also considered. The submarine landslide presented in the experiment of Assier-Rzadkiewicz et al. (1997) was modelled and compared with experimental and computational data. As shown the INNSPH method can improve the state-of-the-art SPH results for realistic environmental applications and can be readily applied to inform industrial and environmental problems involving multi-phase non-Newtonian phenomena.

In the next chapter (see Chapter 7) a more challenging environmental application is examined with the 1958 Lituya Bay terrestrial landslide and tsunami. For the representation of this actual event comparisons with experimental work (Fritz et al., 2001) and computational results are conducted. Crucial methodological amendments are considered with the introduction of a $k - \epsilon$ turbulence model (similar to the work of Violeau, 2004; Leroy et al., 2014) and a simple saturation model in order to achieve a good representation of the experimental data (Fritz et al., 2001) for both the wave run-up as well as the collapsing landslide.
Chapter 7

Concluding Environmental Application - Lituya Bay Tsunami and Landslide

7.1 Introduction

In the previous chapters the INNSPH method has been introduced and expanded to multi-phase applications. Validation has been conducted both for internal and free-surface flows covering Newtonian and non-Newtonian rheologies. Comparisons with analytical, computational and experimental results showed that the proposed methodology can accurately predict complex multi-phase phenomena including both highly transient phenomena and non-Newtonian rheological effects. A new shifting approach has also been introduced, which was found to improve the particle distribution in multi-phase flows with highly-transient phenomena.

In the current chapter, the case of the Lituya Bay tsunami and landslide, which occurred in Alaska in 1958 (Miller, 1960) is going to be examined. The terrestrial landslide which caused the tsunami was triggered by an 8.3 Richter magnitude earthquake, resulting in an estimated $3 \times 10^7$ m$^3$ of rock to slide in a bay generating a wave run-up of 524 m and partial overtopping (Miller, 1960). Figure 7.1 shows an aerial photograph of the Lituya Bay fjord overlaid with a graphical representation of the rock phase and the run-up area.
Scientific interest has gathered around this event due to the devastating consequences similar phenomena may have. Experimental (Fritz et al., 2001) and computational (Schwaiger, 2007; Schwaiger and Higman, 2007; Basu et al., 2010) work has been conducted to reproduce the sequence of events around the Lituya Bay tsunami.

In the current work, the INNSPH multi-phase method is applied to the Lituya Bay tsunami and landslide. The work presented in this chapter has two major focus points:

1. **Real scale simulation with gravitational acceleration input:** Firstly, a model similar to the actual Lituya Bay tsunami and landslide is considered. At this stage, the actual scale of the Lituya Bay fjord is modelled with the landslide being accelerated towards the water from rest due to gravitational force. This model aligns with the natural mechanisms of a landslide and has also been adopted by other state-of-the-art CFD approaches (Schwaiger and Higman, 2007; Basu et al., 2010), with which comparison are made. When the INNSPH simulation results are compared with the benchmark experiment of Fritz et al. (2001) it is shown that the wave run-up height is accurately represented, while the rock-entry profile is significantly underpredicted (similarly with Schwaiger and Higman, 2007). This behaviour is expected due to the different input method adopted in the experiment where the landslide is accelerated with the use of a pneumatic accelerator mechanism (Fritz et al., 2001; Fritz and Moser, 2003).

2. **Reproducing the experimental configuration:** The motivation of this work is then shifted towards the accurate modelling of the experimental study of Fritz et al. (2001), aiming to achieve a good representation of both the wave run-up height as
well as the rock-phase entry profile as reported in the experiment. It is understandable that from this perspective the modelling steps taken will deviate from the actual Lituya Bay tsunami and landslide events, while focusing on the reconstruction of the experimental model. Nevertheless, the experimental data are readily available and allow the rigorous validation of the method, while highlighting modelling deficiencies for which appropriate actions are taken. The following paragraphs present in detail all the investigation and modelling steps taken towards the accurate reproduction of the experimental results. An outline of these steps is given here:

- The pneumatic accelerator input as presented in the experiment of Fritz et al. (2001) is reconstructed, aiming to match the experimental conditions for the rock entry. It is found that this approach improves the rock-entry profile but without modification of the formulation for the water and landslide motion, this also causes a significant overshoot on the wave run-up estimation.

- Following this observation the $k - \epsilon$ turbulence model was implemented in the method and validated, aiming to improve the results due to the dissipative effects of the eddy viscosity. The simulations, which now are conducted on the experimental scale, showed only minor improvements.

- Finally, based on the findings of Mitarai and Nakanishi (2012) a simple saturation model for the landslide is introduced, which increases the effective viscosity under the assumption of a saturation period, aiming to capture with better accuracy the landslide and tsunami evolution presented in the experiment.

The effects of the aforementioned modelling steps are discussed in detail in the current chapter and it is shown that with the modified INNSPH method a sufficiently accurate reproduction of the landslide and tsunami can be achieved. Moreover, the investigations presented herein offer an in-depth analysis in the Lituya Bay case covering both the real-life event, as well as the experimental simulations with a comprehensive and detailed manner. The final method is an entirely novel approach, which can offer improvements in environmental and geotechnical modelling.
7.2 Real-scale simulation with gravitational acceleration input

The Lituya bay event is an excellent validation case for the INNSPH multi-phase method, and the implementation of non-Newtonian rheology, since it involves a multi-phase flow with both Newtonian and non-Newtonian phases. Moreover, both computational (Schwaiger, 2007; Schwaiger and Higman, 2007; Pastor et al., 2009; Basu et al., 2010) and experimental (Fritz et al., 2001) results are available for comparisons.

The computational domain selected for this application is the one shown in Figure 7.2 (Fritz et al., 2001; Basu et al., 2010). It involves a two dimensional prismatic water wave channel, with 1000 m height at either side on a 45° degree angle, separated by a water channel of 122 m depth and 1342 m length. The rock phase is represented by a circular segment which is initialised from rest and accelerated due to the gravitational force towards the water mass (Pastor et al., 2009; Basu et al., 2010). This approach is obviously different from the experiment of Fritz et al. (2001), where the granular material representing the rock phase was accelerated towards the water mass with a pneumatic accelerator (Fritz et al., 2001; Fritz and Moser, 2003). Nevertheless, the experiment of Fritz et al. (2001) is still used as a benchmark since it has been specifically built for the Lituya Bay landslide and tsunami event and it presents a detailed evolution of the flow and an accurate representation of the wave run-up height which matches
CHAPTER 7  Lituya Bay Tsunami and Landslide

The two media simulated in this multi-phase problem are water and a non-Newtonian phase representing the landslide. Typical values of density and viscosity are chosen for the water, with $\rho_{\text{water}} = 1000 \text{ kg/m}^3$ and viscosity $\mu_{\text{water}} = 0.001 \text{ Pa \cdot s}$, while for the rock phase a density of $\rho_{\text{landslide}} = 1650 \text{ kg/m}^3$ is chosen, as reported in the experiment of Fritz et al. (2001). A challenge for this validation case is to accurately determine the rheological parameters of the rock phase, since such details are not provided in the literature. Obviously, in real life the rock phase would be constituted by a mixture of materials with different properties (e.g. rocks, vegetation, soil, ice). Moreover, the rock phase is expected to have non-Newtonian shear-thinning rheological properties. To determine the rheological parameters of the rock phase, computational experiments were carried out. The Bingham rheological model (equation 2.20), the power-law (equation 2.17) and the Herschel-Bulkley model (equation 2.21) have been tested. Figures 7.3 and 7.4 show the comparison between the three rheological models and the experimental results of Fritz et al. (2001) for the wave run-up and the height of the wave at position $x = 885 \text{ m}$. In Table 7.1 the values of the three non-Newtonian models’ rheological parameters are shown. These parameters exhibited the optimal agreement for each rheological model and they were determined after many computational experiments. The discrepancy between the results acquired by the three rheological models and the experimental findings have been calculated using equation (6.16) for $H = 0 \text{ m}$ and presented in Table 7.1. As illustrated (Figures 7.3 and 7.4 and Table 7.1) all three non-Newtonian models present reasonably good agreement with the experimental results. However, the Bingham model shows some extreme values for the wave height at $x = 885 \text{ m}$, while the power-law model under-predicts the wave run-up at the South-West slope (see Figure 7.2). On the other hand, the Herschel-Bulkley model clearly shows the best performance between the three rheological models tested (see Table 7.1) and it is used for all comparisons of the Lituya Bay tsunami test case presented herein. Figure 7.5 shows the shear-rate/shear-stress relationship of the chosen model with optimised rheological parameters $\mu = 10^{5.45} \text{ Pa \cdot s}$, $\tau_Y = 100 \text{ Pa}$ and $N = 0.375$.

For the Lituya Bay test case dummy (fixed) particles (Koshizuka et al., 1998) were
Figure 7.3: Wave run-up for three different rheological models.

Figure 7.4: Wave height at $x = 885$ m for three different rheological models.
CHAPTER 7
Lituya Bay Tsunami and Landslide

Figure 7.5: The shear-rate vs shear-stress relationship of the rock phase using a Herschel-Bulkley rheological model.

![Shear-Stress vs. Shear-Rate for the rock-slide phase](image)

<table>
<thead>
<tr>
<th>Rheological model</th>
<th>Viscosity $\mu$ [Pa·s]</th>
<th>Yield stress $\tau_Y$ [Pa]</th>
<th>Power-law index $N$</th>
<th>Wave run-up $\epsilon_{L2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bingham</td>
<td>$10^{5.2}$</td>
<td>100</td>
<td>-</td>
<td>0.19</td>
</tr>
<tr>
<td>Power-law</td>
<td>$10^{5.4}$</td>
<td>-</td>
<td>0.5</td>
<td>0.21</td>
</tr>
<tr>
<td>Herschel-Bulkley</td>
<td>$10^{5.45}$</td>
<td>100</td>
<td>0.375</td>
<td>0.08</td>
</tr>
</tbody>
</table>

Table 7.1: Rheological parameters for the three inelastic models tested for the Lituya Bay test case and their discrepancy with the findings of Fritz et al. (2001).

used for the boundaries, since they were found to give more stable results than the mirror particle approach. The initial particle spacing used was $dx = 6$ m giving a total of 9219 SPH particles. Computational time was 8 h 50 min on a Intel Core i7-4770 CPU, single-core chip with 3.4 GHz and 16GB of memory.

Similar to the submarine landslide (see Section 6.5.5), the Lituya Bay test case exhibits very high Reynolds number, well within the turbulent region, with the peak value of $Re \approx 8.6 \times 10^9$.

**Comparisons with FLOW-3D results.** Basu et al. (2010) used the commercial software FLOW-3D (Incorporated, 2006) to model the Lituya Bay tsunami event. FLOW-3D is a grid-based CFD software which uses VOF to track the free surface, while the governing equations are solved either the finite volume or finite difference
methods (Incorporated, 2006) (in this case a FVM is used). The configuration of their model was the same as the one used in the current study, initialising the rock phase with a circular segment profile (see Figure 7.2). Nevertheless, some assumptions were made for the computational model. Firstly, the inclined walls in their work (Basu et al., 2010) were modelled with free-slip boundaries, while no-slip boundary conditions are chosen for the water bed. Moreover, the material of the rock phase was modelled as a two-phase mixture of granular material and water for different void fraction values. This resulted in three different densities of the landslide material (2000 kg/m$^3$, 2600 kg/m$^3$)
and 3000 kg/m$^3$). Comparisons are made with the results given for $\rho = 2600$ kg/m$^3$, which appeared to give the best agreement with the experimental results (Basu et al., 2010). Turbulent models were also chosen for their study. In particular, $k - \epsilon$ based turbulence models, available with FLOW-3D were implemented (Basu et al., 2010).

Figure 7.6 shows the comparison between the INNSPH results with the flow parameters shown in Table 7.1 against the grid-based method of Basu et al. (2010). As illustrated the results acquired by INNSPH are significantly better than the FLOW-3D approach of (Basu et al., 2010). Both for the wave run-up and the wave height at position $x = 885$ m (see Figures 7.6a and 7.6b respectively) Basu et al. (2010) results are both misplaced in time and overpredicted in magnitude. On the other hand, INNSPH match the experimental results considerably better than the FLOW-3D results of Basu et al. (2010). As illustrated (Figure 7.6), INNSPH successfully predicts almost exactly the magnitude of the wave height and wave run-up, and manages to agree closely with the experimental results for most of the simulation time.

**Comparisons with WCSPH.** Schwaiger (Schwaiger, 2007; Schwaiger and Higman, 2007) used a WCSPH approach to simulate the Lituya Bay tsunami event. In their model the initial profile of the rock phase had a wedge-like shape unlike the circular segment profile used in other literature (Pastor et al., 2009; Basu et al., 2010) and in the current work. Moreover, the rock phase was assumed to have a Newtonian rheological behaviour, while the water phase was modelled as an inviscid fluid. No turbulence model was considered in their study. All these assumptions deviate from the realistic properties of the problem.

Figure 7.7 shows the results of the INNSPH method with the parameters shown in Table 7.1 against the WCSPH results by Schwaiger (Schwaiger, 2007; Schwaiger and Higman, 2007). Results provided by Schwaiger (Schwaiger, 2007; Schwaiger and Higman, 2007) lasted 50 s after the impact of the rock phase in the water mass. As shown the ability to implement non-Newtonian rheology with the INNSPH offers improved results compared with the WCSPH Newtonian approach (Schwaiger, 2007; Schwaiger and Higman, 2007). The improvement of the wave height at position $x = 885$ m (Figure 7.7b) is particularly promising, where INNSPH achieves a much closer agreement.
CHAPTER 7
Lituya Bay Tsunami and Landslide

(a) Wave runup

Figure 7.7: Comparison between INNSPH, the WCSPH method of Schwaiger (Schwaiger, 2007; Schwaiger and Higman, 2007) and the experimental results of Fritz et al. (2001)

(b) Wave height at \( x = 885 \) m

(c) Rock-entry thickness
with the experimental results throughout the computational simulation. Additionally, improvements are shown in the prediction of the rock-entry profile, which is measured at $x = -67$ m (Figure 7.7c) and the wave run-up (Figure 7.7a). Despite the improved results shown in Figure 7.7c by INNSPH, there are still considerable differences comparing with the experimental data (Fritz et al., 2001). This large deviation is because of the different input methods discussed previously, with Fritz et al. (2001) using a pneumatic accelerator mechanism (presented in detail in Fritz and Moser, 2003) to propel the rock-phase mass towards the water, while in the WCSPH method (Schwaiger, 2007; Schwaiger and Higman, 2007) and the current method a gravitational acceleration was acting solely on the SPH particles.

Having investigated the Lituya Bay landslide and tsunami using the real scale of the problem and a gravitational acceleration input method, which aligns with the state-of-the-art computational methods and the mechanisms of a real-life landslide, it has been found that the proposed method leads to improved results over the state-of-the-art methods (Schwaiger and Higman, 2007; Basu et al., 2010). In the following section the computational model is developed towards the full representation of the experimental findings, aiming to achieve a good agreement with the Fritz et al. (2001) experiment for both the rock-entry profile and the wave run-up height.

### 7.3 Pneumatic accelerated rock entry

In this section the experimental approach of the pneumatic accelerator as used by Fritz et al. (2001) is going to be investigated. As shown in Figure 7.7c with the gravitational acceleration method, the rock entry-profile, achieves only poor agreement with the experimental findings. In order to match the experimental profile of the rock entry as presented in the experimental results of Fritz et al. (2001), a pneumatic accelerator was computationally reconstructed based on the dimensions presented in Figure 7.8. Figure 7.9 shows the initial configuration of the reconstructed landslide accelerator based on the experimental reports (Fritz et al., 2001; Fritz and Moser, 2003). As shown the flow is accelerated on a wall with free-slip boundary conditions, since no-slip boundary conditions result in a trailing effect of the rock phase and an overall
poor agreement with the experimental data. Notably, Basu et al. (2010) used free-slip boundary conditions on the landslide wall, but did not present comparisons with experimental data.

The velocity profile of the pneumatic mechanism was approximated with a combination of two sinusoidal functions as:

\[
\frac{u_{\text{Piston}}}{U_{\text{Piston}}} = \begin{cases} 
\sin\left(\frac{\pi}{2T \times 0.75} t\right), & 0 < t \leq 0.75 \times T \\
\sin\left(\frac{\pi}{2T \times 0.25}(t - 0.5T)\right), & 0.75 \times T < t \leq T
\end{cases} \tag{7.1}
\]

where, \(u_{\text{Piston}}\) is the velocity of the pneumatic accelerator, \(U_{\text{Piston}}\) the maximum velocity of the pneumatic accelerator and \(T\) the overall period of acceleration. Figure 7.10 shows the velocity profile of the pneumatic accelerator used in the current work centred against the one presented by Fritz and Moser (2003). It has been found that matching the experimental data of the velocity profile as well as the reported average impact velocity of 110 m/s (Fritz et al., 2001) was essential to approximate the rock-entry profile.

Similar to the gravitational approach of Section 7.2 computational experiments initially including only the rock phase were conducted in order to determine the rheological parameters that match more closely the experimental findings of Fritz et al. (2001). Figure 7.11 presents the outcome of this parameter fitting procedure for a Newtonian, a Bingham, a power-law and a Herschel-Bulkley fluid. Table 7.2 also presents
the rheological parameters for each of the four cases of Figure 7.11 as well as the discrepancy with the experimental findings (Fritz et al., 2001) calculated using equation (6.16) for $H = 0$ m and $t < 4$ s. As shown each of the three non-Newtonian models performs significantly better than the fluid with the Newtonian rheology, showing that an effective decrease of the viscosity over increasing shear-rate (apparent in both visco-plastic and shear-thinning models), better represents the actual flow of the granular material comparing to a constant viscosity (Newtonian approach). Between the three different non-Newtonian models, the Bingham model shows, in this case, the closest agreement with the experimental results for the rock entry, managing to capture both the initial stage of the rock-phase entry as well as the evolution, and the duration of the rock entry. Hershel-Bulkley and power-law models, also manage an acceptable agreement, failing though to capture the peak of the entry profile, at the latter parts
CHAPTER 7
Lituya Bay Tsunami and Landslide

(a) Newtonian  
(b) Bingham  
(c) Power-law  
(d) Herschel-Bulkley

Figure 7.11: Comparison between INNSPH and experimental results of Fritz et al. (2001) for the rock-phase entry profile versus time at $x = -67$ m.

<table>
<thead>
<tr>
<th>Rheological model</th>
<th>Viscosity $\mu$ [Pa·s]</th>
<th>Yield stress $\tau_Y$ [Pa]</th>
<th>Power-law index $N$</th>
<th>Rock-entry $\epsilon_{L2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Newtonian</td>
<td>$10^{5.6}$</td>
<td>-</td>
<td>-</td>
<td>0.15</td>
</tr>
<tr>
<td>Bingham</td>
<td>$10^{5.6}$</td>
<td>1000</td>
<td>-</td>
<td>0.07</td>
</tr>
<tr>
<td>Power-law</td>
<td>$10^{5.2}$</td>
<td>-</td>
<td>0.375</td>
<td>0.10</td>
</tr>
<tr>
<td>Herschel-Bulkley</td>
<td>$10^{5.2}$</td>
<td>100</td>
<td>0.375</td>
<td>0.11</td>
</tr>
</tbody>
</table>

Table 7.2: Rheological parameters for the three inelastic models tested, with density $\rho = 1650$ kg/m$^3$ and the discrepancy with the findings of Fritz et al. (2001).
Having analysed the single-phase evolution of the flow for the rock phase, the water phase has been reintroduced. Unfortunately though, this time the wave run-up was massively overpredicted, with an estimated wave run-up height of $H = 1450$ m (Figure 7.12), which is approximately three times higher than the findings of the experiment. At this stage in the modelling, this significant deviation from the experimental results was postulated to be caused by one or more of the following factors:

- Only two of the three phases present (i.e. granular material, water and air) are modelled. The exclusion of the air-phase might be an important factor. As discussed by Lind et al. (2015) a compressible air-phase is an important factor to include in slamming processes.

- Diffusion or saturation of the rock phase is not considered. During the slamming processes water enters into the voids between the grains of the material, causing a change into its apparent density and viscosity. Mitarai and Nakanishi (2012) went to the extent to argue that the apparent viscosity difference between a dry and a wet granular material might be such that the rheological behaviour can exhibit even shear-thickening behaviour for wet granular materials. This dramatic change compared to the shear-thinning behaviour of the dry sediment can alter drastically the characteristics of the flow.

- Currently only a two-dimensional analysis has been performed. 3-D phenomena
might be taking place which are not taken into account.

- Although a Reynolds number well within the turbulent region of the flow has been calculated \((Re \approx 8.6 \times 10^9)\), no turbulent model has been considered up to this stage. In a turbulent flow an extra viscosity term, known as eddy viscosity, is included in the flow field. With the introduction of a fluid turbulence model, this extra viscosity term might play an important role in the propagation of the flow.

Clearly, all the aforementioned factors should be present in a realistic model, nevertheless, some of these omissions are more significant than others for this specific application. Specifically, it can be seen that the experimental configuration (Fritz et al., 2001) can be adequately reproduced using a two-dimensional analysis, since the experiment was built to represent a cross section of the Lituya Bay fjord, with parallel tank walls and narrow tank width. Moreover, the omission of the air phase should not be that significant, since it has been shown that the air-phase plays a significant role when the impact zone is large enough for air particles to be entrapped between the impacting bodies (Lind et al., 2015). In our case the landslide enters the water in a wedge like shape, which would reduce the air-phase effect on the impact. On the other hand, the turbulence and diffusion/saturation models may play a much more significant role, because of the effect they have on the viscosity of the two fluids as discussed in the previous paragraphs.

To address these issues, first, in the following section the \(k - \epsilon\) turbulence model as described by Leroy (Leroy et al., 2014; Leroy, 2015) is introduced to the INNSPH method, aiming to improve the results of the Lituya Bay test-case with a more realistic representation of the flow.

### 7.4 Turbulence modelling

The Navier-Stokes equations which are also used in the current study (see equations 2.12 and 2.13) are adequate to simulate turbulent flows. A turbulent flow simulation using purely the Navier-Stokes simulation is called the Direct Numerical
Simulation (DNS) approach. Nevertheless, turbulent phenomena are associated with small-scale structures. Thus, to simulate accurately a turbulent flow using DNS, prohibitively fine 3-D space discretization needs to be adopted (Chen and Jaw, 1998). The computational cost is thus hugely significant and imposes a limitation of the applicability of the DNS approach to small reference cases and problems not relevant to industry (e.g. Kawamura et al., 2000; Abe et al., 2001). To overcome the limitations of the DNS approach, turbulence models have been introduced which model the flow field of a turbulent flow with approximation of the eddies, either with an averaged flow field (Reynolds averaged Navier-Stokes (RANS) approach) or by representing the turbulent eddies up to a lower scale limit (Large Eddy Simulation (LES)). For the latter significant work has been done in SPH by Mayrhofer and co-workers (Mayrhofer, 2014; Mayrhofer et al., 2014, 2015).

In this work, the RANS approach (Pope, 2001) is chosen, since it is the most computationally inexpensive and offers a good representation of the flow field. The RANS approach is based on the observation that although a turbulent flow field has an inherently chaotic and highly-variable nature, its mean is found to be smooth and less variable (Chen and Jaw, 1998). Based on this idea a statistical mean operator $\overline{A}$ of any variable $A$ is introduced to the governing equations. The relationship between $A$ and $\overline{A}$ is expressed by:

$$A = \overline{A} + A', \quad (7.2)$$

where $A'$ represents the fluctuations of the field variable $A$. Applying this idea to the conservation of mass and conservation of momentum (equations 2.12 and 2.13 respectively) one gets:

$$\nabla \cdot \overline{u} = 0 , \quad (7.3)$$

and

$$\frac{d\overline{\Pi}}{dt} = -\frac{1}{\rho} \nabla \overline{p} + \frac{1}{\rho} \nabla \cdot \overline{\tau} + \overline{F} - \nabla \cdot \overline{R} , \quad (7.4)$$

as the Reynolds-averaged conservation of mass equation and the Reynolds averaged conservation of momentum respectively. In equation (7.4) the Reynolds stress tensor
is now appearing which is defined as:

$$R = \mathbf{u}' \otimes \mathbf{u}' . \quad (7.5)$$

This stress tensor is usually approximated with the Boussinesq model (Pope, 2001) as:

$$R = \frac{2}{3}kI - 2\nu_T S , \quad (7.6)$$

where $k$ is the turbulence kinetic energy expressed as $k = tr R/2 = |\mathbf{u}'|^2/2$, $\nu_T$ is the eddy viscosity and $S = (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)/2$ the mean strain rate tensor. The RANS equations are now written as:

$$\nabla \cdot \mathbf{u} = 0 , \quad (7.7)$$

and

$$\frac{d\mathbf{u}}{dt} = -\frac{1}{\rho} \nabla \tilde{p} + \frac{1}{\rho} \nabla \cdot \tilde{\tau} + \mathbf{F} , \quad (7.8)$$

where $\tilde{p} = \bar{p} + 2/3\rho k$ and for Newtonian flows $\tilde{\tau} = 2\mu_E S$ with $\mu_E = \mu + \mu_T$ and $\mu_T = \rho \nu_T$. In order to approximate the turbulent kinetic energy and the eddy viscosity the $k-\epsilon$ turbulence model (Launder and Spalding, 1972) is introduced. This turbulence model has been widely used in the context of SPH in the past (Violeau, 2004; Violeau and Issa, 2007; Violeau, 2012; Ferrand et al., 2013; Leroy et al., 2014; Leroy, 2015). The evolution in time of the turbulence kinetic energy $k$ and the turbulent dissipation $\epsilon$ reads as

$$\frac{dk}{dt} = P - \epsilon + \frac{1}{\rho} \nabla \cdot \left( \left( \mu + \frac{\mu_T}{\sigma_k} \right) \nabla k \right) \quad (7.9)$$

and

$$\frac{d\epsilon}{dt} = \frac{\epsilon}{k} (C_{\epsilon,1} P - C_{\epsilon,2} \epsilon) + \frac{1}{\rho} \nabla \cdot \left( \left( \mu + \frac{\mu_T}{\sigma_\epsilon} \right) \nabla \epsilon \right) \quad (7.10)$$

respectively, while in the SPH formalism equations (7.9) and (7.10) are written for a particle $i$ as:

$$\frac{dk_i}{dt} = P_i - \epsilon_i + \sum_j \frac{m_j}{\rho_j} \left( \mu_i + \mu_j + \frac{\mu_{T,i} + \mu_{T,j}}{\sigma_k} \right) \frac{k_{ij} \mathbf{r}_{ij}}{r_{ij}^2 + \eta^2} \cdot \nabla W_{ij} \quad (7.11)$$
CHAPTER 7

Lituya Bay Tsunami and Landslide

\[
\begin{array}{|c|c|c|c|c|c|}
\hline
C_\mu & C_{\epsilon,1} & C_{\epsilon,2} & \sigma_k & \sigma_\epsilon & \kappa \\
\hline
0.09 & 1.44 & 1.92 & 1.0 & 1.3 & 0.41 \\
\hline
\end{array}
\]

Table 7.3: \(k-\epsilon\) model constants

and

\[
d\epsilon_i \over dt = \frac{\epsilon_i}{k_i} (C_{\epsilon,1} P_i - C_{\epsilon,2} \epsilon_i) + \sum_j \frac{m_j}{\rho_j} \left( \mu_i + \mu_j + \frac{\mu_{T,i} + \mu_{T,j}}{\sigma_\epsilon} \right) \frac{\epsilon_{ij} r_{ij}}{r_{ij}^2 + \eta^2} \cdot \nabla W_{ij} \tag{7.12}
\]

with \(P_i\) being a production term equal to:

\[
P_i = \min \left( \sqrt{C_{\mu}} S_i, \frac{k_i}{\epsilon_i} \right) \left( k_i S_i \right), \tag{7.13}
\]

where \(S_i = \sqrt{2S.S}\). Moreover the eddy viscosity \(\mu_T\) is related to the turbulent kinetic energy and dissipation as:

\[
\mu_{T,i} = C_\mu \rho_i \frac{k_i^2}{\epsilon_i}. \tag{7.14}
\]

Terms \(C_\mu, C_{\epsilon,1}, C_{\epsilon,2}, \sigma_k\) and \(\sigma_\epsilon\) are constants specific to the \(k-\epsilon\) model, taking the values presented in Table 7.3 (Violeau, 2004; Leroy, 2015).

7.4.1 Boundary conditions

The treatment of boundaries is very important in a turbulent flow, since the shearing effects near the walls affect the evolution of the turbulent flow. To reproduce accurately the flow field near the wall a logarithmic law is used for the velocity of the boundary particles (Lin and Liu, 1998; Violeau and Issa, 2007) which describes the velocity near the wall as:

\[
\begin{cases}
|u_i| = y_i^+ & \text{if } y_i^+ \leq y_{\text{lim}}^+ \\
\frac{|u_i|}{u_\ast} = \frac{1}{\kappa} \ln \left( \frac{\delta u_\ast}{\nu_i} \right) + 5.2 & \text{if } y_i^+ > y_{\text{lim}}^+
\end{cases}
\tag{7.15}
\]

where \(|u|\) is the tangential to the wall velocity, \(y_{\text{lim}}^+ = 1/\kappa\) with \(\kappa\) being the Von Karman constant (shown in Table 7.3), \(\delta = \max(|r_{i,\text{wall}}|, dx)\) (Leroy, 2015) and

\[
y_i^+ = \frac{\delta \sqrt{C_\mu k_i}}{\nu_i}. \tag{7.16}
\]
In turbulent flows usually the viscous sub-layer has a very small size, much smaller than the particle resolution that can be realistically used in the current study. Thus, the wall particles are modelled to represent the end of the viscous sub-layer rather than the actual wall. To achieve this the tangential-to-the-wall velocity component of the wall boundary particles is updated at each timestep according to the viscous term of the momentum equation (7.8). This treatment is applied only to the first inner layer of the wall boundary particles (the ones adjacent to the fluid domain), while for the outer layers of the boundary particles the velocity is defined by the log-law of equation (7.15). As explained later in this section dummy particles (Koshizuka et al., 1998; Violeau and Issa, 2007) are used for the boundaries of the turbulent flows presented herein. Thus, their positions remain fixed despite the assignment of slip-like velocities. Similar approaches of introducing a slip-like velocity to the wall have been used previously by state-of-the-art SPH studies (Violeau and Issa, 2007; Ferrand et al., 2013; Leroy, 2015).

Boundary conditions for the turbulence kinetic energy $k$ and dissipation $\epsilon$ also need to be considered. Similar with Leroy et al. (2014) the following boundary conditions of $k$ and $\epsilon$ are applied near the wall:

$$\begin{align*}
\frac{\partial k}{\partial n} &= 0 \\
\frac{\partial \epsilon}{\partial n} &= -\frac{u_{*}^3}{\kappa \delta^2}
\end{align*}$$

(7.17)

while for the free surfaces the following is commonly used

$$\begin{align*}
\frac{\partial k}{\partial n} &= 0 \\
\frac{\partial \epsilon}{\partial n} &= 0
\end{align*}$$

(7.18)

Similar to Violeau and Issa (2007), in the current work dummy boundary particles (Koshizuka et al., 1998) are adjusted accordingly for a turbulent flow and used as SPH boundary particles. In this study a new interaction between fluid particles $i$ and boundary particles $j$ is proposed. Unlike previous works (Violeau and Issa, 2007), the dummy boundary particles are not acquiring a single value at each timestep for their
velocity $u$, turbulent kinetic energy $k$ and dissipation $\epsilon$ which would then be used for the SPH timestep summations. In this work wall parameters $u_j$, $k_j$ and $\epsilon_j$ are determined directly (on-the-fly) for each fluid particle $i$ according to equations (7.15) and (7.17), resulting in unique $u_j$, $k_j$ and $\epsilon_j$ values for each $ij$ relationship. This treatment is imposed to all boundary particles with the exception of the velocity interaction of a fluid particle with a boundary particle belonging to the inner first layer of the boundary particles where a slip-like velocity is imposed (as mentioned above). In this case the boundary particle retains its unique velocity determined by the viscous acceleration.

Finally, the turbulent kinetic energy $k$ and dissipation $\epsilon$ need to be initialised at the beginning of each simulation. In the current work $k$ and $\epsilon$ are initialised as:

$$k = (0.002U)^2$$

and

$$\epsilon = 0.16 \frac{\sqrt{k^3}}{L_m}$$

respectively, with $U$ being the characteristic velocity of the flow and the mixing length $L_m$ is described as $L_m = \max(2d_x, 10^{-5})$ (Leroy, 2015).

Although the $k-\epsilon$ turbulent model as presented in the current section has been applied and validated in the context of SPH before (Violeau and Issa, 2007; Violeau, 2012; Ferrand et al., 2013) and specifically in the ISPH with shifting method (Leroy et al., 2014; Leroy, 2015), in the current work a different boundary approach is used. Therefore, in the next subsections some benchmark validation test-cases are first considered, before applying the proposed turbulent model to the case of the Lituya Bay landslide using the reconstructed pneumatic accelerator input.

### 7.4.2 Validation cases for the $k-\epsilon$ model

In this section the $k-\epsilon$ model as described in Section 7.4 will be validated. The standard validation cases of a turbulent channel flow (Kawamura et al., 2000; Abe et al., 2001) and a schematic fish-pass (Tarrade et al., 2008; Violeau, 2012; Ferrand et al., 2013; Leroy et al., 2014; Leroy, 2015) are considered and validated against the
proposed methodology. Comparisons are held against DNS (Kawamura et al., 2000; Abe et al., 2001), experimental (Tarrade et al., 2008) and computational (Violeau, 2012; Ferrand et al., 2013; Leroy et al., 2014; Leroy, 2015) results. Moreover extension to free-surface flows has been undertaken and comparison against the WCSPH method of Violeau and Issa (2007) is presented for a dam-break case. As presented in the following subsections, the proposed methodology produces acceptable results, allowing the inclusion of the method to the Lituya Bay test case with a pneumatic accelerator input.

7.4.2.1 Turbulent Poiseuille flow

In order to assess the performance of the $k - \epsilon$ turbulent model, the test-case of a turbulent plane Poiseuille flow is first considered. The geometry of the model is the same as the one presented in the laminar test-case of Section 5.2 and Figure 5.1. In the current test-case the Reynolds number at the center of the channel is set to $Re = 640$ to align with the previously published reference work (Kawamura et al., 2000; Abe et al., 2001; Leroy et al., 2014; Leroy, 2015). The characteristic length of the channel in this case is chosen as $L = 1\text{ m}$, while a body force of $F = 1\text{ m/s}^2$ is assigned resulting to a friction velocity of $u_* = 1\text{ m/s}$. By substituting the friction velocity $u_*$ and the characteristic length $L$ to the Reynolds number equation (2.25) one gets that the kinematic viscosity of the fluid is equal with $\nu = 1.5625 \times 10^{-3}\text{ m}^2/\text{s}$.

Figure 7.13 presents the comparison of the current work with the $k - \epsilon$ model implemented against the DNS results of Kawamura et al. (2000) and the ISPH results of Leroy et al. (2014) for a particle spacing $dx = 0.05\text{ m}$. The comparisons are presented for the dimensionless distance from the wall

$$y^+ = \frac{yu_*}{\nu}$$

and the dimensionless values of the velocity component $u$, turbulent kinetic energy $k$ and turbulent dissipation $\epsilon$ expressed as:

$$u^+ = \frac{u}{u_*}, k^+ = \frac{k}{u_*^2} \text{ and } \epsilon^+ = \frac{\epsilon L}{u_*^3},$$

(7.22)
Figure 7.13: Comparison of the turbulent Poiseuille flow with $Re = 640$, between the DNS results of Kawamura et al. (2000), the ISPH results of Leroy et al. (2014) and the current work.
respectively. As shown the ISPH results of the current study are in agreement with both the DNS results (Kawamura et al., 2000; Abe et al., 2001) and the ISPH results with the unified semi-analytical boundary conditions (Leroy et al., 2014; Leroy, 2015). Specifically, for the dimensionless velocity $u^+$ and turbulent dissipation $\epsilon^+$ (Figures 7.13a and 7.13c) the ISPH method of the current study achieves close agreement with the DNS results (Kawamura et al., 2000), although they are found to marginally over-predict comparing with the ISPH method of Leroy et al. (2014). On the other hand, for the dimensionless turbulent kinetic energy $k^+$ (Figure 7.13b) the proposed method is found to give improved results over the ISPH with semi-analytical boundaries method (Leroy et al., 2014), especially near the boundary wall.

Clearly, the boundary condition treatment is very important in the accurate representation of a turbulent flow and since the unified semi-analytical boundary conditions (Ferrand et al., 2013) is in principal more accurate than any other SPH boundary condition, an overall better accuracy is shown in the results of Leroy et al. (2014) compared with the current study, with the single exception of the turbulence kinetic energy near the wall. Nevertheless, it is also easily seen that the proposed methodology produces results with satisfactory accuracy showing the ability of the method to adequately describe a turbulent flow field.

### 7.4.2.2 Schematic fish-pass

For further validation of the $k - \epsilon$ model used in this study, the more complex and more realistic case of the schematic fish-pass is considered. This test-case has been widely used both in WCSPH (Violeau, 2012; Ferrand et al., 2013) and ISPH (Leroy et al., 2014; Leroy, 2015), following the experimental work of Tarrade et al. (2008) (amongst others). In the experiment a series of identical passes were aligned one next to the other, resembling a periodic flow. These series of fish-passes were then placed on a slope, while the volumetric flow was controlled. Although a number of different geometries were tested in the experiment, in the current study as well as in the previous SPH simulations the geometry presented in Violeau (2012) was used. For this specific geometry the experimental fish-passes were positioned on an inclined bed with a 10% gradient (Tarrade et al., 2008).
Comparison with experimental results  To validate the results of the fish-pass test-case, initial comparisons against the experiment of Tarrade et al. (2008) are considered for height of the channel equal with 2 m. Periodic boundary conditions are used in the x direction to simulate the multiple fish-passes used in the experiment. The flow is then accelerated by a gravitational force equal with $F_x = 0.976 \text{ m/s}^2$ which corresponds to a slope with 10% gradient. It is found that at the steady state the volumetric flow through the slot is approximately 770 L/s which is in close comparison with the 736 L/s proposed in the experiment (Tarrade et al., 2008). The working medium is set as water with viscosity $\mu = 0.001 \text{ Pa} \cdot \text{s}$ and density $\rho = 1000 \text{ kg/m}^3$, while an initial particle spacing of $dx = 0.02 \text{ m}$ was chosen. Results are presented after 20 s of simulation, when a steady state has been reached.

Figure 7.14 presents the comparison of the current work against the experimental results of Tarrade et al. (2008) for similar geometries. As shown the proposed method simulates with acceptable accuracy the findings of the experimental work of Tarrade et al. (2008). Specifically, the flow profile matches the main stream of the flow as well as the recirculation area marked by Tarrade et al. (2008) to an acceptable level of accuracy. More differences start to appear near the left-hand side of the top obstacle, where a more profound recirculation is predicted by Tarrade et al. (2008), which is under-predicted in the results of the proposed methodology. Nevertheless, the overall prediction of the flow profile by the proposed methodology shows some very satisfactory results.

Comparison with previous SPH results  In the works of Ferrand et al. (2013) and Leroy et al. (2014) the fish-pass test-case was considered as a validation technique and compared with the results acquired by a FVM (Archambeau et al., 2004). In these test cases periodic boundaries were used in the x direction and the two-dimensional flow was accelerated by a constant acceleration force $F_x = 1.885 \text{ m/s}^2$. Moreover, a particle spacing of $dx = 0.01 \text{ m}$ was adopted. Water with viscosity $\mu = 0.001 \text{ Pa} \cdot \text{s}$ and density $\rho = 1000 \text{ kg/m}^3$ was used, while the characteristic velocity is taken equal to 1 m/s, measured at the slot of width 0.3 m. It should be noted here that the acceleration parameter used by Ferrand et al. (2013) and Leroy et al. (2014) is not relevant to the
Figure 7.14: Comparison of the velocity contours for the schematic fish-pass test-case against the experimental results of Tarrade et al. (2008) after time $t = 20$ s.

Figure 7.15: Comparison of the velocity contours for the schematic fish-pass test-case against the ISPH method of Leroy et al. (2014) for time $t = 20$ s.

Figure 7.16: Comparison of the velocity contours for the schematic fish-pass test-case against the WCSPH method of Ferrand et al. (2013) for time $t = 20$ s.
experiment of Tarrade et al. (2008). Thus, the simulations presented in this section are not meant to reproduce any experimental findings.

Figures 7.15 and 7.16 present the steady-state comparison of the current work against the ISPH results of Leroy et al. (2014) and the WCSPH results of Ferrand et al. (2013) respectively, both of which use the unified semi-analytical boundary method. As shown the results produced by the current work and the unified semi-analytical boundary SPH methods are broadly similar but have some noticeable discrepancies. Although the total velocity magnitudes are very close to the results of Leroy et al. (2014) and Ferrand et al. (2013), the velocity contours start to show some differences, with the ISPH method of the current study showing a more off-centre stream compared to the other SPH methods. For these differences the treatment of the boundaries has to play a significant role. When comparing with the work of Leroy et al. (2014) in particular, the main methodological difference lays in the treatment of the boundary conditions, with dummy particles being used by the current study and unified semi-analytical boundaries being used by Leroy et al. (2014). In fact the main differences in terms of velocity magnitude against the ISPH method of Leroy et al. (2014), are observed on the walls of the fish-pass slot.

It should be noted here, that the test-case of the schematic fish-pass is considerably demanding in terms of boundary complexity and therefore a more sophisticated boundary treatment should give the better results. Additionally, as commented in Section 7.4.2.1 and presented in detail by Leroy et al. (2014) the SPH unified semi-analytical boundary method manages to capture exactly the results acquired by the FVM (Archambeau et al., 2004), although discrepancies were shown with the DNS results (Kawamura et al., 2000; Abe et al., 2001). Therefore, a better agreement against the FVM reference results, should be expected for the fish-pass test-case as well.

7.4.2.3 Turbulent dam-break

In this section the dam-break case as presented in the WCSPH work of Violeau and Issa (2007) based on the experiment of Koshizuka and Oka (1996) is simulated. The configuration of this dam-break simulation is the same as the one shown in Figure 4.5, only this time \( a \) is chosen as \( a = 1 \text{ m} \). The simulated fluid is water with density
\( \rho = 1000 \text{ kg/m}^3 \) and viscosity \( \mu = 0.001 \text{ Pa} \cdot \text{s} \), which is discretized with an initial particle spacing of \( dx = 0.01 \text{ m} \) corresponding to 20000 fluid particles.

Figure 7.17 shows the comparison between the ISPH method of the current work and the WCSPH method of Violeau and Issa (2007). Both methods use the same turbulence model, with different treatment of the dummy-boundary particles (discussed in detail in Section 7.4.1), and it can be seen that the results acquired by the two methodologies are very close in comparison. It should be noted here that the WCSPH velocity results have been interpolated on a fixed Eulerian grid as a post-processing treatment, thus the velocity profiles of the WCSPH method seem much smoother than the ISPH results. Nevertheless, the flow profile of the two methods bear close similarities. Moreover, in Figure 7.18 the propagation of the leading edge of the flow and the drop of the column height are shown, both of which indicate a close agreement with the experimental data of Koshizuka and Oka (1996), while the results of ISPH and WCSPH (Violeau and Issa, 2007) match exactly.

In the current section the \( k - \epsilon \) turbulence implementation has been presented and validated both for internal (DNS turbulent Poiseuille flow, schematic fish-pass of Sections 7.4.2.1 and 7.4.2.2 respectively) and free-surface flows (dam-break case in Section 7.4.2.3). All cases have shown a satisfactory agreement with previous results in the literature for both simple and complex geometries. Thus, the turbulent method presented herein is considered sufficiently validated for implementation in the Lituya Bay test-case with the pneumatic accelerator input as described in Section 7.3.

### 7.5 The Lituya Bay experiment with turbulence implementation

As presented in the beginning of the current chapter, the Lituya Bay test case is a challenging Newtonian/ non-Newtonian multi-phase environmental application, which the state-of-the-art computational methods have failed to capture accurately. In this section the turbulence \( k - \epsilon \) model is implemented in the Lituya Bay test-case and its influence is investigated.
Figure 7.17: Comparison of the velocity contours with the WCSPH dam-break case of Violeau and Issa (2007).
In order for the $k-\epsilon$ turbulent model to be implemented in the context of the Lituya Bay landslide and tsunami certain modifications need to be considered. Firstly, the size of the computational domain constrains the application of the $k-\epsilon$ model. This arises from the restriction imposed by the $k-\epsilon$ model which requires the computational particles closest to the wall to be located in the inertial layer (Leroy, 2015) located at:

$$30 < y^+ < 0.2 \frac{Lu_\ast}{\nu},$$

where $L$ is the half height. With the initial particle spacing of 6 m this requirement was not met and the simulation was failing. Instead of reducing the particle spacing, which would probably make the simulations computationally unaffordable, a rescale of the computational domain was opted. Therefore, the computational domain was scaled down to the experimental dimensions, which correspond to a scale of 1 : 675 of the actual Lituya Bay measurements. This scale allows a much more feasible particle spacing for the implementation of the $k-\epsilon$ model. Moreover, given that the aim is to reproduce the experimental data, it is reasonable to work on the experimental scale. It should be noted that for consistency with the experiment (Fritz et al., 2001) and
Having downscaled the size of the domain to the experimental scale, a parametric analysis is again performed in order to determine the non-Newtonian characteristics of the sediment. Similar to Section 7.3 a parametric analysis is performed between the Bingham, the power-law and the Herschel-Bulkley models. Figure 7.19 shows the comparison between the three non-Newtonian models, which are presented on the Lituya Bay scale. As shown in this new set of images, all three non-Newtonian models perform well, showing small differences. Namely, the Bingham model is shown to capture the initial and final stages of the landslide, while under-predicting the maximum height of the sediment entry profile. On the other hand, the power-law
Table 7.4: Rheological parameters for the three inelastic models tested for the Lituya Bay test case at an experimental scale and the discrepancy with the findings of Fritz et al. (2001).

<table>
<thead>
<tr>
<th>Rheological model</th>
<th>Viscosity $\mu$ [Pa · s]</th>
<th>Yield stress $\tau_Y$ [Pa]</th>
<th>Power-law index $N$</th>
<th>Rock-entry $\epsilon_{L2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bingham</td>
<td>1</td>
<td>500</td>
<td>-</td>
<td>0.16</td>
</tr>
<tr>
<td>Power-law</td>
<td>30</td>
<td>-</td>
<td>0.375</td>
<td>0.18</td>
</tr>
<tr>
<td>Herschel-Bulkley</td>
<td>3</td>
<td>100</td>
<td>0.375</td>
<td>0.12</td>
</tr>
</tbody>
</table>

and Herschel-Bulkley rheological models show a better representation of the height compared with the Bingham model. As before (see Section 7.2), the Herschel-Bulkley rheological model is chosen for the landslide application showing an all around good representation of the experimental findings. Table 7.4 shows the different values of the rheological parameters for the three models as well as the discrepancy with the experimental findings (Fritz et al., 2001) calculated using equation (6.16) for $H = 0$ m and $t < 4$ s.

It should be noted that the collapsing sediment phase is modelled as a laminar flow, having an approximate Reynolds number of $Re \approx 60$, which resides well within the laminar range. Therefore, a laminar model as explained in Chapter 4 is implemented. The $k - \epsilon$ model is applied then solely to the water-phase.

**Results**

Having applied the $k - \epsilon$ model in the water phase and using the Herschel-Bulkley model for the landslide (shown in Figure 7.19c) the Lituya Bay experiment of Fritz et al. (2001) with the pneumatic accelerator input is again reconstructed. As shown in Figure 7.20 the wave run-up is improved comparing with the previous laminar model, but only marginally. Specifically, the wave run-up is now measured to be 1150 m, showing a 20% improvement from the results presented in Section 7.3, but still being more than two times the 523 m reported in the experiment of Fritz et al. (2001).

Clearly, although a turbulence model has been introduced, making the proposed model more realistic, there are still certain modelling assumptions adopted such as the omission of the air-phase, or the disregard of the landslide saturation, amongst others. Of these two, landslide saturation was the physical effect deemed most likely
Due to the limited time of this Ph.D. project, a simple saturation model was chosen as a following step to investigate and presented in the following section.

### 7.6 Saturation of the landslide

Saturation is the procedure during which the voids in a porous material are filled with a liquid, typically water (Mitchell and Soga, 2005). Such phenomena are crucial in many environmental and industrial applications where a sedimentary material is saturated due to its interaction with water, influencing its flow characteristics (Mitchell and Soga, 2005; Mitarai and Nakanishi, 2012). The degree of saturation is defined as the volume of water over the volume of voids (Dyran, 1997).

With their importance in environmental and industrial sciences (i.e. geotechnology, or nuclear engineering) SPH methods with saturation models have been proposed in the past. Bui and co-workers (Bui and Fukagawa, 2013; Bui et al., 2008, 2007) have presented extensive work on saturated soils, incorporating different yield-stress criteria like the Mohr-Coulomb and the Drucker-Prager. A variety of different saturation conditions have been examined, varying from dry to fully saturated soils. The pore water pressure and seepage force are key parameters with which the interaction with
water has been modelled. Ulrich et al. (2013) and Fourtakas and co-workers (Fourtakas, 2014; Fourtakas et al., 2014) have also examined complex applications with saturated soils including scouring effects, again incorporating the Mohr-Coulomb and the Drucker-Prager criteria coupled with soil suspension models.

Evidently, there are several saturation models that can be considered for the case of the Lituya Bay landslide and tsunami. Nevertheless, a novel empirical solution was adopted with the introduction of a simple saturation model, described below.

The saturation of the sediment in this work is modelled based on the assumption of a transition time period, which begins when each sediment particle reaches the water level and during which the sediment transitions from dry to fully saturated. The parameters that change over this time period is the yield stress $\tau_Y$, the viscosity $\mu$ and power-law exponent $N$ of the Herschel-Bulkley model. Moreover, the density of the sediment $\rho$ is changing instantly when a sediment particle reaches the water level as well as the interaction of the sediment with the boundary particles which transitions from free-slip to no-slip (modelling the transition from rolling dry granular particles at the boundary to fully saturated sediment). It should be noted here that the density and boundary condition transition were found to have a better performance with an instant transition rather than following the transition time period mentioned before.

Table 7.5 shows the dry and saturated values which were found to perform better for the case of the Lituya Bay landslide. The change of the density corresponds to a 39% void-fraction (Fritz et al., 2001), assumed fully filled with water during saturation. Moreover, the transition of the rheological parameters are based on the findings of Mitarai and Nakanishi (2012) who have reported that the rheological behaviour of a sediment shows a transition to shear-thickening behaviour during saturation. In the saturation model used herein a transition from a shear-thinning Herschel-Bulkley model to a pseudo-plastic Bingham model, with higher yield stress and viscosity was found to give the closest results to the experiment. Figure 7.21 shows the shear-rate/shear-stress (normalised with yield stress) relation of the two different rheological models corresponding to the dry and saturated sediment conditions, showing clearly the shear-thinning and pseudo-plastic rheological behaviours.

Figure 7.22 shows the comparison for the wave run-up, the wave height at $x =$
CHAPTER 7

Lituya Bay Tsunami and Landslide

Figure 7.21: The shear-rate/shear-stress over yield stress comparison of the rheological models for dry and saturated sediment.

<table>
<thead>
<tr>
<th>Flow parameter</th>
<th>Dry value</th>
<th>Saturated value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho$ [kg/m$^3$]</td>
<td>1650</td>
<td>2600</td>
</tr>
<tr>
<td>$\mu$ [Pa·s]</td>
<td>150</td>
<td>750</td>
</tr>
<tr>
<td>$\tau_Y$ [Pa]</td>
<td>100</td>
<td>1000</td>
</tr>
<tr>
<td>$N$</td>
<td>0.1</td>
<td>1.0</td>
</tr>
</tbody>
</table>

Table 7.5: Transition of sediment’s rheological parameters from dry to saturated state.

885 m and the rock-entry profile between the experimental results of Fritz et al. (2001) and the SPH results of the current work. The results presented here are simulated using the rheological parameters shown in Table 7.5 with a transition period of $T = 1.95$ s. It is clearly shown that with the proposed model a good overall agreement with the experiment of Fritz et al. (2001) is achieved, having a close agreement for the wave run-up while retaining a good representation of the sediment-entry profile. Figure 7.23 shows the comparison of the newly modified INNSPH and the experimental photographs (Fritz et al., 2001) for the wave run-up. Evidently a close agreement with experimental findings is observed for the same time intervals. Noticeably, even the breaking of the wave before the run-up is captured in the INNSPH results (see Figure 7.23ia).

From the results shown here, only the wave height at position $x = 885$ m (see Figure 7.22b) was found to under-perform comparing to initial gravity acceleration profile, described in Section 7.2 (see Figure 7.4c). This observation can possibly be
Figure 7.22: Comparison between INNSPH with saturation and the experimental results of Fritz et al. (2001)
Figure 7.23: Wave run-up at 1.73s intervals: i) The INNSPH results, ii) the experimental results of Fritz et al. (2001).

Figure 7.24: Impact of the two phases at 1.73s intervals: i) The INNSPH results, ii) the experimental results of Fritz et al. (2001).
explained when comparing the experimental and INNSPH profiles of the sediment, as shown in Figure 7.24. It is illustrated that, although in the initial stages of the impact the sediment profile predicted by the proposed method matches very closely the experimental images, at later stages discrepancies start to appear. Specifically, there is a volume of water trapped in the lower left hand corner which is normally not expected and the propagation of the sediment is slower in comparison with the experiment (Fritz et al., 2001). Moreover, the displaced water profile appears to raise vertically on a steeper angle in the computational INNSPH results compared with the experiment. This behaviour was observed with a wide variety of different simulation parameters. Suggestions on how to improve these results are made in Chapter 8.

7.7 Summary

In the current chapter the complex environmental case of the Lituya Bay landslide and tsunami has been thoroughly examined. The problem has first been examined in a real-scale with gravitational acceleration acting at the input of the landslide. This input method aligned with what one would expect to happen in an actual landslide event and the results of the wave height and run-up produced had a good agreement with the experimental data (Fritz et al., 2001). Moreover, improvements were shown over the state-of-the-art (Basu et al., 2010; Schwaiger, 2007; Schwaiger and Higman, 2007). Nevertheless, the landslide entry profile deviated significantly from the experimental data.

The focus of the research is then shifted towards a more thorough representation of the experimental findings Fritz et al. (2001). For that to be achieved the landslide pneumatic accelerator, as described in the experimental reports (Fritz et al., 2001; Fritz and Moser, 2003), was computationally reconstructed. Following a parametric analysis the rheological characteristics which best suited the landslide profile were chosen, achieving a much improved entry profile. Nevertheless, this improvement came at the expense of the wave run-up measurements which were found to be three times higher than the experimental findings.

New models were then added to the existing methodology presented in this work,
firstly with the addition of the $k - \epsilon$ turbulence model. Validations were carried out against experimental (Tarrade et al., 2008) and computational results (Ferrand et al., 2013; Kawamura et al., 2000; Leroy, 2015; Leroy et al., 2014; Violeau and Issa, 2007) and showed satisfactory agreement with the benchmark cases overall. However, when the model was implemented to the Lituya Bay case only a small improvement was observed against the laminar counterpart.

The implementation of a simple empirical saturation model was then considered. The change of the rheological behaviour due to saturation was modelled to represent a more shear-thickening behaviour, based on the report of Mitarai and Nakanishi (2012). The final results showed an improved overall agreement with the experiment, crucially managing to capture both the landslide profile prior to the water impact, as well as the wave run-up on the opposite slope. These results are wholly novel and offer improvements over the state-of-the-art methods, making the proposed methodology a good basis for the study of complex multi-phase environmental phenomena.

In the next chapter (see Chapter 8) the concluding remarks of this thesis are summarized. The key outcomes of this research project are highlighted with emphasis given to the improvements that INNSPH can bring over conventional CFD analysis. Moreover, suggestions for appropriate ways to continue the ongoing research are given and some of the many applications where the proposed methodology can be implemented are outlined.
Chapter 8

Conclusions & Future Work

8.1 Introductory comments

This thesis has presented a novel incompressible non-Newtonian and multi-phase SPH method, applicable in both environmental and industrial problems. In this chapter the concluding remarks are presented followed by suggestions for future work.

8.2 Conclusions

The conclusions of this work are presented in this section, and are divided into three parts: firstly, the modelling of inelastic non-Newtonian flows using the ISPH with shifting method; the extension to multi-phase flows; and the final application of the Lituya Bay tsunami and landslide.

8.2.1 Modelling of inelastic non-Newtonian flows using the ISPH with shifting method

Chapters 4 and 5 have presented the development and validation of the incompressible SPH method for inelastic non-Newtonian flows employing shifting, as first proposed by Xu et al. (2009) and improved by Lind et al. (2012) and later by Skillen et al. (2013) for Newtonian flows. The novel incompressible non-Newtonian SPH (INNSPH) method has been tested and validated for internal non-Newtonian flows (Poiseuille and circular...
Couette flows), where analytical solutions are readily available. The comparisons with analytical results showed that INNSPH can accurately predict the flow fields of a wide range of inelastic non-Newtonian rheological models, covering all shear-thinning, shear-thickening and pseudo-plastic rheological behaviours. The method was found to be stable up to extreme non-Newtonian model parameters ($0.05 < N < 6$, $Bn < 3.5$, for the Poiseuille flow test case described in Section 5.2). When the aforementioned limits were exceeded a steady-state solution could not be reached. Although these extreme non-Newtonian values are rarely relevant to real-life applications, improvements are suggested in Section 8.3.

Free-surface single-phase non-Newtonian flows were consequently examined. A dam-break case (Komatina and Jovanovic, 1997) of a Bingham fluid was modelled where comparisons were made with experimental results, showing that INNSPH accurately predicts the flow profile in free-surface non-Newtonian flows. More complex industrial moulding flows were also tested (Fan et al., 2010; Ren et al., 2012), where INNSPH showed significant qualitative improvement of the pressure fields compared with state-of-the-art SPH methods. Specifically for the moulding flow of Ren et al. (2012) (see Section 5.6) despite the improvement of the pressure field, slight discrepancies in the temporal evolution of the flow have been identified compared with Ren et al. (2012). A reference solution from an independent third method would be useful to identify if any improvements to the INNSPH method should be considered.

The importance of the shifting method for the applicability of the divergence-free method of Cummins and Rudman (1999) in inelastic non-Newtonian flows has also been demonstrated. It has been shown that shifting is essential for the stability of the method in industrially relevant non-Newtonian flows (such as the moulding flows tested Fan et al., 2010; Ren et al., 2012), since particle clumping and stretching is avoided and a stable solution can be reached.

The accuracy of the pressure results was also rigorously tested. Quantitative comparisons with analytical results, as well as with WCSPH and CVFEM computational results were carried out. INNSPH performed better than WCSPH in the estimation of the pressure fields, while analytical and commercial CVFEM results proved that INNSPH offers accurate predictions of the pressure field for complex non-Newtonian
SPH flows. Crucially, this is a very important flow measure which it was either poorly represented or omitted by previous non-Newtonian SPH techniques.

The single-phase incompressible non-Newtonian SPH algorithm proposed in this work is entirely novel and offers a unique representation of both the flow profile as well as the pressure field, a very important flow parameter often omitted or infrequently presented in the majority of the SPH literature. Further improvements of the method are proposed as future work in Section 8.3.

### 8.2.2 Extension to multi-phase flows

With the introduction of a colour function INNSPH was successfully expanded to solve multi-phase flow problems in low density ratios (e.g. environmental flows as presented in Chapter 6). The numerical method is limited to relatively low density ratios ($< 3$) but it can handle Newtonian/ non-Newtonian interactions.

Artificial mixing and anisotropic particle distribution on the interface were avoided by imposing a single-phase shifting restriction normal to the interface. This treatment resembles the shifting treatment of free surfaces. Moreover a novel shifting approach is proposed for multi-phase flows which incorporates the diffusion coefficient of Lind et al. (2012) as a lower threshold to the shifting method of Skillen et al. (2013). This treatment has been shown to improve particle distribution compared with the shifting of Lind et al. (2012) or Skillen et al. (2013), which were otherwise found to be stable methods.

To validate the accuracy and the effectiveness of the new incompressible multi-phase SPH technique, a selection of two-phase problems with a variety of properties have been modelled. Newtonian/ Newtonian multi-phase flows (Rayleigh-Taylor instability and gravity waves) have been tested and show that two-phase INNSPH agrees well with both analytical and computational results. Moreover, INNSPH demonstrated increased efficiency compared with other particle-based methods (Hu and Adams, 2007; Quinlan et al., 2014), since it matched their interface predictions using lower particle resolution. Limitations in the computational efficiency of the method were also recognised showing that a small particle spacing is needed to allow the simulation of
low-amplitude waves (see Section 6.5.3). Furthermore, by using serial computing only a relatively low number of particles is feasible ($< 10^5$ as presented in Section 6.5.4). Suggestions for improving the computational efficiency are made in the future work section (see Section 8.3).

Complex Newtonian/ non-Newtonian multi-phase problems have been tested, and showed good agreement against analytical and experimental results. A novel analytical solution of a Newtonian/ Bingham-fluid Poiseuille flow has been derived as well as a semi-analytical solution for power-law flows. Excellent agreement has been exhibited for these flows for a variety of non-Newtonian parameters. Moreover, a more demanding submarine landslide case has been examined, where comparisons were held against experimental and other SPH results. Evidently, the proposed methodology can match the experimental results with greater success than previous state-of-the-art ISPH and WCSPH methods (Ataie-Ashtiani and Shobeyri, 2008; Capone et al., 2010). Although an improved free-surface profile has been achieved by INNSPH compared with the previous state-of-the-art methods (Ataie-Ashtiani and Shobeyri, 2008; Capone et al., 2010) the predicted water/ landslide interface deviates from the experimental findings (similarly with the state-of-the-art). In Section 8.3 future work is proposed to improve these interface predictions.

It is clear that INNSPH offers advantages compared with previously published SPH results, both for single and multi-phase flows. Moreover, with INNSPH being a branch of ISPH with shifting (Lind et al., 2012; Xu et al., 2009) a good quality pressure prediction is offered, which in many SPH cases is poorly demonstrated (as shown in Chapter 5 and in Xenakis et al. (2015) for single-phase flows).

8.2.3 The Lituya Bay tsunami and landslide

Finally, the developed methodology was applied to the complex environmental problem of the Lituya Bay tsunami and landslide. The problem has initially been approached on the real-life length-scale, with gravity acceleration solely acting on the landslide. Comparisons against state-of-the-art methods (Schwaiger and Higman, 2007; Basu et al., 2010) have shown that the proposed methodology offers improvements in the wave
run-up and wave height when compared with the scaled up benchmark experimental results (Fritz et al., 2001).

The focus of the study was then shifted towards the thorough representation of the experimental results for which the proposed algorithm as well as the state-of-the-art methods failed to capture the rock-entry profile. The method was modified as follows:

- The pneumatic accelerator used in the experiment (Fritz et al., 2001; Fritz and Moser, 2003) was computationally reconstructed.
- A $k - \varepsilon$ turbulence model with novel treatment of the dummy particles for boundary conditions on turbulent kinetic energy $k$ and dissipation $\varepsilon$ was introduced and validated, showing satisfactory results against the state-of-the-art (Leroy et al., 2014; Violeau and Issa, 2007).
- A simple saturation model, which increases the effective viscosity of the landslide from shear-thinning to pseudo-plastic, was introduced.

The final results of the INNSPH method exhibit very good agreement with the experimental results of Fritz et al. (2001) for both the rock-entry profile, as well as the wave run-up height. To date this is the first CFD method capable of achieving this agreement. The work that has been done towards these results illustrates the detailed modelling steps that need to be taken towards an effective multi-phase algorithm applicable to complex environmental flows.

### 8.3 Proposed future work

#### 8.3.1 Suggestions for future applications

It has been shown that the proposed method can perform well in selected environmental and industrial applications. Nevertheless, INNSPH is a novel computational technique which can be used to inform a greater variety of non-Newtonian multi-phase applications. Specifically:

- The method could be readily applied in biological applications. Blood flows,
the movement of the synovial fluid could be modelled and inform bioengineering, medical or biological research.

- In the food and chemical industry moulding flows are commonly used and involve non-Newtonian liquids. The implementation of INNSPH would offer accurate prediction of the pressure fields, combined with an ability to describe complex free surfaces.

- In the nuclear industry sludge flows and water/sludge interactions is of particular interest. INNSPH can be used in the design process to optimise the sludge removal and therefore increase the efficiency of such units.

- As shown in Chapter 7, INNSPH can offer a very good representation of complex geological phenomena such as landslide and tsunami events. INNSPH can be employed to simulate such events in high-risk areas, which would allow with appropriate action to prevent or minimise possible damage.

The suggested applications presented here are not exhaustive and the presented methodology could be applied to numerous Newtonian/non-Newtonian multi-phase problems, as well as non-Newtonian/non-Newtonian interactions, covering further areas of industrial or scientific research.

### 8.3.2 Suggestions for improvement of the INNSPH method

Despite the proven applicability of the INNSPH method there are several steps that can be taken in order to further improve the efficiency and the application spectrum of the proposed methodology. In detail:

- As discussed in Section 6.4, INNSPH is currently limited to multi-phase flow with low density ratios (< 3), which was adequate for the applications considered in the context of this thesis. There are different approaches that can be followed in order to accommodate larger density ratios which would increase the spectrum of the possible applications (for example air erosion or flow of bubbles in non-Newtonian liquids). Lind et al. (2015) have proposed a coupling of the
ISPH method with shifting with a WCSPH method, to model incompressible-compressible multi-phase flows. In their method the compressible phase (solved with the WCSPH method) provided pressure boundary conditions on the interface for the incompressible phase (modelled with ISPH), which in turn was used to inform the velocity field of the compressible phase. Another approach is proposed by Hu and Adams (2009) where two prediction-correction steps are used per timestep in order to enforce a constant density of the particles. Either of the two aforementioned methods could be used to facilitate larger density ratio multi-phase flows, although the one proposed by Lind et al. (2015) would be easier to introduce in the numerical method. It should be noted that either of this approaches may increase the computational cost of the numerical algorithm, since the pressure-Poisson equation would be required to be solved multiple times per timestep, although each time for smaller parts of the computational domain (in the case of Lind et al., 2015).

Once the larger density ratios can be accommodated an air-phase could be introduced, which would allow the detailed examination of applications in which air plays a significant role, such as wave erosion on beach or land. For greater Mach numbers, a similar approach to Lind et al. (2015) could be used to take into account the compressible effects of the air phase.

- In this thesis only single-phase and two-phase flows have been considered. For multi-phase flows with more than two phases, the method can be easily amended using an approach similar with Hu and Adams (2006). Multiple phase interactions can then be modelled, which would allow the application of INNSPH to a wider range of multi-phase problems, such as water-ice-sediment interaction, landslide and tsunamis including an air-phase, or interaction between blood thrombus, blood plasma and other blood cells to study at a microscopic scale the mechanisms with which a arterial-stenosis can form and develop.

- In Chapter 7 it has been shown that the evolution of the landslide as well as the initial formation of the wave did not match exactly the experimental findings (see Figure 7.24) although the rock-entry profile and the wave run-up height
were greatly improved with the implementation of the proposed saturation model. Clearly, this saturation model is very simplistic, so a more sophisticated saturation model should be implemented in order to improve even more the INNSPH results. A model such as the ones previously used by Bui and co-workers (Bui and Fukagawa, 2013; Bui et al., 2008, 2007) in the context of SPH, could be considered for the presented methodology.

- The introduction of elastic rheology can also be considered. Visco-elastic models like Oldroyd-B or the upper convected Maxwell model (briefly discussed in Section 2.3.2) can be employed to allow the simulation of more complex visco-elastic rheological problems. INNSPH would thereafter be applicable to more complex industrially relevant non-Newtonian flows, like flows of polymer materials.

- Increasing the computational efficiency of this method would also be beneficial. Implementation of modern parallelisation techniques, like GPU programming, could make the algorithm much more efficient as shown in previous SPH literature (Mokos, 2014; Mokos et al., 2015; Crespo et al., 2009; Dominguez et al., 2013a; Szewc, 2014).

- The efficiency of the method can be further improved with the introduction of variable particle resolution which has recently been explored in the context of SPH (Vacondio et al., 2013; Barcaroloa et al., 2014; Spreng et al., 2014). As discussed in Section 5.2 for the non-Newtonian Poiseuille flows, the accuracy of the INNSPH results decreases near the wall for increasing solid zones of non-Newtonian liquids. This is because decreasing shear layers become under-resolved at fixed particle distributions. Moreover, the applicability of $k - \epsilon$ model as presented in Section 7.4 is restricted by particle resolution near the boundaries. Therefore, with the introduction of variable resolution these issues can be addressed with minimum increase of computational cost.

- The method could easily be expanded to 3-D applications. Relevant work has been recently performed by Guo et al. (2015) using the Newtonian ISPH with shifting method of Lind et al. (2012). After these contributions are considered,
more complex test cases could be examined. Real events of land erosion or submarine scouring could be reproduced and the mechanisms that lead to such phenomena could be analysed. Other landslide events, where 3D geological morphology plays a crucial role could be also simulated, with greater detail and efficiency.

It is clear that the work presented in this thesis can form the basis of further future research which will then inform important engineering and environmental problems. Evidently, with contributions like the ones presented herein, INNSPH has the potential of becoming a powerful CFD tool applicable in a broad spectrum of non-Newtonian and multi-phase applications.
References


Kopal, Z. (1947), Tables of Supersonic Flow Around Cones, Depart of Electrical Engineering, Center of Analysis, Massachusetts Intitute ot Technology.


Richardson, J. F. (1922), Weather prediction by numerical process, Cambridge U.P.


Xu, R. (2009), An Improved Incompressible Smoothed Particle Hydrodynamics Method and Its Application in Free-Surface Simulations, PhD thesis, School of Mechanical, Aerospace and Civil Engineering, University of Manchester.


Appendix A

Derivation of non-Newtonian Analytical Solutions for Single-phase Plane Poiseuille flows

In this section, the analytical solution of the steady state, two-dimensional Power-Law, Bingham and Herschel-Bulkley plane Poiseuille flow is going to be presented.

In this case we assume that the Poiseuille flow is driven by a body force $F$ and not from a pressure difference $dp/dx$. Thus, the conservation of momentum equation becomes:

$$\frac{\partial u_x}{\partial t} = \frac{\partial \tau_{yx}}{\partial y} + F_x, \quad (A.1)$$

where $u_x$ and $F_x$ the velocity and the body force on $x$ direction and $\tau_{yx}$ is the shear stress on the fluid. For a steady state problem we have $\partial u_x/\partial t = 0$. The equation for the stress term can be retrieved from Vola et al. (2004). We begin with the expression for a Herschel-Bulkley fluid, from which the expressions of Power-Law and Bingham fluid can be easily retrieved. Thus,

$$\tau = \left( \frac{\tau_y}{|D|} + \mu|D|^{N-1} \right) D, \quad (A.2)$$
where,
\[
|D| = \sqrt{\frac{1}{2} \sum_{ij} D_{ij} D_{ij}} = \sqrt{\left(\frac{\partial u}{\partial x}\right)^2 + 2\left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2}, \tag{A.3}
\]
where \(\partial u/\partial x = \partial v/\partial y = \partial v/\partial x = 0\). Thus,
\[
|D| = \frac{\partial u}{\partial y}. \tag{A.4}
\]
If we combine equations A.2 and A.4, we get:
\[
\tau_{yx} = \tau_Y + \mu \left(\frac{\partial u}{\partial y}\right)^N. \tag{A.5}
\]
Moreover, for steady state solution we have \(\partial u/\partial t = 0\). Thus, equation A.1 becomes:
\[
\frac{\partial \tau_{yx}}{\partial y} + F_x = 0 \Rightarrow \frac{\partial \tau_{yx}}{\partial y} = -F_x. \tag{A.6}
\]
After combining equations A.5 and A.6, and integrating once upon \(y\), we get:
\[
\frac{\partial u}{\partial y} = \left(\frac{1}{\mu}\right)^{1/N} (-F_x \cdot y - \tau_Y + c_1)^{1/N}. \tag{A.7}
\]
By applying the boundary condition \(\partial u/\partial y = 0\) at \(y = y_0\), where \(y_o = \tau_Y / F_x\) denotes the yield point, i.e. the point at which the material yields, the constant \(c_1\) is determined as:
\[
c_1 = F_x \cdot y_0 + \tau_Y. \tag{A.8}
\]
So, equation A.7 becomes:
\[
\frac{\partial u}{\partial y} = \left(\frac{-F_x}{\mu}\right)^{1/N} (y - y_0)^{1/N}. \tag{A.9}
\]
Integrating once more upon \(y\), we get:
\[
u(y) = \left(\frac{-F_x}{\mu}\right)^{1/N} \cdot \frac{N}{N+1} (y - y_0)^{1/N+1} + c_2, \tag{A.10}
\]
where constant $c_2$ is determined by applying the boundary condition $u = 0$ on the wall $y = L/2$, where $L$ the height of the channel. Thus we get:

$$c_2 = -\left(\frac{-F_x}{\mu}\right)^{1/N} \cdot \frac{N}{N+1} \left(\frac{L}{2} - y_0\right)^{1/(N+1)}, \quad (A.11)$$

and finally:

$$u(y) = \left(\frac{2(\nu-1)/2}{\mu}\right)^{1/N} \cdot \frac{N}{N+1} \left\{ \begin{array}{ll}
\left(\frac{L}{2} - y_0\right)^{1/(N+1)}, & 0 \leq y \leq y_0 \\
\left(\frac{L}{2} - y_0\right)^{1/(N+1)} - (y - y_0)^{1/(N+1)}, & y_0 < y \leq \frac{L}{2} 
\end{array} \right. \quad (A.12)$$

The flow rate $Q$ for a channel of width $W$ is given by:

$$Q = W \int_{-L/2}^{L/2} u(y) \, dy, \quad (A.13)$$

which as a symmetrical flow becomes:

$$Q = 2W \int_0^{L/2} u(y) \, dy = 2W \left[ \int_0^{y_0} u(y) \, dy + \int_{y_0}^{L/2} u(y) \, dy \right]. \quad (A.14)$$

Combining equations $A.12$ and $A.14$ one gets the final expression of flow rate:

$$Q = 2W \left(\frac{2(\nu-1)/2}{\mu}\right)^{1/N} \cdot \frac{N}{N+1} \left[ \left(\frac{L}{2} - y_0\right)^{1/(N+1)} - \frac{N}{1+2N} \left(\frac{L}{2} - y_0\right)^{1/(N+2)} \right]. \quad (A.15)$$

Having calculated the flow rate, the mean velocity $\bar{u}$, is retrieved from:

$$\bar{u} = \frac{Q}{WL}. \quad (A.16)$$

The normalized velocity $u/\bar{u}$ can then be expressed as:

$$\frac{u}{\bar{u}} = \left(\frac{2(\nu-1)/2}{\mu}\right)^{1/N} \cdot \frac{N}{N+1} \left\{ \begin{array}{ll}
\left(\frac{L}{2} - y_0\right)^{1/(N+1)}, & 0 \leq y \leq y_0 \\
\left(\frac{L}{2} - y_0\right)^{1/(N+1)} - (y - y_0)^{1/(N+1)}, & y_0 < y \leq \frac{L}{2} 
\end{array} \right. \quad (A.17)$$
which finally becomes:

\[
\frac{u}{\bar{u}} = \frac{1 + 2N}{1 + N \left(1 + \frac{y_0}{L/2} \right)} \begin{cases} 
1, & 0 \leq y \leq y_0 \\
1 - \left(\frac{y - y_0}{L/2 - y_0} \right)^{1/N+1}, & y_0 < y \leq L/2
\end{cases}, \quad (A.18)
\]

which represents the normalized velocity of Poiseuille flow of Herschel-Bulkley fluid. The corresponding equations for Power-Law and Bingham fluids are retrieved if we set \(\tau_Y = 0 = y_0\) and \(N = 1\) respectively. If we set both \(\tau_Y = 0\) and \(N = 1\), the familiar Newtonian formulation is retrieved.
Appendix B


In this section, the analytical solution of the steady state, two-dimensional, two-phase Newtonian/non-Newtonian plane Poiseuille flow is going to be presented. The definition and the configuration of the problem can be found in Section 6.5.2 and Figure 6.5 respectively.

B.1 Bingham fluid

In this case we assume that the Poiseuille flow is driven by a body force $\mathbf{F}$ and not from a pressure difference $dp/dx$. Thus, the conservation of momentum equation becomes:

$$\frac{\partial u^x}{\partial t} = \frac{\partial \tau_{yx}}{\partial y} + \mathbf{F}, \quad (B.1)$$

where $u^x$ the velocity on $x$ direction and $\tau_{yx}$ is the shear stress on the fluid. For a steady state problem we have $\partial u^x/\partial t = 0$. The equation for the stress term of the Newtonian fluid of phase $a$ is

$$\mathbf{\tau} = \mu \mathbf{D}, \quad (B.2)$$
while for the Bingham fluid of phase $b$ the stress term can be retrieved from Vola et al. (2004), with

\[
\begin{cases}
   |\tau| \leq \tau_Y & \Rightarrow \quad D = 0 \\
   |\tau| > \tau_Y & \Rightarrow \quad \tau = \left( \frac{\tau_Y}{D} + \mu \right) D
\end{cases}, \quad (B.3)
\]

where

\[
|D| = \sqrt{\frac{1}{2} \sum_{ij} D_{ij} D_{ij}} = \sqrt{2 \left( \frac{\partial u}{\partial x} \right)^2 + 2 \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2}, \quad (B.4)
\]

with $\partial u/\partial x = \partial v/\partial y = \partial v/\partial x = 0$. Thus,

\[
|D| = \frac{\partial u}{\partial y} \quad (B.5)
\]

and therefore, equations B.2 and B.3 become

\[
\tau^{yx} = \mu \frac{\partial u}{\partial y} \quad (B.6)
\]

and

\[
\begin{cases}
   |\tau| \leq \tau_Y & \Rightarrow \quad \frac{\partial u}{\partial y} = 0 \\
   |\tau| > \tau_Y & \Rightarrow \quad \tau^{yx} = \tau_Y + \mu \frac{\partial u}{\partial y}
\end{cases}, \quad (B.7)
\]

respectively. Moreover, for steady state solution we have $\partial u/\partial t = 0$. Thus, equation B.1 becomes:

\[
\frac{\partial \tau^{yx}}{\partial y} + F_x = 0 \Rightarrow \frac{\partial \tau^{yx}}{\partial y} = -F_x \quad . \quad (B.8)
\]

After combining equations B.6, B.7 with B.8, and integrating once upon $y$, we get the following system:

\[
\begin{cases}
   \tau^{yx}_{I} = -F_x \cdot y + c_I^I & \text{for phase } a \\
   \tau^{yx}_{II} = -F_x \cdot y + c_{II}^I & \text{for un-yielded part of phase } b \\
   \tau^{yx}_{III} = -F_x \cdot y + c_{III}^I & \text{for yielded part of phase } b
\end{cases}, \quad (B.9)
\]
or by substituting B.6 and B.7

\[
\begin{align*}
\mu_a \frac{\partial u^I}{\partial y} &= -F_x \cdot y + c^I_1 & \text{for phase } a \\
\frac{\partial u^{II}}{\partial y} &= 0 & \text{for un-yielded part of phase } b \\
\tau_Y + \mu_b \frac{\partial u^{III}}{\partial y} &= -F_x \cdot y + c^{III}_1 & \text{for yielded part of phase } b
\end{align*}
\]  \quad \text{(B.10)}

Assuming that the interphase of the two fluids lies on \( y = 0 \) of a channel of height 2\( L \) with the walls being on \( y = -L \) and \( y = L \) the following boundary conditions are retrieved:

\[
\begin{align*}
y = 0 \quad &\rightarrow \tau_{yI}^{yx} = \tau_{yI}^{yx} \\
y = y_0 \quad &\rightarrow \tau_{yI}^{yx} = \tau_{yIII}^{yx}
\end{align*}
\]

where \( y_0 \) the point where the Bingham fluid transitions from yielded to un-yielded zones. Applying the boundary conditions of equation B.11 to B.9 one can easily get \( c^I_1 = c^{II}_1 = c^{III}_1 = c_1 \). Also from second part of equation B.7 at \( y_0 \)

\[
\tau_{yIII}^{yx} = \tau_Y, \quad \text{(B.12)}
\]

which if substituted on the third equation of B.9 for \( y = y_0 \) gives

\[
y_0 = \frac{-\tau_Y + c_1}{F_x}. \quad \text{(B.13)}
\]

If equation B.10 is rewritten in terms of \( \partial u/\partial y \) and integrated once more upon \( y \), one gets:

\[
\begin{align*}
u^I &= \frac{-F_x}{2\mu_a} y^2 + \frac{c_1}{\mu_a} y + c^I_2 & \text{for phase } a \\
u^{II} &= c^{II}_2 & \text{for un-yielded part of phase } b \\
u^{III} &= \frac{-F_x}{2\mu_b} y^2 - \frac{\tau_Y}{\mu_b} y + \frac{c_1}{\mu_b} y + c^{III}_2 & \text{for yielded part of phase } b
\end{align*}
\]

\[ \text{(B.14)} \]
The following set of boundary conditions are applied:

\[
\begin{align*}
& y = 0 \quad \rightarrow \quad u^I = u^{II} \quad (i) \\
& y = y_0 \quad \rightarrow \quad u^{II} = u^{III} \quad (ii) \\
& y = L \quad \rightarrow \quad u^I = 0 \quad (iii) \\
& y = -L \quad \rightarrow \quad u^{III} = 0 \quad (iv)
\end{align*}
\]  
(B.15)

Substituting B.15(i) to B.14 one gets  \( c^I_2 = c^{II}_2 = c_2 \). Moreover from B.15(iii) and B.15(iv)

\[
0 = -\frac{F_x}{2\mu_a} L^2 + \frac{c_1}{\mu_a} L + c_2 \Rightarrow
\]

\[
\Rightarrow c_2 = \frac{F_x}{2\mu_a} L^2 - \frac{c_1}{\mu_a} L ,
\]  
(B.16)

and

\[
0 = -\frac{F_x}{2\mu_b} L^2 + \frac{\tau_Y}{\mu_b} L - \frac{c_1}{\mu_b} L + c^{III}_2 \Rightarrow
\]

\[
\Rightarrow c^{III}_2 = \frac{F_x}{2\mu_b} L^2 - \frac{\tau_Y}{\mu_b} L + \frac{c_1}{\mu_b} L ,
\]  
(B.17)

respectively. Finally, from B.15(ii) substituted in B.14 one gets:

\[
c_2 = -\frac{F_x}{2\mu_b} y_0^2 - \frac{\tau_Y}{\mu_b} y_0 + \frac{c_1}{\mu_b} y_0 + c^{II}_2 \Rightarrow
\]

\[
\Rightarrow \frac{F_x}{2\mu_b} y_0^2 + \frac{\tau_Y}{\mu_b} y_0 - \frac{c_1}{\mu_b} y_0 - c^{II}_2 + c_2 = 0 .
\]  
(B.18)

If B.16 and B.17 are substituted in B.18, one gets:

\[
\frac{F_x}{2\mu_b} y_0^2 + \frac{\tau_Y}{\mu_b} y_0 - \frac{c_1}{\mu_b} y_0 - \frac{F_x}{2\mu_b} L^2 + \frac{\tau_Y}{\mu_b} L - \frac{c_1}{\mu_b} L + \frac{F_x}{2\mu_a} L^2 - \frac{c_1}{\mu_a} L = 0 \Rightarrow
\]

\[
\Rightarrow \frac{F_x}{2\mu_b} y_0^2 + \frac{\tau_Y}{\mu_b} y_0 - \frac{c_1}{\mu_b} y_0 + \left( \frac{\mu_b - \mu_a}{\mu_b \mu_a} \right) \frac{F_x}{2} L^2 - \left( \frac{\mu_b + \mu_a}{\mu_b \mu_a} \right) c_1 L + \frac{\tau_Y}{\mu_b} L = 0 \Rightarrow
\]

\[
\Rightarrow \frac{F_x}{2\mu_b} y_0^2 + (\tau_Y - c_1) y_0 + \left( \frac{\mu_b - \mu_a}{\mu_a} \right) \frac{F_x}{2} L^2 - \left( \frac{\mu_b + \mu_a}{\mu_a} \right) c_1 L + \tau_Y L = 0 .
\]  
(B.19)
Substituting $y_0$ from equation B.13 in B.19 one gets:

$$\frac{F_x}{2} \left( -\tau_Y + c_1 \right)^2 + (\tau_Y - c_1) \frac{-\tau_Y + c_1}{F_x} + \left( \frac{\mu_b - \mu_a}{\mu_a} \right) \frac{F_x}{2} L^2 - \left( \frac{\mu_b + \mu_a}{\mu_a} \right) c_1 L + \tau_Y L = 0 .$$

By multiplying with $2F_x$ B.20 becomes

$$(-\tau_Y + c_1)^2 - 2(\tau_Y - c_1)^2 + \left( \frac{\mu_b - \mu_a}{\mu_a} \right) F_x^2 L^2 - 2F_x \left( \frac{\mu_b + \mu_a}{\mu_a} \right) c_1 L + 2F_x \tau_Y L = 0 \Rightarrow$$

$$\Rightarrow -\left( \tau_Y - c_1 \right)^2 + \left( \frac{\mu_b - \mu_a}{\mu_a} \right) F_x^2 L^2 - 2F_x \left( \frac{\mu_b + \mu_a}{\mu_a} \right) c_1 L + 2F_x \tau_Y L = 0 \Rightarrow$$

$$\Rightarrow -c_1^2 + 2 \left( \tau_y - F_x L \frac{\mu_b + \mu_a}{\mu_a} \right) c_1 - \tau_Y^2 + \left( \frac{\mu_b - \mu_a}{\mu_a} \right) F_x^2 L^2 + 2F_x \tau_Y L = 0 . \quad (B.21)$$

Equation B.20 is a quadratic equation with single unknown the constant $c_1$. Its two possible solutions, $c_1^{I,II}$ read:

$$c_1^{I,II} = \frac{-2 \left( \tau_y - F_x L \frac{\mu_b + \mu_a}{\mu_a} \right) \pm \sqrt{4 \left( \tau_y - F_x L \frac{\mu_b + \mu_a}{\mu_a} \right)^2 + 4 \left( \tau_Y^2 + \frac{\mu_b - \mu_a}{\mu_a} \right) F_x^2 L^2 + 2F_x \tau_Y L}}{2} \Rightarrow$$

$$\Rightarrow c_1^{I,II} = \tau_y - F_x L \frac{\mu_b + \mu_a}{\mu_a} \pm \sqrt{ \left( \tau_y - F_x L \frac{\mu_b + \mu_a}{\mu_a} \right)^2 - \tau_Y^2 + \left( \frac{\mu_b - \mu_a}{\mu_a} \right) F_x^2 L^2 + 2F_x \tau_Y L} \quad (B.22)$$

Having defined $c_1$, values of constants $c_2$, $c_2^{III}$ and $y_0$ are recovered from equations B.16, B.17 and B.13 respectively and substituted in equation B.14 to give the velocity profiles of the three different parts of the flow.

### B.2 Power-law fluid

Similarly, with Bingham case the power-law Poiseuille flow is assumed to be driven by a body force $F$ and not from a pressure difference $dp/dx$. Thus, the conservation of momentum equation becomes:

$$\frac{\partial u^x}{\partial t} = \frac{\partial u^{yx}}{\partial y} + F . \quad (B.23)$$
where $u^x$ the velocity on $x$ direction and $\tau^{yx}$ is the shear stress on the fluid. For a steady-state problem we have $\partial u^x/\partial t = 0$. The equation for the stress term of a Newtonian fluid is

$$\tau = \mu D , \quad \text{(B.24)}$$

while for a power-law fluid the stress term can be retrieved from Vola et al. (2004), with

$$\tau = (\mu |D|^{(N-1)}) D , \quad \text{(B.25)}$$

where $N$ is the power-law exponent. Similar with the Bingham case,

$$|D| = \frac{\partial u}{\partial y} \quad \text{(B.26)}$$

and therefore, equations B.24 and B.25 become

$$\tau^{yx} = \mu \frac{\partial u}{\partial y} \quad \text{(B.27)}$$

and

$$\tau^{yx} = \mu \left( \frac{\partial u}{\partial y} \right)^N , \quad \text{(B.28)}$$

respectively. Moreover, for steady state solution we have $\partial u/\partial t = 0$. Thus, equation B.23 becomes:

$$\frac{\partial \tau^{yx}}{\partial y} + F_x = 0 \Rightarrow \frac{\partial \tau^{yx}}{\partial y} = -F_x . \quad \text{(B.29)}$$

After combining equations B.27, B.28 with B.29, and integrating once upon $y$, we get the following system:

$$\begin{cases}
\tau^{yx}_I = -F_x \cdot y + c^I_1 \quad \text{for phase } a \\
\tau^{yx}_{II} = -F_x \cdot y + c^{II}_1 \quad \text{for phase } b
\end{cases} , \quad \text{(B.30)}$$

or by substituting B.6 and B.7

$$\begin{cases}
\mu_a \frac{\partial u^I}{\partial y} = -F_x \cdot y + c^I_1 \quad \text{for phase } a \\
\mu_b \left( \frac{\partial u^{II}}{\partial y} \right)^N = -F_x \cdot y + c^{II}_1 \quad \text{for phase } b
\end{cases} . \quad \text{(B.31)}$$
Assuming that the interphase of the two fluids lies on \( y = 0 \) of a channel of height \( 2L \) with the walls being on \( y = -L \) and \( y = L \) the following boundary condition is retrieved:

\[
\tau_{yx}^I = \tau_{yx}^{II}. \tag{B.32}
\]

Applying the boundary condition of equation B.32 to B.30 one can easily get \( c_1^I = c_1^{II} = c_1 \). If equation B.31 is rewritten in terms of \( \partial u / \partial y \) and integrated once more upon \( y \), one gets:

\[
\begin{cases}
  u^I = \frac{-F_x}{2\mu_a} y^2 + \frac{c_1}{\mu_a} y + c_1^I \\
  u^{II} = -\left( \frac{1}{\mu_b} \right)^{1/N} \left( \frac{N}{N+1} \right) \left( -\frac{F_x y + c_1}{F_x} \right)^{1/N+1} + c_2^{II}
\end{cases}
\text{for phase } a \tag{B.33}
\]

The following set of boundary conditions are applied:

\[
\begin{cases}
  y = 0 \rightarrow u^I = u^{II} \ (i) \\
  y = L \rightarrow u^I = 0 \ (ii) \\
  y = -L \rightarrow u^{II} = 0 \ (iii)
\end{cases}
\tag{B.34}
\]

Substituting B.34(i) to B.33 one gets

\[
c_2^I = -\left( \frac{1}{\mu_b} \right)^{1/N} \left( \frac{N}{N+1} \right) \frac{c_1^{1/N+1}}{F_x} + c_2^{II} \tag{B.35}
\]

Moreover from B.34(ii) and B.34(iii)

\[
0 = \frac{-F_x}{2\mu_a} L^2 + \frac{c_1}{\mu_a} L + c_2^I \Rightarrow
\]

\[
\Rightarrow c_2^I = \frac{F_x}{2\mu_a} L^2 - \frac{c_1}{\mu_a} L, \tag{B.36}
\]

and

\[
0 = -\left( \frac{1}{\mu_b} \right)^{1/N} \left( \frac{N}{N+1} \right) \frac{(F_x L + c_1)^{1/N+1}}{F_x} + c_2^{II} \Rightarrow
\]

\[
\Rightarrow c_2^{II} = \left( \frac{1}{\mu_b} \right)^{1/N} \left( \frac{N}{N+1} \right) \frac{(F_x L + c_1)^{1/N+1}}{F_x}, \tag{B.37}
\]
respectively. If B.36 and B.37 are substituted in B.35, one gets:

\[
\frac{F_x}{2\mu_a} L^2 - \frac{c_1}{\mu_a} L = - \left( \frac{1}{\mu_b} \right)^{1/N} \left( \frac{N}{N + 1} \right) \frac{c_1^{1/N+1}}{F_x} + \frac{1}{\mu_b} \left( \frac{N}{N + 1} \right) \frac{(F_x L + c_1)^{1/N+1}}{F_x} \Rightarrow
\]

\[
\Rightarrow \left( \frac{1}{\mu_b} \right)^{1/N} \left( \frac{N}{N + 1} \right) \left( F_x L + c_1 \right)^{1/N+1} - \frac{c_1^{1/N+1}}{F_x} + \frac{\mu_a}{\mu_b} L - \frac{F_x}{2\mu_a} L^2 = 0. \quad (B.38)
\]

Equation B.38 has as a single unknown the constant \( c_1 \), which can be defined arithmetically (e.g. with Newton-Raphson method). Having defined \( c_1 \), values of constants \( c_2 \) and \( c_2' \) are recovered from equations B.36 and B.37 respectively and substituted in equation B.33 to give the velocity profiles of the Newtonian and the power-law parts of the flow.