SMOOTHED PARTICLE HYDRODYNAMICS: DEVELOPMENT AND APPLICATION TO PROBLEMS OF HYDRODYNAMICS

DISSERTATION

Submitted in Partial Fulfillment of

the Requirements for

the Degree of

DOCTOR OF PHILOSOPHY

 \mathbf{in}

Mechanical Engineering

at the

NEW YORK UNIVERSITY TANDON SCHOOL OF ENGINEERING

by

Angelantonio Tafuni

January 2016

Approved:

Department Chair Signature

University ID: N11831804

Date

Approved by the Guidance Committee:

Major: Mechanical Engineering

Iskender Sahin, Ph.D., Chair

Industry Professor of Mechanical Engineering NYU Tandon School of Engineering

Date

Matteo Aureli, Ph.D. Assistant Professor of Mechanical Engineering University of Nevada, Reno

Date

Richard S. Thorsen, Ph.D. Department Chair and Vice President Emeritus NYU Tandon School of Engineering

Date

George Vradis, Ph.D.

Associate Professor of Mechanical Engineering NYU Tandon School of Engineering

Date

Microfilm or copies of this dissertation may be obtained from:

UMI Dissertation Publishing ProQuest CSA 789 E. Eisenhower Parkway P.O. Box 1346 Ann Arbor, MI 48106-1346

Vita

Angelo Tafuni was born in Altamura (Bari, IT) on June 07, 1985. He earned a Bachelor of Science degree in Mechanical Engineering from the Polytechnic University of Bari (Bari, IT) in 2009. After a year as an exchange student and trainee at the Eindhoven University of Technology (Eindhoven, NL), he returned to Bari and enrolled in a double-degree program between the Polytechnic University of Bari and the (then) Polytechnic Institute of New York University (Brooklyn, USA). He earned a Master of Science degree in Mechanical Engineering from both institutions in December 2014 and January 2012, respectively. Mr. Tafuni entered the Ph.D. program in the Department of Mechanical and Aerospace Engineering of the Polytechnic Institute of New York University in January 2012. In September 2012, he was appointed instructor of the Fluid Mechanics and Heat Transfer laboratory courses in the same department, with a four-year teaching fellowship that also funded his doctoral studies. He has received several grants and scholarships, among which the ASME Outreach for Engineers scholarship by the OOAE Division of ASME-IPTI, Graduate Merit-Based scholarships from both Polytechnic of Bari and New York University, a three-year New York City housing grant from the Honors Center of Italian University (H2CU) and several travel grants from the New York University. He has served as a peer reviewer for several scientific journals. He is also a member of The New York Academy of Science and the American Society of Mechanical Engineers (ASME), for which he has served as a peer reviewer for different Mechanical and Ocean Engineering conferences. All the research presented hereinafter was carried out by Mr. Tafuni during the period January 2012 to December 2015.

Vita

Angelo Tafuni nasce ad Altamura (Bari, IT) il 07 Giugno 1985. Nel 2009, riceve la laurea di primo livello in Ingegneria Meccanica presso il Politecnico di Bari (Bari, IT). Trascorre i mesi successivi presso la Eindhoven University of Technology (Eindhoven, NL) come studente internazionale e tirocinante, per poi ritornare a Bari ed iscriversi al programma di double-degree tra il Politecnico di Bari e l'allora Polytechnic Institute of New York University (Brooklyn, USA). Ottiene la laurea magistrale in Ingegneria Meccanica da entrambe le università nel Dicembre 2014 e Gennaio 2012, rispettivamente. Si candida al dottorato di ricerca nel dipartimento di Ingegneria Meccanica ed Aerospaziale del Polytechnic Institute of New York University nel Gennaio 2012. Successivamente viene nominato per l'insegnamento dei laboratori di Fluidodinamica e Fisica Tecnica nello stesso dipartimento nel Settembre 2012, grazie ad una teaching fellowship della durata di quattro anni che contemporaneamente finanzierà i suoi studi. Angelo Tafuni ha ricevuto diversi riconoscimenti e borse di studio, tra cui la ASME Outreach for Engineers Scholarship dalla OOAE Division di ASME-IPTI, borse di merito dal Politecnico di Bari e dalla New York University, tre anni di alloggio presso i prestigiosi appartamenti di Battery Park City (New York, USA) gestiti dall'Honors Center of Italian University (H2CU) e diverse borse di studio per viaggi dalla New York University. Ha lavorato come revisore per riviste di carattere scientifico. E inoltre membro della New York Academy of Science e dell'American Society of Mechanical Engineers (ASME), per la quale ha lavorato come revisore in diverse conferenze di Ingegneria Meccanica e Navale. Il lavoro di ricerca presentato in questa tesi è stato svolto da Angelo Tafuni nel periodo Gennaio 2012 – Dicembre 2015.

Acknowledgements

I wish to express my sincere appreciation to my advisor, Professor Iskender Sahin, for giving me the opportunity to develop this research and for his guidance throughout all phases of my doctorate.

I am also grateful for having three distinguished professors of Mechanical Engineering in my Ph.D. guidance committee. I have consolidated my knowledge of Fluid Dynamics, Heat Transfer, and CFD from the courses instructed here at NYU by Professor Richard Thorsen and Professor George Vradis, whose support has been crucial in many stages of my doctorate. Moreover, I am indebted to Professor Matteo Aureli at the University of Nevada, Reno, for many fruitful discussions and Skype meetings that have led to essential improvements of this research work.

I would like to thank Professor Leslie Greengard at the Courant Institute of Mathematical Sciences (NYU) for teaching me much about scientific computing and fast algorithms in his unique and effective way.

Special thanks to the faculty of Mechanical Engineering of the Politecnico di Bari, my Italian alma mater, for giving me a solid academic background and the proper forma mentis to engage in any challenge.

A thousand thanks to Dr. Alejandro Crespo, Dr. José Domínguez and Dr. Renato Vacondio for the constant help and advice with the DualSPHysics code and the SPH method in general: this work would not have been the same without your support.

A dear thank you to all friends and colleagues that I have been fortunate enough to meet in the United States and to those living in my beautiful Italy, with whom I have shared some of the best moments of my life.

Most importantly, I would like to express my deep gratitude to my girlfriend and partner in crime, Carlotta, for sharing this endeavour with me since day one, and to my family, mamma, Ciccio, Deborah, Antonello and Karole, grandma Lina and grandpa Nicola and all the members of the beautiful Colonna family: each of you has contributed to shaping me into the man that I am today, and for this I am forever in your debt.

Ringraziamenti

Vorrei esprimere la mia profonda gratitudine nei confronti del mio advisor, Professor Iskender Sahin, per avermi dato l'opportunità di sviluppare questo lavoro di ricerca e per la sua guida durante tutte le fasi del mio dottorato.

Sono inoltre grato per aver avuto tre illustri professori di Ingegneria Meccanica nella mia commissione di dottorato. Ho approfondito la mia conoscenza della fluidodinamica teorica e numerica, nonchè della termodinamica, grazie ai corsi presso la New York University dei Professori Richard Thorsen e George Vradis, il cui sostegno è stato fondamentale in molte delle fasi del mio dottorato. Mi sento inoltre in debito con il Professor Matteo Aureli dell'Università del Nevada, Reno, per le tante discussioni produttive e gli incontri via Skype che hanno contribuito notevolmente al miglioramento di questo lavoro di ricerca.

Vorrei ringraziare il Professor Leslie Greengard del Courant Institute of Mathematical Sciences (NYU) per avermi insegnato tanto nell'ambito della scienza computazionale e degli algoritmi veloci con il suo stile unico ed efficace.

Uno speciale ringraziamento va al corpo docenti di Ingegneria Meccanica del Politecnico di Bari, l'università nella quale ho condotto gli studi in Italia, per avermi fornito solide basi scientifiche e la giusta forma mentis per affrontare qualsiasi sfida.

Mille grazie al Dr. Alejandro Crespo, Dr. José Domínguez e Dr. Renato Vacondio per la loro costante guida durante l'uso del codice DualSPHysics ed il metodo SPH in generale: questo lavoro non sarebbe stato lo stesso senza il vostro aiuto.

Un sentito ringraziamento a tutti gli amici e colleghi che ho avuto la fortuna di conoscere negli Stati Uniti e quelli che invece vivono nella mia bella Italia, e con i quali ho condiviso alcuni dei più bei momenti della mia vita.

Infine, come cosa più importante, vorrei esprimere la mia profonda gratitudine alla mia compagna e complice, Carlotta, per aver condiviso questa esperienza con me sin dal primo giorno, ed alla mia famiglia, mamma, Ciccio, Deborah, Antonello e Karole, nonna Lina, nonno Nicola e tutti i membri della bellissima famiglia Colonna: ognuno di voi ha contribuito a rendermi l'uomo che sono oggi, e per questo vi sono per sempre debitore. Questa tesi è dedicata a mio nonno, Nicola.

ABSTRACT

SMOOTHED PARTICLE HYDRODYNAMICS: DEVELOPMENT AND APPLICATION TO PROBLEMS OF HYDRODYNAMICS

by

Angelantonio Tafuni

Advisor: Prof. Iskender Sahin, Ph.D., P.E.

Submitted in Partial Fulfillment of the Requirements for

the Degree of Doctor of Philosophy in Mechanical Engineering

January 2016

The research work in this dissertation focuses on Smoothed Particle Hydrodynamics (SPH), a fully Lagrangian particle method for modelling the behavior of fluids in a computational framework. The main goal is to use and improve aspects of this Computational Fluid Dynamics (CFD) technique for the solution of novel problems of hydrodynamics including free surface flows. To this extent, simulations of several CFD problems are presented and the accuracy of the SPH results is compared with numerous works in the cited literature.

A SPH investigation of the wave and bottom pressure fields generated by a fast hull in finite-depth water is conducted for different flow conditions. This subject is relevant to several engineering studies, therefore an assessment of the correlation between water waves and pressure disturbances at the seafloor is made in a quantitative manner. Furthermore, SPH is used for the first time to study harmonic oscillations of a thin rigid lamina in a viscous fluid with and without a free surface. The results provide useful insights about the hydrodynamic load dependence on phenomena of vortex shedding and advection, which are well captured by the SPH method. Finally, a functional relationship is introduced to express the total fluid force as added mass and damping coefficients and to quantify their dependence on the control parameters governing the problem.

Computational findings are also used to identify the shortcomings of SPH and part of the research is devoted to understanding and improving these aspects. Particularly, two current SPH challenges are of interest in this dissertation: the first is concerned with the concept of adaptivity in SPH and the testing of particle refinement/derefinement techniques for introducing higher resolutions in areas where detailed flow information is critical (e.g. boundary layer, free surface, stagnation areas). To this extent, numerical solutions of a dam break and sloshing problems are presented, corroborating an increase in the accuracy of the simulations and a substantial gain in computational time with respect to using uniformly distributed particles. The second major direction of improvement regards the development of a new type of boundary condition to simulate open boundaries in SPH. This is not a trivial task in Lagrangian methods as particles must be inserted in/removed from the domain while preserving physical quantities and numerical conditions such as stability and consistency. Simulations presented herein are validated against several other works in the literature for both 2-D and 3-D problems, highlighting a good agreement among the results.

Overall, data from the computational studies in this research supports the importance of both adaptivity and open boundary conditions towards achieving accurate SPH solutions of real-life engineering problems.

Contents

1	Intr	oducti	ion	1
	1.1	Appro	oaches in Computational Fluid Dynamics	1
	1.2	Resear	rch purpose and contributions	4
	1.3	Thesis	soutline	5
2	Smo	oothed	Particle Hydrodynamics	7
	2.1	SPH f	ormalism	7
		2.1.1	Integral approximation	7
		2.1.2	Particle approximation	8
		2.1.3	The smoothing function	9
	2.2	Gover	ning equations	11
		2.2.1	Incompressible vs Weakly-Compressible SPH	11
		2.2.2	Derivation of the SPH conservation equations	13
	2.3	Nume	rical aspects of the SPH methodology	16
		2.3.1	Time integration	16
		2.3.2	Stability	17
		2.3.3	Accuracy	19
		2.3.4	Consistency	20
		2.3.5	Convergence	21
	2.4	Viscos	sity models	22
		2.4.1	Laminar formulation	22
		2.4.2	Sub Particle Scaling (SPS): an approach to turbulence	23
	2.5	Bound	lary conditions	23
		2.5.1	Dynamic boundary particles	24
		2.5.2	Free-surface considerations	25
	2.6	The D	DualSPHysics project	26

3	3-D	flow f	field generated by a planing hull in finite-depth water	27
	3.1	Litera	ture review	27
	3.2	Proble	em formulation and numerical set-up	29
	3.3	SPH s	solver and parameters	31
	3.4	RANS	S solver	32
	3.5	Result	ts with the artificial viscosity model	33
		3.5.1	Bottom pressure	33
	3.6	Result	ts with the SPS viscosity model	37
		3.6.1	Wave elevation	38
		3.6.2	Bottom Pressure	40
		3.6.3	Comparison with RANS solutions and assessment of the low-	
			pressure wake angle	44
4	Har	rmonic	oscillations of a rigid lamina in a viscous fluid with an	d
	wit	hout a	free surface	47
	4.1	Litera	ture review	48
	4.2	Gover	ning equations	49
	4.3	Result	ts for a fluid of infinite extent	52
		4.3.1	Velocity and vorticity fields	52
		4.3.2	Pressure field	57
		4.3.3	Total hydrodynamic force	58
		4.3.4	Hydrodynamic function	62
		4.3.5	Correlation with existing results	66
	4.4	Result	ts for a fluid with a free surface	67
		4.4.1	Boundary integral formulation	67
		4.4.2	Added mass and damping for infinitely small oscillations $\ . \ .$	70
		4.4.3	Effects of finite amplitudes: a preliminary study	74
5	Var	iable r	resolution SPH	77
	5.1	The p	article splitting and coalescing technique	77
	5.2	Prelin	ninary results	80
		5.2.1	Dam break	80
		5.2.2	Water sloshing under external tank excitation	99
6	Ope	en bou	ndary conditions in DualSPHysics	104
	6.1	Imple	mentation \ldots	104
	6.2	2-D te	est cases	108

		6.2.1	Liquid jet impinging on a flat plate	108
		6.2.2	Channel flow past a cylinder	112
	6.3	3-D te	st cases	122
		6.3.1	Liquid jet impinging on a flat and inclined plate	122
		6.3.2	Channel flow past a cube	126
7	Con	clusio	ns and future works	134
\mathbf{Li}	st of	public	ations	138
Bi	Bibliography 14		140	

List of Figures

1.1	Representation of a generic domain with and without the use of a mesh.	2
2.1	Some common kernel functions (solid curves) with their first (red dashed curves) and second (green dashed curves) derivatives and comparison	11
0.0	with the Gaussian kernel (dotted curves) (Price, 2012)	11
2.2	The DualSPHysics logo.	26
3.1	Sketch of the proposed 3-D model.	30
3.2	Sketch of the computational domain with top and side views	31
3.3	CAD model of the planing boat	31
3.4	Pressure contour plots for $h_w = 0.62L$ and $Fr_h = 1.00$ (top-left panel),	
	$\mathrm{Fr}_{\mathrm{h}}=2.00$ (top-right panel), $\mathrm{Fr}_{\mathrm{h}}=3.00$ (bottom-left panel), and $\mathrm{Fr}_{\mathrm{h}}=4.0$	0
	(bottom-right panel)	34
3.5	Bottom pressure at several lateral positions y/d	35
3.6	Bottom pressure signatures at the centerline, $y/d = 0$, as a function of	
	the Froude number	37
3.7	Contours of nondimensional wave elevation, $\eta,$ for different depth Froude	
	numbers	38
3.8	Wave elevation along $y/L = 0$ for different depth Froude numbers	39
3.9	Contours of nondimensional bottom pressure, θ , for different depth	
	Froude numbers.	40
3.10	Bottom pressure along $y/L = 0$ for Fr _h = 2.00: simulation data (dashed	
	line), fitted curves (solid lines).	41
3.11	Nondimensional bottom pressure signatures for $Fr_h \in [1.00 - 4.00]$.	42
3.12	Nondimensional bottom pressure signatures at the boat centerline as	
	a function of the depth Froude number	42

3.13	Plots of nondimensional bottom pressure, θ , (solid curve) and wave elevation, η , (dashed curve) at different x/L positions for Fr _h = 3.00.	
	Scales are relative to nondimensional pressure. Wave elevation is scaled	
	for comparison.	4
3.14	RANS bottom pressure contours, $Fr_L = 2.00$	4
3.15	SPH bottom pressure contours, $Fr_L = 2.00$	4
3.16	Low-pressure wake half-angle (radians) as a function of the depth	
	Froude number.	4
4.1	3-D sketch of a thin cantilever beam with clamped base and free end	
	(left) and projection of the cross-section Ψ onto its plane (right)	5
4.2	Velocity (left panels) and vorticity (right panels) contours for $\beta = 976$	
	and $\text{KC} = \pi/10$	5
4.3	Velocity (left panels) and vorticity (right panels) contours for $\beta = 976$	
	and $KC = 3\pi/5$	5
4.4	Velocity (left panels) and vorticity (right panels) contours for $\beta = 976$	
	and $KC = 6\pi/5$	5
4.5	Evolution of the pressure field for $\beta = 976$ and KC = $\pi/10.$	5
4.6	Evolution of the pressure field for $\beta = 976$ and $\text{KC} = 6\pi/5$	5
4.7	Hydrodynamic force, $F(t)$, and displacement, $z(t)$, as a function of the	
	non-dimensional time for $\beta = 1952$. Forces are in Nm ⁻¹ . Displacement	
	(blue line) is scaled to allow for comparison. (a) Time history of $F(t)$	
	for KC = $\pi/25$ with domain size $3B \times 2B$ (red vertically-dashed line),	
	$3B \times 3B$ (red dashed line), $4B \times 4B$ (red solid line), $5B \times 5B$ (black	
	solid line). (b) Time history of $F(t)$ for KC = $3\pi/5$ with domain size	
	$5B \times 5B$ (red vertically-dashed line), $10B \times 10B$ (red dashed line), $15B \times 15B$ (red vertically-dashed line),	٣
1 0	$15B \times 15B$ (red solid line), $20B \times 20B$ (black solid line)	Э
4.0	Coefficients of the fundamental harmonic, A_1, D_1 , as a function of the Keulegen Competer number	6
4.0	Coefficients A A and B B as a function of the Keulogen	0
4.9	Coefficients A_0 , A_2 ,, A_5 and D_2 ,, D_5 as a function of the Keulegan- Corportor number for $\beta = 244$, $\beta = 488$, $\beta = 076$, and $\beta = 1052$. (a)	
	(d) absolute value of the cosine coefficients (a) (b) absolute value of	
	(a) absolute value of the cosine coefficients. (e) - (n) absolute value of the sine coefficients	6
4 10	Beal (top) and imaginary (bottom) parts of $\Lambda(\beta KC)$	6
4 11	Surface representation of the damping coefficient $-\text{Im}[\Theta(\beta \text{KC})]$ i.e.	U
1.11	the imaginary part of Equation (4.9)	6
	$\frac{1}{100}$	U

4.12	Linear correlation between numerical findings from SPH simulations	
	(squares) and results from the fit applied to Equation (4.9) (line). \therefore	64
4.13	Comparison of added mass (left) and damping (right) coefficients from	
	present findings with other works in the literature	66
4.14	Nomenclature for the vibrational problem	67
4.15	Non-dimensional hydrodynamic force as a function of the non-dimensional	
	depth of submergence, H	71
4.16	Total hydrodynamic force per unit length expressed as added mass and	
	damping coefficients as a function of the Reynolds number	72
4.17	Numerical results from BEM (upward triangles) and SPH (dots).	73
4.18	Contours of velocity magnitude for $\beta = 500$, KC = 0.628 and $\chi = 2.50$.	74
4.19	Contours of velocity magnitude for $\beta = 500$, KC = 0.628, and $\chi = 0.50$.	75
4.20	Time history of the fluid force for $\beta = 500$ and several values of the	
	depth.	76
5.1	Optimal particle splitting pattern with current code implementation	
	(Vacondio et al., 2015). \ldots \ldots \ldots \ldots \ldots \ldots \ldots	78
5.2	Nomenclature for the 2-D dam break problem	81
5.3	Velocity field at time $t = 0$ s	82
5.4	Velocity field at time $t = 0.15$ s	83
5.5	Velocity field at time $t = 0.40$ s	84
5.6	Velocity field at time $t = 0.70$ s	85
5.7	Velocity field at time $t = 0.99$ s	86
5.8	Velocity field at time $t = 1.01$ s	87
5.9	Velocity field at time $t = 1.05$ s	88
5.10	Velocity field at time $t = 1.11$ s	89
5.11	Pressure field at time $t = 0$ s	90
5.12	Pressure field at time $t = 0.15$ s	91
5.13	Pressure field at time $t = 0.40$ s	92
5.14	Pressure field at time $t = 0.70$ s	93
5.15	Pressure field at time $t = 0.99$ s	94
5.16	Pressure field at time $t = 1.01$ s	95
5.17	Pressure field at time $t = 1.05$ s	96
5.18	Pressure field at time $t = 1.11$ s	97
5.19	Close-up on the velocity field around the basin right corner for both	
	coarse resolution (top) and two levels of splitting (bottom)	98

5.20	Nomenclature for the 2-D sloshing problem	99
5.21	Snapshots of velocity magnitude.	101
5.22	Time history of the free-surface elevation at $x(t) = 0.05 + A \cos\left(\frac{2\pi t}{T}\right)$	
	during the first ~ 9 oscillation cycles	102
6.1	Sketch of a SPH channel flow with the implemented I/O model. $\ . \ .$.	105
6.2	Contiguous array notation for the insertion/removal process of particles	
	in DualSPHysics	106
6.3	Schematics of the 2-D jet impingement problem (Lokman Hosain, 2013)	.108
6.4	Time snapshots of velocity contours for a jet speed $U_{\infty} = 20$ m/s	109
6.5	Normalized pressure for $U_{\infty} = 20 \text{ m/s.}$	110
6.6	Normalized pressure for $U_{\infty} = 30 \text{ m/s.}$	110
6.7	Normalized pressure for $U_{\infty} = 40$ m/s	110
6.8	Comparison of the pressure coefficient for $U_{\infty} = 20$ m/s	111
6.9	Numerical domain for the 2-D channel flow past a cylinder	112
6.10	Close-up contours of nondimensional velocity (top) and vorticity (bot-	
	tom) for $Re = 20. \dots \dots$	114
6.11	Vectorial representation of velocity field around the stagnation region.	115
6.12	Streamlines for $Re = 20. \dots \dots$	115
6.13	Close-up contours of nondimensional velocity (top) and vorticity (bot-	
	tom) for $Re = 50$	116
6.14	Streamlines for $Re = 50.$	117
6.15	Close-up contours of nondimensional velocity (top) and vorticity (bot-	
	tom) for $Re = 100.$	118
6.16	Close-up contours of nondimensional velocity (top) and vorticity (bot-	
	tom) for $Re = 200$	119
6.17	Close-up contours of nondimensional velocity (top) and vorticity (bot-	
	tom) for $Re = 500$	120
6.18	Streamlines for $Re = 100.$	121
6.19	Streamlines for $Re = 200$	121
6.20	Streamlines for $Re = 500$	121
6.21	Pressure coefficient along the plate.	122
6.22	3-D contours of normalized velocity.	123
6.23	Free-surface profile along the positive XZ-plane for $U_{\infty} = 20$ m/s	124
6.24	Velocity contours for the inclined plate case.	125
6.25	Pressure contours for the inclined plate case	125

6.26	3-D visualization of the wake behind the cube for $\text{Re} = 30$ (top) and	
	$Re = 300 \text{ (bottom)}. \dots \dots$	126
6.27	Vorticity contours along the (XY) , $(XY - 1.5L)$, $(XY + 1.5L)$ center-	
	planes for $Re = 30.$	127
6.28	Vorticity contours along the (XZ) , $(XZ - 1.5L)$, $(XZ + 1.5L)$ center-	
	planes for $Re = 30.$	128
6.29	Vorticity contours along the (YZ) , $(YZ - 1.5L)$, $(YZ + 1.5L)$ center-	
	planes for $Re = 30.$	129
6.30	Vorticity contours along the (XY) , $(XY - 1.5L)$, $(XY + 1.5L)$ center-	
	planes for $Re = 300$	130
6.31	Vorticity contours along the (XZ) , $(XZ - 1.5L)$, $(XZ + 1.5L)$ center-	
	planes for $Re = 300$	131
6.32	Vorticity contours along the (YZ) , $(YZ - 1.5L)$, $(YZ + 1.5L)$ center-	
	planes for $Re = 300$	132

List of Tables

2.1	Specifications of the superclocked eVGA GeForce GTX TITAN card.	26
3.1	Simulation parameters	33
3.2	Comparison of the low pressure wake half-angle, β , obtained from	
	RANS solutions and SPH simulations.	46
4.1	Simulation parameters	52
4.2	Comparison of the proposed hydrodynamic function in Equation (4.11)	
	with that of Aureli et al. (2012) in Equation (4.6)	65
5.1	Smoothing ratio, α , and relative distance, ε , of daughter particles found	
	by Vacondio et al. (2015). \ldots	78
5.2	Properties of mother and daughter particles (Feldman and Bonet, 2007).	79
5.3	Simulation details for the dam break problem	98
6.1	Strouhal number for $Re = 100, 200$ and 500	118
6.2	Runtime specifications for $10 \ [s]$ of simulated physical time on a 12-core	
	CPU	133

Chapter 1

Introduction

1.1 Approaches in Computational Fluid Dynamics

The physics of a fluid such as air or water is often described through the use of continuum mechanics theory, where physical properties are aggregated over an enormous number of individual molecules. Two different mathematical representations are usually adopted in the description of a fluid flow, i.e. the Eulerian and the Lagrangian pictures. In the former, a control volume is fixed in space and flow variables like density, velocity or pressure are written as spatio-temporal functions through which the evolution of the fluid is determined within the control volume. On the contrary, the Lagrangian approach is a fixed-mass representation of the physical domain, where the fluid flow is analyzed by tracking the properties of fluid blobs or "particles" as they evolve in space and time. In both cases, closed-form solutions obtained through analytical modeling are certainly appealing because they are indeed exact solutions of the partial differential equations (PDEs) governing the motion of the fluid. Nevertheless, a limited number of problems can be solved with an analytical approach, for example PDEs with relatively simple coefficients or very smooth boundary conditions. Over the past few decades, numerical methods for the solution of partial differential equations of mathematical physics have received surging interest as a solid alternative to analytical methods. The increasing availability of computer memory and power at reasonable costs has enabled the design of highly accurate and efficient methods in the branch of Computational Fluid Dynamics (CFD), allowing to solve three-dimensional fluid problems that were once thought untreatable.

A large portion of the numerical algorithms used to simulate fluid flows is based on the Eulerian approach, since these techniques have been studied extensively for many decades and are indeed well understood. For example, most of the commercial



(a) Mesh-based discretization. (b) Mesh-less discretization.

Figure 1.1: Representation of a generic domain with and without the use of a mesh.

codes like ANSYS[®] FLUENT[®], STAR-CCM+[®] or ADINA[®] have been developed using finite volume methods (FVM), finite element methods (FEM) or finite difference methods (FDM). These are usually a better choice for flows involving moderate gradients or fixed boundaries, whereas they are not especially suited for problems involving, for example, large deformations or complex free-surface flows, for which particle methods seem a better option. Admittedly, particle methods are potentially stable and robust for problems with many kinds of boundary and flow conditions, and therefore they are considered the next generation of computational methods (Liu and Liu, 2010). This and other important aspects that will be described later in this thesis constitute the central reason for the choice of particle methods over traditional grid-based methodologies.

Generally, computer simulations follow a common procedure to resolve a particular problem and one of the critical steps in this process is the discretization of the physical domain. Depending on whether one chooses a mesh-based or a mesh-less method, the computational space can be represented by connected nodes in a predefined manner (grid/mesh) or arbitrarily distributed nodes (particles). This is not always a net distinction since, for example, some particle techniques such as the Lattice Boltzmann method (LBM) still require a grid for the computation (Chen et al., 2011). At the present moment, there are many particle methodologies available, each with its advantages and disadvantages, which make them well suited for a specific type of study. Herein, a purely Lagrangian particle method named Smoothed Particle Hydrodynamics (SPH) is chosen as the focus of the research work.

The choice of SPH as a viable option for the simulation of free-surface flows and

other hydrodynamics problems is mainly due to its versatility and simple approach to numerical fluid dynamics. The ease with which this methodology can provide a large dynamical range in spatial resolution and density, as well as an automatically adaptive resolution, are unmatched in Eulerian methods. Moreover, SPH has excellent conservation properties, not only for energy and linear momentum, but also for angular momentum. The latter is not guaranteed in Eulerian codes, although it is generally maintained at an acceptable level for well-resolved flows. When coupled to self-gravity, SPH conserves the total energy exactly, which is again not necessarily true in most grid-based approaches. Finally, SPH is Galilean-invariant and free of any errors from advection alone, which is another advantage compared to Eulerian mesh-based approaches (Springel and Dullemond, 2011). The complete absence of a grid allows SPH to easily deal with complicated geometric settings and large regions of space characterized by low particle resolution. Implementing SPH equations in a numerical code is generally simple, although the resulting algorithms are still very robust. As an example, negative densities can not occur in SPH by construction, whereas this represents a problem in some mesh-based codes. On the other hand, shock wave-fronts are smeared by SPH, even though the properties of the post-shock flow are correct. The main disadvantage of SPH is its limited accuracy in multidimensional flows. Errors arise in the approximation of local kernel interpolants through discrete sums over a small set of nearest neighbours. While in 1-D this error can be neglected, particle motion in multiple dimensions has a much higher degree of freedom. Here the mutually repulsive forces of pressurized neighbouring particle pairs do not easily cancel in all dimensions simultaneously, especially not given the errors of the discretized kernel interpolants. As a result, some issues in the particles motion can readily develop and give rise to velocity noise up to a few percent of the local speed of sound. This noise seriously influences the accuracy that can be reached with the technique, especially for subsonic flow, and also leads to a slow convergence rate. Particularly problematic in SPH are fluid instabilities across contact discontinuities, such as Kelvin-Helmholtz instabilities (Springel and Dullemond, 2011). These are usually found to be suppressed in their growth. Another generic problem is that the numerical dissipation introduced to stabilize the method in presence of shock waves or other significant density gradients operates at some level also outside of shocks, leading to relatively high numerical viscosity that limits the Reynolds numbers that can be easily reached with SPH in the absence of correction techniques.

1.2 Research purpose and contributions

The aim of the proposed research is to find a suitable methodology for the solution of non-linear free surface flows and a variety of other problems in hydrodynamics. To this extent, a broad literature research is conducted to identify pros and cons of several CFD techniques, leading to the choice of Smoothed Particle Hydrodynamics to tackle the research objectives. This method is primarily chosen because of the fairly simple form of its algorithms and many other attractive features that will be explained in detail in the next chapter. Moreover, traditional CFD techniques have been around for over a century, whereas SPH is less than 40 years old, and no application in the field of free-surface hydrodynamics using this approach is noted until the mid 1990s. The relatively young age of SPH has thus played a role in the choice of a suitable CFD method as it presumed many viable research directions, from novel applications to method improvements.

Major research contributions in terms of novel applications include an assessment of the relation between hydrodynamic pressure and surface waves generated by fast boats. This problem has received particular attention in recent time as the presence of high-speed vessels becomes more and more significant in areas of finite-depth water, such as harbors and shorelines. The ability of predicting pressure signatures at the seafloor or wave fields produced by these boats can serve different purposes, including military and environmental applications. A thorough study presented herein has addressed this topic, showing a clear correlation between water waves and pressure disturbances at the seafloor. This dependence is quantified mathematically by relating the shape of the disturbances with the parameters governing the problem.

A second novel implementation of SPH focuses on modeling the hydrodynamics of oscillating rigid plates in a viscous fluid with different boundary conditions. As will be seen Chapter 4, this subject relates directly to many areas of engineering, such as underwater vibrations or bio-inspired locomotion, and no similar attempts of using a SPH approach for this problem have been recognized in the literature to the best of the author knowledge. Results of the simulations are validated against numerical and experimental works of similar nature, suggesting that SPH is fully capable of modeling vortex formation, shedding, and advection caused by the oscillating lamina in a variety of flow conditions.

The extensive amount of simulations carried out in the present research work has permitted to identify some of the recurrent shortcomings of Smoothed Particle Hydrodynamics, both intrinsic to the method and relative to its implementation in DualSPHysics (Crespo et al., 2015), the code package herein chosen for computations. Despite SPH being considered one of the most efficient and robust among particles methods, there are indeed a number of grand challenges that still need to be addressed. The SPHERIC international organization has been constituted in the early 2000s to connect SPH practitioners around the world and to group possible research directions under four main branches: convergence, numerical stability, boundary conditions, and adaptivity. Indeed, two of these have been chosen as research objectives in this dissertation.

A particle and coalescing scheme (Vacondio et al., 2013) currently ongoing implementation in DualSPHysics has been tested for some free-surface flow problems chosen by the author to assess the capabilities of variable resolution SPH. Results from the simulations validate the effectiveness of adaptivity as a means to resolve fluid flows accurately with reasonable computational times. Furthermore, a novel algorithm that introduces inflow and outflow regions in the computational domain has been developed to simulate open boundaries with DualSPHysics. Several test cases for both two and three dimensions suggest the validity of the implemented algorithm, and furnish further directions for improvement.

Both variable resolution and open boundary conditions push the applicability of SPH towards engineering problems of higher complexity.

1.3 Thesis outline

In the next chapter, a brief overview of the Smoothed Particle Hydrodynamics method is presented. Important aspects such as integral and particle approximations are discussed, including the derivation of the SPH conservation equations and the analogy with the continuity and Navier-Stokes equations for a weakly-compressible fluid. Several numerical concepts are elaborated, including stability, consistency, accuracy, and convergence of the method. A few models of SPH viscosity and boundary conditions are then introduced together with information about the hardware and software herein used for the simulations.

Chapter 3 focuses the attention on SPH solutions of the wave and bottom pressure fields generated by a fast hull in finite-depth water in a variety of flow conditions. Two viscosity models are employed and their effects are commented. Several contours are presented, highlighting a strong dependence of the hydrodynamic pressure on the behavior of surface waves. A functional relation is developed and validated to quantify this correlation. Harmonic oscillations of a thin rigid lamina in a viscous fluid with and without a free surface are studied next in Chapter 4. For both cases, SPH flow contours are provided in a broad range of the non-dimensional parameters governing the problem, namely the frequency and amplitude of oscillations and the depth of submergence for the free surface case. A novel hydrodynamic function is cast following the approach in similar previous works, with added mass and damping coefficients obtained and commented.

In Chapter 5, the topic of variable resolution SPH is discussed together with the algorithm strategy chosen by the DualSPHysics developers to achieve adaptivity. Solutions of a dam break and sloshing problems are presented to test the variable resolution algorithm. Both cases demonstrate that adaptive SPH successfully increases the simulation accuracy in specific areas of the numerical domain while gaining significant reductions in computational time with respect to high-resolution flows with uniformly distributed particles.

Chapter 6 is concerned with the concept of boundary conditions in SPH as one of the big challenges of this methodology. Particularly, a novel algorithm is presented for the treatment of inlets and outlets of the computational domain through the use of buffers made of SPH particles. In these regions, it is possible to assign desired flow properties or extrapolate them from the domain interior to simulate a variety of inflow and outflow conditions. Simulations are then presented to validate the algorithm in 2-D and 3-D, with good agreement of the results with several other works in the literature.

Finally, Chapter 7 summarizes the achievements of the research work and provides suggestions for future research directions.

Chapter 2

Smoothed Particle Hydrodynamics

Smoothed Particle Hydrodynamics is a Lagrangian method of probabilistic nature formulated nearly forty years ago by Gingold and Monaghan (1977) and Lucy (1977) to study problems of astrophysical gas dynamics. During the last three decades, the method has been modified for deterministic use in the field of hydrodynamics, with the first free-surface works being published in the mid 1990s (Monaghan, 1994, 1996). Currently, SPH represents one of the most robust among particle methods for numerical hydrodynamics and therefore it is chosen as the main methodology in this thesis. In the following sections, a brief overview of SPH fundamentals is presented together with the description of the main algorithms in DualSPHysics, the open-source code adopted for the simulations in this work (Crespo et al., 2015; Gómez-Gesteira et al., 2012a,b).

2.1 SPH formalism

2.1.1 Integral approximation

The types of problems addressed with SPH usually involve finding an approximate solution of the Partial Differential Equations (PDEs) of mathematical physics for which no analytical treatment is possible. Numerical results consist of a bunch of scalar and vector functions (e.g., strain, velocity, pressure, etc.) describing the space and time evolution of the medium under investigation.

The first concept towards formulating a set of conservation equations for the SPH method is the integral representation of a field function, $f(\mathbf{x})$. Starting from the identity

$$f(\mathbf{x}) = \int f(\mathbf{x}')\delta(\mathbf{x} - \mathbf{x}') \, d\mathbf{x}' \tag{2.1}$$

valid for any $f(\mathbf{x})$, it is possible to replace the Dirac delta function, $\delta(\mathbf{x} - \mathbf{x}')$, with a compactly supported interpolant, $W(\mathbf{x} - \mathbf{x}', h)$, usually called *kernel* or *smoothing* function

$$\langle f(\mathbf{x}) \rangle \triangleq \int_{\Omega} f(\mathbf{x}') W(\mathbf{x} - \mathbf{x}', h) \, d\mathbf{x}' + \mathcal{O}(h^2)$$
 (2.2)

Equation (2.2) takes the name of *kernel approximation* of $f(\mathbf{x})$, and is marked by angular brackets. The parameter h governs the size of the kernel support, Ω , and is named smoothing length. It will be further clarified later in this chapter together with some mathematical conditions that the kernel must satisfy to guarantee consistency and $\mathcal{O}(h^2)$ accuracy.

The kernel approximation also applies to the gradient of $f(\mathbf{x})$

$$\langle \nabla f(\mathbf{x}) \rangle = \int_{\Omega} \nabla f(\mathbf{x}') W(\mathbf{x} - \mathbf{x}', h) \, d\mathbf{x}'$$
 (2.3)

which upon integration by parts becomes

$$\langle \nabla f(\mathbf{x}) \rangle = \int_{\partial \Omega} f(\mathbf{x}') W(\mathbf{x} - \mathbf{x}', h) \cdot \mathbf{n} \, dS - \int_{\Omega} f(\mathbf{x}') \nabla W(\mathbf{x} - \mathbf{x}', h) \, d\mathbf{x}' \tag{2.4}$$

The first integral is performed over the boundary $\partial\Omega$ of the kernel support, with **n** being the normal to the boundary surface. Remarkably, the computation of $\nabla f(\mathbf{x})$ through Equation (2.4) is obtained by simply knowing $f(\mathbf{x})$ and the gradient of the smoothing function. The same procedure can be repeated for higher order derivatives, as shown in more details by Liu and Liu (2003) and Liu et al. (2003). To simplify the notation, the use of angular brackets will be hereinafter omitted and considered understood.

2.1.2 Particle approximation

In the SPH framework, the computational domain is discretized by Lagrangian particles. These constitute mesh-less nodes with material properties, such as mass, velocity or pressure, that can be used to study the macroscopic evolution of the flow. Furthermore, they represent targets in the discrete form of the kernel approximations of Equations (2.2) and (2.3) as their contribution to this step becomes relevant whenever they are located within the kernel support. A second important approximation thus occurs when transitioning from the physical domain, where continuum relations hold, to the particle domain, where Equation (2.2) becomes

$$f(\mathbf{x}_k) = \sum_{l=1}^{N} \frac{m_l}{\rho(\mathbf{x}_l)} f(\mathbf{x}_l) W_{k,l}$$
(2.5)

Here \mathbf{x}_k is the position vector of the k-th target particle, \mathbf{x}_l refers to the l-th particle located within the kernel support centered at the target particle, N is the total number of particles inside the support domain, m_l and $\rho(\mathbf{x}_l)$ are the mass and density of the interpolating particle, and $W_{k,l} = W(\mathbf{x}_k - \mathbf{x}_l, h)$. A simple substitution of $f(\mathbf{x}_k)$ with the density function, $\rho(\mathbf{x}_k)$, in Equation (2.5) leads to the SPH density estimate

$$\rho_k = \sum_{l=1}^N m_l W_{k,l} \tag{2.6}$$

where the notation $\rho_k = \rho(\mathbf{x}_k)$ is henceforth adopted. It is remarked that the goodness of the approximations in Equations (2.5) and (2.6) is affected by the number of particles contributing to the estimates, a problem referred to as particle inconsistency and discussed in Section 2.3.4.

2.1.3 The smoothing function

The role of the smoothing function in SPH is important in that it determines the accuracy of the particle approximation in Equation (2.5) and also the computational efficiency. The former is assessed by expanding f_k in Equation (2.5) in a Taylor series about \mathbf{x}_k , thus obtaining

$$f_k = f_k \sum_{l=1}^N \frac{m_l}{\rho_l} W_{k,l} + \nabla f_k \cdot \sum_{l=1}^N \frac{m_l}{\rho_l} (\mathbf{x}_l - \mathbf{x}_k) W_{k,l} + \mathcal{O}(h^2)$$
(2.7)

As will be shown momentarily, the $\mathcal{O}(h^2)$ accuracy is the highest order of precision that can be obtained in the integral approximation of a function with SPH. From Equation (2.7) it can be seen that this order of accuracy is maintained only in presence of kernels satisfying a normalization condition

$$\sum_{l=1}^{N} \frac{m_l}{\rho_l} W_{k,l} \approx 1 \tag{2.8}$$

and a symmetry condition (even function)

$$\sum_{l=1}^{N} \frac{m_l}{\rho_l} (\mathbf{x}_l - \mathbf{x}_k) W_{k,l} \approx 0$$
(2.9)

Equations (2.8) and (2.9) identify the first two constraints in the choice of suitable smoothing functions. Other two key properties are given by

$$\lim_{h \to 0} W(\mathbf{x} - \mathbf{x}', h) = \delta(\mathbf{x} - \mathbf{x}')$$
(2.10)

$$W(\mathbf{x} - \mathbf{x}', h) = 0, \quad \text{for } |\mathbf{x} - \mathbf{x}'| > \lambda h, \ \lambda \in \mathbb{R}^+, \ \lambda \ge 1$$
(2.11)

whereas Liu and Liu (2003) remark the importance of positive and sufficiently smooth kernels for obtaining physically meaningful results and less sensitivity to disordered particle distributions.

In choosing the smoothing length, Price (2012) suggests that h should relate to the local number density of particles, $n(\mathbf{x}_k)$, defined as

$$n_k = \sum_{l=1}^{N} W_{k,l}(h_k)$$
 (2.12)

with a $1/\sqrt[d]{n_k}$ dependence, where d is the number of dimensions of the simulated problem. For particles with equal masses, this translates into a direct proportionality between $h(\mathbf{x})$ and $\rho(\mathbf{x})$ that must be taken into account when solving for the density equation. Several smoothing functions have been proposed, including the Gaussian, bell-shaped functions, B-splines, quintic splines, etc. and a few examples are shown in Figure 2.1, with their first and second derivatives as a function of the distance from the center of the kernel, therein referred to as q = r/h.



Figure 2.1: Some common kernel functions (solid curves) with their first (red dashed curves) and second (green dashed curves) derivatives and comparison with the Gaussian kernel (dotted curves) (Price, 2012).

As an example, the Wendland kernel (Wendland, 1995) is reported

$$W(\mathbf{x} - \mathbf{x}') = \sigma \left(1 + 2\frac{|\mathbf{x} - \mathbf{x}'|}{h} \right) \left(1 - \frac{1}{2}\frac{|\mathbf{x} - \mathbf{x}'|}{h} \right)^4 \quad 0 \le |\mathbf{x} - \mathbf{x}'| \le b, \ b \in \mathbb{R}^+, \ b \ge 1$$
(2.13)

as it will be a recurrent choice in this thesis due to its simplicity and stable behavior with simulations involving high number of particles (Dehnen and Aly, 2012). The coefficient σ is a function of the spatial dimensions of the problem, and is obtained after normalization of the kernel in one-, two-, and three-dimensions. Further details on other types of kernels and the process of building a smoothing function can be found in Liu et al. (2003); Liu and Liu (2003, 2010); Price (2012).

2.2 Governing equations

2.2.1 Incompressible vs Weakly-Compressible SPH

Generally, there are two main approaches in CFD that one can follow for the solution of incompressible flows. The distinction arises in the treatment of the pressure term, which appears only in one of the conservation equations, i.e. the Navier-Stokes equations. The first option is to use the so-called pressure-based methods, where governing equations are dealt with separately and a Poisson problem or other correction techniques are carried out for computing the pressure field, see for example Bentson and Vradis (1990); Chorin (1968); Patankar and Spalding (1972). A second group of algorithms (Chorin, 1967) exploits an artificial compressibility term in the continuity equation to obtain a well-posed time-marching framework for incompressible cases, paving the way to preconditioning algorithms to control and optimize timeiterative processes (Turkel, 1999; van Leer et al., 1991). This distinction is made in SPH as well. Incompressible SPH (ISPH) is a pressure-based formulation (Cummins and Rudman, 1999; Lee et al., 2008; Shao and Lo, 2003), where the projection of the standard SPH velocity field onto a divergence-free space is used to enforce incompressibility and the solution of a Poisson equation is derived from an approximate pressure projection. Conversely, Weakly-Compressible SPH (WCSPH) employs the concept of artificial compressibility, where small variations in the fluid density are allowed and the derivative of the density function is retained in the continuity equation. The pressure is then coupled with the density through an ad hoc equation of state, whose parameters are chosen to reproduce the physical behavior of the simulated fluid.

In this work the interest is in using SPH to simulate incompressible free-surface flows in the field of ocean engineering, where the characteristic Mach numbers are very moderate and compressibility effects are small. In light of this, it is reasonable to assume a barotropic fluid, i.e. a fluid for which the pressure is a function of the density only and changes in the internal energy can thus be neglected. Although both approaches above are viable, the choice of WCSPH is preferred because of its simplicity over the more complicated ISPH, where the computational effort for solving the pressure equation can become considerable. The continuity and Navier-Stokes equations in Lagrangian form for a weakly-compressible fluid are

$$\frac{\mathrm{D}\rho}{\mathrm{D}t} + \rho\nabla\cdot\mathbf{u} = 0 \tag{2.14}$$

$$\frac{\mathrm{D}\mathbf{u}}{\mathrm{D}t} = -\frac{1}{\rho}\nabla P + \mathbf{g} + \frac{1}{\rho}\nabla\cdot\tau \qquad(2.15)$$

where D/Dt is the total derivative, $\nabla \cdot$ is the divergence operator, ∇ is the gradient operator, **u** is the velocity vector, P is the pressure, ρ is the fluid density, **g** is gravity and τ is the deviatoric component of the total stress tensor, given by

$$\tau = \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) \tag{2.16}$$

As can be seen from Equations (2.14) and (2.15), the main difference with a set of truly incompressible equations is the presence of the density derivative in Equation (2.14). Moreover, a third equation in the form $P = P(\rho)$ will be introduced to relate pressure and density.

2.2.2 Derivation of the SPH conservation equations

Continuity equation

The derivation of the continuity equation in Smoothed Particle Hydrodynamics is the direct result of differentiation of the density estimate in Equation (2.6) with respect to time

$$\frac{d\rho_k}{dt} = \sum_{l=1}^{N} m_l (\mathbf{u}_k - \mathbf{u}_l) \nabla_k W_{k,l}$$
(2.17)

Since SPH particles constitute a fixed-mass system, it can be inferred that Equation (2.6) is an exact solution of the SPH continuity equation in the particular form above (Price, 2012). Equation (2.17) is not the only possible formalism of mass conservation in SPH (Liu and Liu, 2003), however it corresponds to the one implemented in DualSPHysics. The equivalence with the continuum form of Equation (2.14) can be seen by considering the particle approximation of the divergence of f_k

$$\nabla \cdot f_k = \sum_{l=1}^N m_l \frac{f_k}{\rho_l} \nabla_k W_{k,l}$$
(2.18)

Applying this relation to the quantity $f_k = \rho_l(\mathbf{u}_k - \mathbf{u}_l)$ in Equation (2.17) yields

$$\frac{d\rho_k}{dt} = \mathbf{u}_k \cdot \sum_{l=1}^N m_l \nabla_k W_{k,l} - \sum_{l=1}^N m_l \mathbf{u}_l \cdot \nabla_k W_{k,l} \approx \mathbf{u} \cdot \nabla \rho - \nabla \cdot (\rho \mathbf{u}) = -\rho(\nabla \cdot \mathbf{u}) \quad (2.19)$$

Momentum equation

For the derivation of the momentum equation, the Lagrangian for a system of N particles is firstly introduced

$$L = \sum_{l=1}^{N} m_l \left[\frac{1}{2} \mathbf{u}_l^2 - e_l(\rho_l, s_l) \right]$$
(2.20)

with e_l representing the internal energy per unit mass. The choice of the Lagrangian as a starting point is advantageous as it confers symmetric and conservative properties to the derived equation. Following Price (2012), the least action principle is applied to L and yields the Euler-Lagrange equations for a particle k

$$\frac{d}{dt} \left(\frac{\partial L}{\partial \mathbf{u}_k} \right) - \frac{\partial L}{\partial \mathbf{x}_k} = 0 \tag{2.21}$$

with

$$\frac{\partial L}{\partial \mathbf{u}_k} = m_k \mathbf{u}_k \tag{2.22}$$

$$\frac{\partial L}{\partial \mathbf{x}_k} = -\sum_{l=1}^N m_l \frac{\partial e_l}{\partial \rho_l} \bigg|_s \frac{\partial \rho_l}{\partial \mathbf{x}_k}$$
(2.23)

Equation (2.23) is valid for constant entropy, s. This is reasonable since a conservative form of the governing equations is being derived, hence no dissipative terms are considered at this point. Using the first principle of thermodynamics for constant entropy

$$\left. \frac{\partial e_l}{\partial \rho_l} \right|_s = \frac{P}{\rho^2} \tag{2.24}$$

and rewriting the density gradient, $\partial \rho_l / \partial \mathbf{x}_k$, in Equation (2.23) as

$$\frac{\partial \rho_l}{\partial \mathbf{x}_k} = \left[1 - \frac{\partial h_l}{\partial \rho_l} \sum_{p=1}^N m_p \frac{\partial W_{l,p}(h_l)}{\partial h(\mathbf{x}_l)}\right]^{-1} \sum_{p=1}^N m_p \frac{\partial W_{l,p}(h_l)}{\partial \mathbf{x}_k} (\delta_{l,k} - \delta_{p,k})$$
(2.25)

it is possible to combine Equations (2.22) to (2.25) and to obtain a new form of Equation (2.21) for a constant smoothing length

$$\frac{d\mathbf{u}_k}{dt} = -\sum_{l=1}^N m_l \left(\frac{P_l}{\rho_l^2} + \frac{P_k}{\rho_k^2}\right) \cdot \nabla_k W_{k,l}$$
(2.26)

Equation (2.26) represents the SPH momentum equation in its simplest form. No external force field is thus considered and methods for including the effects of viscosity and body forces will be discussed later. As for the continuity equation, an equivalent with the continuum case in Equation (2.15) and the discrete case above can be made by considering

$$\frac{d\mathbf{u}_k}{dt} = -\sum_{l=1}^N m_l \left(\frac{P_l}{\rho_l^2} + \frac{P_k}{\rho_k^2}\right) \cdot \nabla_k W_{k,l} \approx -\frac{P}{\rho^2} \nabla \rho - \nabla \left(\frac{P}{\rho}\right) = -\frac{1}{\rho} \nabla P \qquad (2.27)$$

Although not considered in this work, several formulations of the energy equation are also derived in a similar fashion, with a few examples reported in Liu and Liu (2003); Price (2012); Violeau (2012).

Pressure equation

From Equation (2.26), it is evident that the pressure at the target particle, P_k , and its neighbors, P_l , is required to solve for the velocity. Among the several equations of state available for WCSPH the most commonly used is

$$P_k = \varphi(\kappa_k^\gamma - 1) \tag{2.28}$$

with $\kappa_k = \rho_k/\rho_0$, γ being a coefficient and ρ_0 being the fluid reference density. Works presented later in this thesis adopt the values $\gamma = 7$ and $\rho_0 = 1,000$ kg m⁻³, since these are commonly accepted as optimal for water simulations with WCSPH algorithms (Gómez-Gesteira et al., 2012a). The coefficient φ controls the density variations within the limits imposed by stability criteria presented in Section 2.3.2. An expression for φ is given by

$$\varphi = \frac{\rho_0 c_s^2}{\gamma} \tag{2.29}$$

where c_s is the speed of sound. From thermodynamics

$$M^2 = \frac{\mid \mathbf{u}_c \mid^2}{c_s^2} \sim \frac{\mid \Delta \rho \mid}{\rho_0}$$
(2.30)

where M is the Mach number and \mathbf{u}_c is the flow characteristic velocity. In the presence of liquids like water, the use of the true speed of sound would yield very high values of φ and this would result in excessively small time steps, as will be shown in Section 2.3.2. Therefore, a numerical value of $c_s \sim 10 | \mathbf{u}_c |$ is commonly accepted in SPH since it allows reasonable time stepping while limiting the relative density variations to approximately 1% of the reference density.

Among the advantages of solving for the pressure field with an algebraic equation is that the resulting system of conservation equations can be solved explicitly, favoring an easy algorithm implementation and parallelization (Ferrari et al., 2009). On the other hand, an explicit system of equations such as Equations (2.17), (2.26) and (2.28) that uses centered operators is unconditionally unstable and therefore numerical diffusion is introduced to restore stability in SPH. The artificial viscosity is one of the classical methods used to address this issue and will be discussed in the next section.

2.3 Numerical aspects of the SPH methodology

2.3.1 Time integration

Equations (2.17) and (2.26) represent the SPH conservation equations for an incompressible inviscid fluid. Coupled with Equation (2.28), they constitute a system of ordinary differential equations (ODEs) to be solved numerically through standard time-integration techniques. Using a more compact notation yields

$$\begin{cases} \frac{\mathrm{d}\rho_k}{\mathrm{d}t} = \varrho_k \\ \frac{\mathrm{d}\mathbf{u}_k}{\mathrm{d}t} = \mathbf{F}_k \\ \frac{\mathrm{d}\mathbf{x}_k}{\mathrm{d}t} = \mathbf{U}_k \end{cases}$$
(2.31)

where the quantities ρ_k and \mathbf{F}_k symbolize the right-hand side of Equations (2.17) and (2.26), respectively, and the third equation describes the time evolution of the position vector, \mathbf{x}_k , as a function of the velocity vector, \mathbf{U}_k in compact form. The coupled system (2.31) can be solved by numerous techniques, some of which are described in details in Issa (2004) and Gómez-Gesteira et al. (2012b).

As an example, a symplectic predictor-corrector scheme with second order accuracy in time is hereby described due to its frequent application in the simulations. During the predictor step, the value of the acceleration, $d\mathbf{u}_k/dt$ is initially estimated at the mid-time step according to

$$\mathbf{x}_{k}^{n+\frac{1}{2}} = \mathbf{x}_{k}^{n} + \frac{\Delta t}{2} \mathbf{U}_{k}^{n}$$

$$(2.32)$$

$$\rho_k^{n+\frac{1}{2}} = \rho_k^n + \frac{\Delta t}{2} \varrho_k^n \tag{2.33}$$

with the superscript n indicating quantities at the generic $t = n\Delta t$. The corrector step consists in calculating velocity and hence position of the k-th particle using the estimated $d\mathbf{u}_k^{n+\frac{1}{2}}/dt$. This yields

$$\mathbf{u}_{k}^{n+1} = \mathbf{u}_{k}^{n+\frac{1}{2}} + \frac{\Delta t}{2} \mathbf{F}_{k}^{n+1}$$
(2.34)

$$\mathbf{x}_{k}^{n+1} = \mathbf{x}_{k}^{n+\frac{1}{2}} + \frac{\Delta t}{2} \mathbf{U}_{k}^{n+1}$$
(2.35)

Ultimately, $d\rho_k^{n+1}/dt$ is calculated at the end of the time step using \mathbf{u}_k^{n+1} and \mathbf{x}_k^{n+1} .

2.3.2 Stability

The primary reason for the need of a viscosity model in SPH is numerical. The time discretization of Equations (2.17) and (2.26), which in their form represent the Euler equations, yields an unconditionally unstable scheme. One of the first approaches to deal with this problem is to introduce an artificial viscosity term in Equation (2.26), with the initial purpose of treating shock discontinuities in high-speed flows (Monaghan and Gingold, 1983; Monaghan, 1985). This has the form

$$T_{k,l} = \begin{cases} \frac{-\alpha \ \bar{c}_{sk,l} \ \mu_{k,l} + \beta \mu_{k,l}^2}{\bar{\rho}_{k,l}} & \mathbf{x}_{k,l} \cdot \mathbf{u}_{k,l} \le 0\\ 0 & \text{otherwise} \end{cases}$$
(2.36)
where $\mu_{k,l} = \frac{h \mathbf{u}_{k,l} \cdot \mathbf{x}_{k,l}}{|\mathbf{x}_{k,l}|^2 + \eta^2}$, and $\eta = 0.1h$ is used to avoid singularities in the denominator. The bar operator denotes the average, e.g. $\bar{c}_{sk,l} = 0.5(c_{s,k} + c_{s,l})$. The first term in Equation (2.36) is linear in $\mathbf{u}_{k,l}$ and causes a bulk viscosity in the flow, which is ideal when modeling viscous fluids. Values in the range [0 - 0.4] are generally used for the coefficient α . The second term is useful in reducing particle interpenetration at high Mach number flows, which are not relevant to this work. Therefore $\beta = 0$ is imposed hereinafter. It is worth noting that since the effect of this viscosity term becomes significant for particles approaching each other, it can cause artificial edge pressures in the presence of a solid boundary, as mentioned in Morris and Monaghan (1997), where a limiter for the artificial viscosity is proposed.

Another diffusive model has been proposed in Molteni and Colagrossi (2009) with the aim of dampening the numerical oscillations affecting the pressure field in WC-SPH when dealing with liquid simulations. This correction is referred to as δ -SPH and is applied to the continuity equation as follows

$$\frac{d\rho_k}{dt} = \sum_{l=1}^N m_l \mathbf{u}_{k,l} \cdot \nabla_k W_{k,l} + 2\delta h \sum_{l=1}^N m_l \bar{c}_{sk,l} \left(\kappa_{k,l} - 1\right) \frac{1}{\mathbf{x}_{k,l} + \eta^2} \cdot \nabla_k W_{k,l} \qquad (2.37)$$

 δ is a parameter tuning the filter intensity, usually in the order of $\delta \approx 10^{-1}$ for simulations involving moderate-to-high flow velocities (Molteni and Colagrossi, 2009; Crespo et al., 2015).

As mentioned earlier, SPH Equations (2.17) and (2.26) are integrated in time explicitly. In this regard, a stability condition is dictated by the forcing terms that appear in the momentum equation. The requirement for the time step is thus

$$\Delta t_1 = \min_k \sqrt{\left(\frac{h}{f_k}\right)} \tag{2.38}$$

where f_k represents any internal or external force associated with the particle k (Monaghan, 1992), e.g. gravity. An additional stability condition is represented by the standard time step requirements of an explicit finite-difference method simulating diffusion. For this, an upper bound to the time step is obtained by taking into account the Courant-Friedrichs-Levy criterion (CFL) and the viscous terms as follows

$$\Delta t_2 = \min_k \frac{h}{c_s + h \max_l \frac{\mathbf{u}_{k,l} \cdot \mathbf{x}_{k,l}}{|\mathbf{x}_{k,l}|^2}}$$
(2.39)

Equation (2.39) highlights the importance of using a numerical speed of sound rather than the physical one to obtain reasonable time steps. Combining Equations (2.38) and (2.39), a global stability constraint is found adaptively at each iteration following

$$\Delta t = \xi \min(\Delta t_1, \Delta t_2) \tag{2.40}$$

where Δt is the time step and ξ is the Courant number, chosen heuristically in the range [0.1 - 0.3].

The SPH method suffers from another type of instability which is not related to the temporal integration. A von Neumann stability analysis carried out by Swegle et al. (1995) shows the presence of an unstable regime for SPH particles subjected to a state of tensile stress. This issue, usually referred to as the tensile instability, is a consequence of both the stress state of the particles and the second derivative of the kernel function, and is therefore intrinsic to the nature of the method rather than to the numerics of the solution scheme. Some techniques for preventing this instability can be found in Dehnen and Aly (2012); Monaghan (2000); Swegle et al. (1995).

2.3.3 Accuracy

As seen in Equations (2.2) and (2.4), the kernel function, W, its gradient, ∇W , and higher order derivatives determine the goodness of the approximation of $f(\mathbf{x})$, $\nabla f(\mathbf{x})$, etc. For this reason, the concept of accuracy of the SPH algorithms is strictly connected to the mathematical properties of the kernel function shown in Section 2.1.3. Liu et al. (2003) argue that the second order accuracy claimed in Equation (2.2) is obtained by expanding $f(\mathbf{x}')$ in a *n*-th order Taylor series about \mathbf{x} and use this to express $f(\mathbf{x}')$ in Equation (2.2). As a result, a series of *n* equations

$$M_0 = \int_{\Omega} W(\mathbf{x} - \mathbf{x}', h) \, d\mathbf{x}' = 1 \tag{2.41}$$

$$M_1 = \int_{\Omega} (\mathbf{x} - \mathbf{x}') W(\mathbf{x} - \mathbf{x}', h) \, d\mathbf{x}' = 0$$
(2.42)

:

$$M_n = \int_{\Omega} (\mathbf{x} - \mathbf{x}')^n W(\mathbf{x} - \mathbf{x}', h) \, d\mathbf{x}' = 0 \qquad (2.43)$$

is obtained, representing the moments of the smoothing function. The satisfaction of Equations (2.41) to (2.43) guarantees a *n*-th order accurate approximation of $f(\mathbf{x})$. Some of the requirements of $W(\mathbf{x} - \mathbf{x}', h)$ in Section 2.1.3 are a direct consequence of the kernel compliance with its moments, see for example the equivalence between Equations (2.8) and (2.41). The simultaneous satisfaction of all moment equations is however not possible because of the positivity of the smoothing function. In other words, while a *n*-th order accuracy is mathematically allowed, physically meaningful results are obtained only with strictly positive kernels, some of which represented in Figure 2.1. This constraint implies that, for n = 2, Equation (2.43) is not always satisfied unless the kernel is identically null. Therefore the accuracy of the kernel interpolation is at most two.

In the transition to the discrete world where particles live, some precision is lost in general, and particularly in areas of the domain where not enough SPH particles are available for interpolations, e.g. domain boundaries or free surface. This topic will be briefly discussed in the following two sections.

2.3.4 Consistency

The idea of consistency in the SPH method is presented in Liu and Liu (2003) in close connection with the kernel requirements seen in the previous section. The set of Equations (2.41) to (2.43) can be easily used to assess this topic by noting that the SPH integral approximation reproduces a k-th order polynomial consistently so long as the first k moments are satisfied. On the other hand, this condition does not guarantee consistency in the discrete counterpart of the equations since the support of the kernel operator is sampled through a finite number of particles, and therefore the goodness of this approximation will strongly depend on the amount of particles located within the kernel width. This is especially important in the presence of a solid boundary or at the free surface, where the average number of particles across the kernel width decreases dramatically and therefore the need of restoring this particle "void" becomes essential.

Many algorithms have been proposed to address this issue, among which the Shepard filter (Shepard, 1968), the Moving Least Squares (MLS) approach (Dilts, 1999) and the Reproducing Kernel Particle Method (RKPM) (Liu et al., 1995). The common objective of these techniques is to increase the number of neighbors, N_n , in the particle approximation to obtain a discrete representation which is as close as possible to the continuum case. The parameter N_n will hence play an important role also in the convergence of the SPH method since, according to the Lax–Richtmyer equivalence theorem, a consistent method is the foundation of a convergent well-posed problem.

2.3.5 Convergence

As previously seen, SPH integral approximations converge towards exact solutions with an error that goes as $\sim h^2$, and this is ensured by the kernel requirements in Equations (2.10) and (2.11). However, when considering the discrete scenario, it is not straightforward to determine the numerical convergence of the method. Colagrossi (2003) has shown that the interpolation error after discretization converges as

$$h^2 + h^{-p} \left(\frac{\Delta x}{h}\right)^2 \tag{2.44}$$

where Δx is the particle spacing and p is the order of the derivative being approximated, e.g. p = 0 in the case of $f(\mathbf{x})$. The form of Equation (2.44) suggests that Δx should decrease faster than h to guarantee that the numerical solution approaches the continuous solution as the total number of SPH particles increases. In practice, however, this is computationally expensive because it translates in a rapidly increasing number of neighbors inside the kernel support as one refines the particle resolution. Hence, as also seen in Section 2.1.3, it is a common practice in SPH to achieve a desired resolution by varying Δx while keeping the ratio $\Delta x/h$ approximately constant. As an example, Springel (2010) follows this approach and presents a number of simulations in astrophysical gas dynamics to assess the rate of convergence of SPH solutions as a function of the total number of particles and the amount of numerical dissipation introduced. In increasing the totality of SPH particles, N, the smoothing length is systematically decreased resulting in a local number of neighbors, N_n , approximately constant. Several values of the viscosity coefficient, α , are also explored. Results therein report the presence of a residual error which is irrespective of N (hence Δx or h) and α , suggesting the necessity of a more strict convergence criterion. In a recent work by Zhu et al. (2015) this issue is addressed by recognizing the existence of a discreteness error for multi-dimensional SPH operators that scales as $\sim N_n^{-1} \log N_n$. For common types of smoothing functions and a sufficiently regular distribution of particles, their findings suggest that a numerical convergence in SPH is possible provided that discrete convolutions are carried out with a local number of neighbors $N_n \sim \sqrt{N}$, therefore with a $h \sim 1/\sqrt[6]{N}$ relation rather than the $h \sim 1/\sqrt[3]{N}$ generally employed. It is worth nothing that the convergence study in Zhu et al. (2015) applies only to multi-dimensional SPH simulations, as the error estimate therein vanishes in presence of 1-D computations.

2.4 Viscosity models

Despite providing a good overall approximation of real viscosity effects in highspeed compressible flows, the artificial viscosity formulation is usually unsuitable for modeling low Mach number problems. Two more appropriate schemes are summarized below as they are implemented in DualSPHysics for simulating the effects of viscous stresses in real liquids.

2.4.1 Laminar formulation

A discretization of the stress tensor in Equation (2.15) for low Reynolds numbers is simply obtained by relating τ with the discrete strain tensor, $\epsilon_{k,l}$, defined as follows

$$\tau_{k,l} = \tau_{l,k} = 2\mu\epsilon_{k,l} = \mu\left(\frac{\partial\mathbf{u}_k}{\partial\mathbf{x}_l} + \frac{\partial\mathbf{u}_l}{\partial\mathbf{x}_k}\right)$$
(2.45)

where μ is the dynamic viscosity. Following the procedure in Shao and Lo (2003) to decompose the partial derivatives and applying incompressibility yields

$$\left(\frac{1}{\rho}\nabla\cdot\tau\right)_{k}\approx\left(\frac{\mu}{\rho}\nabla^{2}\mathbf{u}\right)_{k}=\sum_{l=1}^{N}m_{l}\left(\frac{4\mu\mathbf{x}_{k,l}\cdot\nabla_{k}W_{k,l}}{\rho_{0}(\rho_{k}+\rho_{l})\mathbf{x}_{k,l}^{2}}\right)\mathbf{u}_{k,l}$$
(2.46)

which is a discretized stress term added to Equation (2.26).

2.4.2 Sub Particle Scaling (SPS): an approach to turbulence

A Large-Eddy Simulation (LES)-based algorithm named Sub-Particle Scaling (SPS) is available in DualSPHysics for its better suitability in modeling water waves and high Reynolds number flows in general. In this technique, large eddies are resolved at the inter-particle distance scale, whereas small turbulent eddies are identified through sub-particle scaling with a special averaging methodology. Both laminar and turbulent stress components are hence considered, leading to following form of the SPH momentum equation

$$\frac{D\mathbf{u}_{k}}{Dt} = -\sum_{l=1}^{N} m_{l} \left(\frac{P_{l}}{\rho_{l}^{2}} + \frac{P_{k}}{\rho_{k}^{2}} \right) \nabla_{k} W_{k,l} + \sum_{l=1}^{N} m_{l} \left(\frac{4\mu \mathbf{x}_{k,l} \cdot \nabla_{k} W_{k,l}}{\rho_{0}(\rho_{k} + \rho_{l}) \mathbf{x}_{k,l}^{2}} \right) \mathbf{u}_{k,l} + \sum_{l=1}^{N} m_{l} \left(\frac{\tau_{l}^{*}}{\rho_{l}^{2}} + \frac{\tau_{k}^{*}}{\rho_{k}^{2}} \right) \nabla_{k} W_{k,l} + \mathbf{g}$$
(2.47)

The second term on the right-hand side is responsible for modeling the laminar viscous stresses as seen previously. To include the effect of the motion at the smaller scales, a flat-top spatial filter is applied to both Equations (2.37) and (2.47), using Favre-averaging to avoid the generation of SPS terms in the continuity equation. The shear term, τ^* , arising from the filtering procedure represents the SPS stress tensor, identifying the effects of the motion occurring at scales smaller than the particle spacing. Further details on the formulation are available in Lo and Shao (2002) and Dalrymple and Rogers (2006).

2.5 Boundary conditions

SPH operators and equations derived up to this point are applied throughout the computational domain and therefore operate at the edges of the domain, where certain physical requirements need to be satisfied. To achieve this, special techniques for particle interactions at the interface with a boundary must be designed without compromising the overall accuracy and convergence of the SPH method. The enforcement of boundary conditions in SPH is one of the big challenges of particle methods in general as these do not usually adopt grids, which are instead advantageous in the process of discretizing a given boundary curve or surface. While numerous works address the topic of boundary conditions in SPH depending on the application, this section focuses the attention on the default methodology implemented in DualSPHysics: the dynamic boundary particles (DBPs). These particles are effective in discretizing solid rigid boundaries, either fixed or movable. Support for periodic boundary conditions is also available in the code. A few remarks on the free-surface treatment are then presented later. Finally, a discussion on the subject of open boundaries in SPH for achieving certain inflow and outflow (I/O) conditions is postponed in Chapter 6, where a new algorithm devised by the author for dealing with I/O conditions is introduced and validated.

2.5.1 Dynamic boundary particles

Dynamic boundary particles were firstly used in Dalrymple and Knio (2001) and later in Gómez-Gesteira and Dalrymple (2004) to model the impact of water waves on coastal structures. For a rigid structure, the main idea is to use a set of particles that comply with the same conservation equations of the fluid particles but move according to prescribed conditions on their position or velocity. Assuming momentum conservation in the form of Equation (2.26), this implies that, as the k-th fluid particle approaches a DBP, it will experience a change in momentum due to an increase in density (hence pressure) of the DBP and equal to

$$\frac{du_{r_k}}{dt} = m \left(\frac{P_{DBP}}{\rho_{DBP}^2} + \frac{P_k}{\rho_k^2} \right) \frac{\partial}{\partial r} W_{k,DBP}$$
(2.48)

Here r denotes the direction of the inter-particle distance, and u_{r_k} is the velocity of fluid particle k along r. Combining Equation (2.48) with the continuity equation in the form shown by Monaghan (1996) and Equation (2.28) and considering the action of viscosity and gravity yields a general expression of the force experienced by the fluid particle near a DBP

$$\frac{du_{r_k}}{dt} = -\left(2c_s^2 \frac{W_{k,DBP}}{(W_{k,DBP} + W_0)^2} + mT_{k,DBP}\right)\frac{\partial}{\partial r}W_{k,DBP} - g \qquad (2.49)$$

with $W_0 = W_{k,k} = W_{DBP,DBP} = W_{r_{(k,DBP)}=0}$. A detailed derivation with an expression of du_{r_k}/dt for a Gaussian kernel can be found in Crespo et al. (2007); Crespo (2008).

2.5.2 Free-surface considerations

No explicit mention of how SPH deals with a free-surface has been made so far. In fact, this is one aspect that gained SPH popularity in hydrodynamics because, in general, no particular treatment is needed to enforce free-surface requirements when using the method and this is appealing due to the complexity and non-linearity of free-surface flows. When dealing with free surfaces there are usually a kinematic and a dynamic constraint that must be satisfied. The first is intrinsically accomplished by the Lagrangian formalism that identifies the method. Each SPH fluid element has material properties, hence particles at the free surface at a given time step will remain at the free-surface and move with their velocity, which is the velocity of the free surface. The second requirement is the continuity of the stress tensor along the free surface, which translates in equating the pressure to the external pressure, P_e . If surface tension and air effects are neglected, i.e. a simplified one-phase flow is considered, a trivial change in pressure reference yields $P_e = 0$. This condition is readily imposed to free surface particles by SPH practitioners to satisfy the dynamic requirement. Nevertheless, an important work in Colagrossi et al. (2009) shows that the verification of the dynamic boundary condition is not so simple due to the kernel inconsistency at the domain edges. When using SPH operators to smooth a gradient quantity such as in Equation (2.3), the surface convolution arising from integration by parts in Equation (2.4) generally vanishes for estimates in the domain interior because the integral is evaluated along the edges of the kernel support, $\partial \Omega$, where W is identically null, see Equation (2.11). This is beneficial in that it avoids burdensome surface integrations to maintain $\mathcal{O}(h^2)$ accuracy. As previously stated in Section 2.3, some level of accuracy is lost in the transition to discrete operators, especially when the number of particles in the smoothed estimates is not substantial. More importantly, the kernel gets truncated when operating nearby the domain boundaries (including the free surface). In the evaluation of the pressure gradient term in Equation (2.26), both these effects cause small pressure oscillations in the vicinity of the free surface. Techniques for the enforcement of the dynamic boundary condition have thus been devised, including periodic re-initialization of the density distribution and other mathematical artifices illustrated in Colagrossi (2003); Colagrossi et al. (2009).



Figure 2.2: The DualSPHysics logo.

2.6 The DualSPHysics project

The DualSPHysics project (Crespo et al., 2015) is a set of open-source C++ and CUDA algorithms based on the Smoothed Particle Hydrodynamics implementation provided in the SPHysics code (Gómez-Gesteira et al., 2012a,b). The project is a joint collaboration among the universities of Vigo (Spain), Manchester (United Kingdom), Parma (Italy) and the Centre for Hydrosystems Research (CEHIDRO) in Lisbon (Portugal). As outlined in the DualSPHysics website, the code is constantly under development and many test cases are available to show its robustness and accuracy. Moreover, DualSPHysics features a dual CPU-GPU implementation, allowing simulations on both central processing units (CPU) and graphics processing units (GPU). The advantage of a GPU implementation has allowed the author to run simulations on several high-end graphics cards, such as the eVGA GeForce GTX TITAN with specifications provided in Table 2.1. This has yielded considerable speed-ups with respect to CPU runs, estimated from three to five times slower.

Base Clock	876 MHz
Boost Clock	928 MHz
Memory Clock	6008 MHz Effective
CUDA Cores	2688
Bus Type	PCI-E 3.0
Memory	6144 MB GDDR5
Memory Bit Width	384 Bit
Memory Speed	0.33 ns
Memory Bandwidth	288.38 GB/s

Table 2.1: Specifications of the superclocked eVGA GeForce GTX TITAN card.

Chapter 3

3-D flow field generated by a planing hull in finite-depth water

One of the problems studied in this thesis concerns the investigation of the bottom pressure field and wave elevation generated by a planing hull in finite-depth water. While the existing literature generally addresses these phenomena separately, the work herein proposes a comprehensive comparison between these two hydrodynamic quantities. Both the classical artificial viscosity and the sub-particle scales (SPS) models are implemented in the simulations and their efficacy is assessed. In addition, Reynolds averaged Navier Stokes (RANS) simulations are carried out to validate SPH findings, providing means of comparison between the two methods. The results show a direct correlation between surface wave dynamics and hydrodynamic pressure disturbances at the ocean floor as the value of the Froude number is changed. This correlation is assessed by studying the inverse dependence of the low-pressure wake angle with the Froude number and by comparison of SPH results with RANS simulations and the cited literature.

3.1 Literature review

The hydrodynamics of fast hulls cruising in finite-depth water is a recurring topic in the marine community that has received notable consideration in recent times. One of the driving factors of this rising interest is the ever increasing presence of high-speed vessels (HSV) near the coastlines, which not only affect the surrounding environment (e.g. banks erosion, seafloor disruption, etc.), but also influence the water navigability for nearby craft (Whittaker et al., 2001). Therefore, the accurate prediction of field variables such as hydrodynamic pressure and wave elevation generated by HSV is the current subject of several marine engineering studies, see for example Akimoto (2013); Benassai et al. (2013); Deng et al. (2014).

The study of water waves generated by a moving disturbance on the free-surface is a subject that has been investigated for many decades, with the pioneering work of Lord Kelvin dating back to the late 1800s (Thomson, 1887a,b). In his work, Kelvin had shown that the wake generated by a ship moving in calm water consists of a Vshaped wedge with a wake half-angle of $\sim 19^{\circ}$. Furthermore, he showed that this value remains approximately constant regardless of the ship speed. More recently, Rabaud and Moisy (2013) have presented airborne images of ship wakes with a significant narrower angle than the one predicted by Kelvin, and have proposed an analytical model of this effect that has later been verified by Darmon et al. (2014) and Tunaley (2014b) and reviewed in Rabaud and Moisy (2014). Several other studies have been directed towards understanding the wake generated by HSV, with emphasis on the influence of the water depth (Fang et al., 2011; Tunaley, 2014a), the hull shape (Tuck and Lazauskas, 1998; Tuck et al., 2002) and the interference between the divergent or transverse waves generated by the bow and the stern (Noblesse et al., 2014a,b), illustrated numerically in earlier works of Barnell and Noblesse (1986); Noblesse (1986) and Noblesse and Hendrix (1991).

In addition to free-surface disturbances, the passage of a ship also generates hydrodynamic pressure signals that can reach the coast or the seafloor when the craft is relatively close to them. When displacement hulls such as trawlers or sailing boats are considered, notable pressure disturbances at the ocean bottom consist of a large depression approximately below amidships and two smaller areas of high pressure, one beneath the bow and one beneath the stern. At cruising speeds characteristic of these craft, the intensity of the low-pressure disturbance can reach two to ten times that of the high pressure, depending on the boat speed and water depth (Bielański, 2012, 2014; Lazauskas, 2007). A completely different bottom pressure field is observed for a boat in planing regime: only one high-pressure area with a significantly larger magnitude is now present and located in correspondence of the beam waterline, at the point of contact between the hull and the water. The low-pressure region attains a V-shape that resembles that of water waves previously mentioned, with narrower angles at larger depth Froude numbers, $Fr_h = U/\sqrt{gh}$ (Nguyen and Hyman, 2010; Tafuni and Sahin, 2014). One topic of importance for the safety of ships operating in finite-depth water is the presence of mines on the ocean bottom and their pressureinduced detonation due to the passage of a craft nearby. The neutralization process (mine-sweeping) of such devices consists in utilizing magnetic or acoustic fields to detonate the mine in absence of a target vehicle (Lazauskas, 2007). In general, the pressure field at the seabed depends on the ambient sea conditions, and can vary dramatically with weather and location, as well as with the action of waves and tides, see for example Wang et al. (2007). However, the design and fabrication of underwater mines allows for an easy recognition of signals with shorter or longer period than the one characterizing the transit of a ship (Barnes, 1998). Therefore, it is crucial to predict the hydrodynamic pressure field generated by a craft in an accurate manner for a better understanding of ships vulnerability to underwater mines and to design suitable mine countermeasures. Analytical and numerical methods have been employed over the past few decades to predict the bottom pressure field generated by travelling surface disturbances. Sahin and Hyman (1993) and Sahin et al. (1994) have proposed a numerical solution of the potential flow around a submerged Rankine body in a uniform stream of finite-depth water. Pressure coefficients therein are calculated for several geometries and depth Froude numbers ranging in [0.3 - 0.9], showing spatial features of the bottom pressure field and its dependence on Fr_h. In a later work, Sahin and Hyman (2001) have used this methodology to predict the flow field and bottom pressure generated by an air-cushion vehicle, modeled as a pressure patch moving on the surface of water. In their work, a few supercritical cases are considered up to $Fr_h = 2.00$, providing the first useful comparisons between bottom pressure generated at subcritical and supercritical speeds. More recently, Nguyen and Hyman (2010) have presented a hybrid methodology for the computation of the near and far-fields generated by planing boats. For the flow in the vicinity of the hull, a Reynolds averaged Navier Stokes (RANS) solver is employed to accurately model viscous and nonlinear effects, while potential theory is utilized in the outside region representing the far-field, with the RANS solution used as a boundary condition at the interface. Results from their work have been subsequently verified with a good qualitative agreement by Tafuni and Sahin (2014), with numerous simulations conducted to investigate the variation of the bottom pressure due to the passage of a planing boat at several supercritical speeds.

3.2 Problem formulation and numerical set-up

To study the flow generated by a high-speed vessel, a global Cartesian coordinate system O(X, Y, Z) is defined with the origin fixed such that Y = 0 is the boat centerplane, Z = 0 corresponds to the undisturbed free surface and X = 0 so that every SPH particle has a positive X-coordinate, as illustrated in Figure 3.1. Furthermore, a



Figure 3.1: Sketch of the proposed 3-D model.

second Cartesian coordinate system O(x, y, z) relative to the boat motion is defined, such that x = 0 is the plane passing through amidships, y = 0 is the boat centerplane and z = 0 corresponds to the undisturbed free-surface. In this second reference system, x, y and z axes represent the roll, pitch and yaw axes, respectively.

In Figure 3.2, the nomenclature for the numerical set-up is shown. The length, width and depth of the water domain are equal to 28L, 12L and 0.62L, respectively. In the range of the Froude numbers considered herein, these values represent a good compromise for the size of the simulation domain, allowing a fully converged solution in terms of waves and pressure field, a negligible effect from the domain boundaries and reasonable computational times. The boat length overall (LOA), L = 16 feet, is used to nondimensionalize all the other lengths. Sizes and shape of the hull, whose CAD representation is illustrated in Figure 3.3, are chosen to match those used by Nguyen and Hyman (2010), allowing for the comparison of the results. Numerical simulations are performed for depth Froude numbers, Fr_h, and hull Froude numbers, Fr_L , both within [1.00 - 4.00], varying the boat speed, U, while maintaining the water depth, h, and L constant. The hull sinkage and trim angle are fixed to arbitrarily small values of 0.05L and 3 degrees, respectively, and they are held constant throughout the simulations. Although in planing conditions the trim angle is expected to decrease as the speed of the boat increases (Savitsky, 1964), the choice of a relatively small trim value will not affect the wave field significantly.



Figure 3.2: Sketch of the computational domain with top and side views.



Figure 3.3: CAD model of the planing boat.

3.3 SPH solver and parameters

DualSPHysics v3.1 is employed for simulating this problem (Crespo et al., 2015). The system of ODEs represented by Equations (2.37) and (2.47) is integrated in time with the use of the symplectic scheme described in Crespo et al. (2015), whereas Equation (2.28) is simply updated using the density value computed from Equation (2.37). Through the procedure described in Monaghan and Kos (1999), optimal time-stepping is achieved while respecting the Courant-Friedrichs-Levy criterion and ensuring stability of the viscous and forcing terms. A lower bound of $\Delta t_{\min} = 10^{-7}$ s is imposed for the numerical time step, although the average value of Δt in the simulations is noted two orders of magnitude higher on average. The initial spacing, Δx , among SPH particles is set to the same value throughout the computational

domain, as no adaptive spacing algorithm is available in this version of DualSPHysics. Therefore, an iterative process is used for determining the optimal value of Δx , based on the accuracy of the solutions and the computational time, with further details shown in the results section.

The flow is studied by considering the hull moving with a constant velocity on the free-surface of a volume of still water enclosed in a solid tank. As seen in Section 2.5.2, both kinematic and dynamic requirements of the free-surface boundary condition are satisfied implicitly by the Lagrangian formulation and integral interpolation adopted in SPH. Dynamic boundary particles are employed to enforce no-slip condition at the solid-fluid interface. These particles are used to discretize the tank walls and the hull, for which the algorithm controlling the displacement is set to zero and a uniform rectilinear motion, respectively.

3.4 RANS solver

The RANS solver used for comparing with the SPH solver results is CFDSHIP-Iowa (Carrica et al., 2010). This solver is designed for use in computing the unsteady flow over vehicles on or near the free-surface, including displacement and planing ships and surfaced submarines as well as breaking waves. The solver operates on a multiblock structured overset grid with user-selected discretization schemes. The pressure solution can be obtained via Pressure Implicit with Splitting of Operator (PISO) or projection methods. The free-surface is treated as the interface between the gas phase (atmosphere) and the liquid phase (water) via a level-set surface capturing model. Coupling between the gas and liquid phase is one-way, i.e. the gas phase does not affect the liquid phase. The solver utilizes static or dynamic overset blocks so that relative motion between blocks is allowed. This is accomplished by computing the interpolation coefficients needed for block-to-block communication at each time step via the overset solver SUGGAR (Noack, 2006). Block movement is especially helpful when treating dynamic sinkage and trim of a surface craft or treating the effects of moving appendages.

In the present application, the vehicle is placed in a static orientation relative to the undisturbed free-surface and the unsteady solution process is begun. Interpolation coefficients thus need to be computed only at the start of each solution. The solutions are continued until all forces and moments become stationary with time. The grid is constructed with y+=1 and expanded from the wall. Approximately 15 points are in the boundary layer and the grid resolution degrades with distance from the hull. The vertical resolution near the free-surface, needed for surface capturing, is maintained to approximately L/2. Far from the hull, large spacing is generally used to control overall problem size. In the current case, the gas phase region is defined by especially coarse gridding as the solution there is of no interest.

3.5 Results with the artificial viscosity model

3.5.1 Bottom pressure

The first group of results is obtained without implementing any model of physical viscous stresses but the artificial viscosity alone, therefore flow fields shown in this section are essentially inviscid. This is considered a good initial approximation to start tackling this problem as a variety of results obtained with other inviscid techniques is also available in the literature.

For each simulated case, the water depth, h_w , is kept constant while the boat speed, hence the depth Froude number, is varied according to Table 3.1. Bottom pressure contours and non-dimensional pressure signatures at lateral positions are presented, whereas free-surface elevation is not considered at the moment due to the rather high wave dissipation introduced by the use of the artificial viscosity.

Figure 3.4 shows the pressure contours at water depth $h_w = 0.62L$ and several values of the depth Froude number. Other than the hydrostatic configuration, P_0 , represented in green, two areas of low and high pressures are observed behind and

Fr_{h}	Speed [knots]	Depth
1.00	10.62	0.62L
1.25	13.28	0.62L
1.50	15.94	0.62L
1.75	18.59	0.62L
2.00	21.25	0.62L
2.25	23.91	0.62L
2.50	26.56	0.62L
2.75	29.22	0.62L
3.00	31.87	0.62L
3.25	34.53	0.62L
3.50	37.19	0.62L
3.75	39.84	0.62L
4.00	42.50	0.62L

 Table 3.1:
 Simulation parameters.



Figure 3.4: Pressure contour plots for $h_w = 0.62L$ and $Fr_h = 1.00$ (top-left panel), $Fr_h = 2.00$ (top-right panel), $Fr_h = 3.00$ (bottom-left panel), and $Fr_h = 4.00$ (bottom-right panel).

ahead of the boat, respectively. The round-shaped high pressure region is located underneath the bow, enclosed in its vicinity until the pressure adjusts to the hydrostatic value. Conversely, the low pressure region, generated after the boat has passed by, extends for quite a large portion of the domain, assuming a V-shape, with the lowest pressure value located just behind the stern. At $Fr_h = 2.00$, the pressure distribution is in good qualitative agreement with results in Nguyen and Hyman (2010). The angle between the low pressure wake and the horizontal direction, x, varies with the depth Froude number in a manner also shown in Figure 3.4. As the Froude number



Figure 3.5: Bottom pressure at several lateral positions y/d.

increases, a reduction in the wake angle is observed, from a value of about 50° at $Fr_{h} = 1.00$, to a value of about 10° at $Fr_{h} = 4.00$. As expected, a similar phenomenon with surface waves generated in the same dynamic conditions is observed. The angular decrease of the low pressure wake with the increasing velocity of the craft is estimated for two ranges of the depth Froude number. Results are compared with findings in Rabaud and Moisy (2013), where a study on surface waves is carried out for a craft of comparable size. In the transition from $Fr_{h} = 1.00$ to $Fr_{h} = 2.00$, the pressure wake angle herein decreases by 60%, whereas the surface wake angle in Rabaud and Moisy (2013) decreases by 63.5%. Similarly, from $Fr_{h} = 2.00$ to $Fr_{h} = 4.00$, the pressure angle reduces by 52.5% while the wave angle reduces by 50%, showing how the two phenomena follow the same physical trend.

In Figures 3.5a to 3.5d four pressure tracks along the direction of motion are shown at different lateral positions for $Fr_h = [1 - 4]$. The net bottom pressure, P, is non-dimensionalized with respect to the hydrostatic term, $P_0 = \rho g h_w$. Distances over the x and y directions are non-dimensionalized with respect to the reference lengths L and d, corresponding to the LOA and an arbitrary distance of $h_w/2$, respectively. As from previous observations, at the centerplane, y/d = 0, the pressure has a peak in the vicinity of the bow region, followed by a sharp decrease and a minimum of P/P_0 as one moves away from the stern. Finally, the pressure increases again and adjusts to the hydrostatic value. Similar trends are observed for y/d = 1, y/d = 2, and y/d = 3, with the magnitudes of high and low pressures gradually decreasing as the distance y/d increases. At supercritical speeds (Figures 3.5b to 3.5d), the characteristics of the pressure signatures differ due to a change in the magnitude of pressure disturbances generated by the craft. Specifically, the pressure under the bow increases quite steeply at the centerline y/d = 0 and in its close proximity, while it decreases substantially already at lateral distances y/d = 2 and farther. As expected, a similar but opposite trend is exhibited by the low pressure behind the stern, although a singular behavior is observed for the critical Froude number, where the pressure reaches a significant low value. Pressure minima obtained at the same location for the first supercritical speed are, in fact, lower than values for $Fr_h = 1.00$, especially at large y/d, see Figure 3.5b. As Fr_h increases, reaching a certain threshold located between $Fr_h = 2.00$ and $Fr_h = 3.00$, the pressure reduction generated behind the stern becomes more significant than the critical case. At $Fr_h = 2.00$, magnitudes of high and low pressures obtained with SPH are compared with results obtained with the hybrid method in Nguyen and Hyman (2010), showing good qualitative agreement. For the latter, pressure peaks at the centerline of the boat, y/d = 0, are of the order $0.025 \ P/P_0$ for the high pressure and $-0.016 \ P/P_0$ for the low pressure, showing that SPH overestimates pressure magnitudes by a factor of approximately 4 for the high pressure and 2.5 for the low pressure. There are several possible reasons behind these large predictions, starting with the dissipative nature of the dynamic particles used to discretize the domain boundaries, which are sometimes responsible for the generation of unphysical force fields in the proximity of a solid boundary (Crespo et al., 2007). Moreover, in order to enforce the stability of the WCSPH algorithms, relatively high values of the artificial viscosity coefficient, α , have been used. On the other hand, no information on the hull draft or wetted surface area is available for the results in Nguyen and Hyman (2010), hence no a priori conclusion can be drawn on the intensity of the generated pressure disturbance.



Figure 3.6: Bottom pressure signatures at the centerline, y/d = 0, as a function of the Froude number.

Pressure signals along the centerline of the boat are grouped in Figure 3.6 for all four Froude numbers. High pressure values are aligned about the same x/L value, just ahead of the bow, and they have a fairly quadratic dependence on the Froude number, especially for the first three speeds, namely Fr = 1.00, Fr = 2.00 and Fr = 3.00. This indicates the physical dependence of the dynamic pressure with the square of the boat speed, which will be assessed in the next section. Conversely, the location of pressure minima is always located behind the stern and it moves towards decreasing x/L for increasing Froude numbers.

3.6 Results with the SPS viscosity model

Results from SPH simulations are presented hereinafter using the sub-particle scales (SPS) approach illustrated in previous chapters. This viscosity formulation, which includes a model of turbulence based on LES, is particularly indicated for wave problems and high-speed regimes, making it a good candidate for this investigation.



Figure 3.7: Contours of nondimensional wave elevation, η , for different depth Froude numbers.

3.6.1 Wave elevation

Figure 3.7 shows contours of terms nondimensional wave elevation, $\eta = z/L$, generated by the boat at several cruising speeds, i.e. a depth Froude number in the range [1.00 - 4.00]. The bow and stern positions correspond to x/L = 0.5 and x/L = -0.5, respectively. In all cases, the wake is characterized mainly by large divergent waves that form a V-shaped wedge with an angle that decreases notably as the depth Froude number increases, in agreement with predictions of Rabaud and Moisy (2013, 2014) and He et al. (2015). The absence of transverse waves behind the stern is observed and expected. The length of such waves is outside the wave spectrum excited by the hull, and their amplitude typically starts to damp out at $Fr_L = U/\sqrt{gL} = 0.5$. This particular value of the hull Froude number is often considered a threshold beyond which a craft can be defined a HSV (Faltinsen, 2010), and the generation of narrower V-wakes begins as the Froude number increases thereafter. At $Fr_h = 1.00$ in Figure 3.7a, several crests and troughs can be noted, with the first wave in the group having the largest amplitude. For this particular case, a good qualitative agreement is observed with the numerical results in Scullen and Tuck (2011) for a rectangular pressure patch moving in calm water at $Fr_L = 0.71$, a value of the length Froude number



Figure 3.8: Wave elevation along y/L = 0 for different depth Froude numbers.

that is close to the critical depth Froude number used in this work, e.g. $Fr_L = 0.78$.

Only a small portion of the domain displayed in Figure 3.7a corresponds to undisturbed free-surface, whereas greater areas of $\eta = 0$ can be observed for larger values of Fr_h in Figures 3.7b to 3.7d due to the significant decrease in the wake angle. Beyond the critical regime, the presence of spray is noted, initially confined around the hull and aft for $Fr_h = 2.00$ and then extending for several boat lengths at higher Froude numbers. Furthermore, rooster tails and hull hollows resulting from the flow separation at the stern are also observed, with shape and magnitude that are strongly dependent on Fr_h . These regions can be better examined in Figure 3.8, where a cut of the wave field through the boat centerline is shown for the same Froude numbers as above. At the critical speed, the minimum and maximum amplitudes of the waves, i.e. $\eta \approx -0.035$ and $\eta \approx 0.02$, correspond to the water height at the hull hollow and rooster tail, respectively. At $Fr_h = 2.00$ and larger, the spray sideways of the hull dominates the maximum free-surface height, whereas lowest values of η are still attained at the boat centerline, in correspondence of the hollow behind the stern. The span of this region widens with the increase of Fr_h , reaching four hull lengths at $Fr_h = 4.00$. A similar pattern is also observed for the rooster tail behind the hollow, which is followed by a number of additional regions of troughs and crests, especially noticeable at supercritical Froude numbers.



Figure 3.9: Contours of nondimensional bottom pressure, θ , for different depth Froude numbers.

3.6.2 Bottom Pressure

The pressure field generated by the boat at the seafloor is hereby presented via contours of nondimensional pressure, $\theta = 2(P - P_{hyd})/\rho U^2$, as shown in Figure 3.9. Common spatial features can be noted for the supercritical cases, Figures 3.9b to 3.9d, with two significant areas other than hydrostatics corresponding to $\theta > 0$ and $\theta < 0$, respectively. The high pressure region is located underneath amidships, and it is given by a disk with a radius of approximately 3/4L. The second area is represented by a V-shaped low-pressure wedge that occupies the entire domain region behind the stern. A different pressure field is generated at $Fr_h = 1.00$ in Figure 3.9a, where the low pressure area with highest magnitude is now confined in a significantly smaller portion of the domain, behind which the presence of distinct secondary wakes of high and low pressure is observed. By comparing the contour plots, it is possible to note a spatial dependence of θ from the dynamics of waves at the surface, as the angle amid the arms of the low-pressure wedge decreases with increasing Fr_h in the same way previously seen for η . For the $Fr_h = 2.00$ case, a very good agreement of the pressure contours is noted with Nguyen and Hyman (2010). Moreover, comparison of these contours with findings from simulations with the artificial viscosity above



Figure 3.10: Bottom pressure along y/L = 0 for $Fr_h = 2.00$: simulation data (dashed line), fitted curves (solid lines).

is made. A qualitative agreement is observed for the shape of the pressure field calculated with both viscosities, although the inviscid model is not able to capture secondary pressure wakes, especially seen in the critical case when using SPS. This likely depends on the dampening nature of the artificial viscosity, with waves being suppressed at the surface. The diffusive nature of the artificial viscosity allows for smoother pressure fields than the SPS ones, regardless of the use of density filtering in both cases. It is worth mentioning that the code version used in this work does not implement the δ -SPH correction for the density of boundary particles, therefore this could also be treated to further improve the spurious oscillations affecting θ .

To clear the pressure field from non-physical quantities, the MATLAB[®] smoothing spline algorithm is applied, using weights and a smoothing parameter to filter out the noise from the data. An example of how the algorithm is adapted to this study is shown in Figure 3.10, where a plot of θ along the boat centerline is shown for Fr_h = 2.00. Physically meaningful variations of pressure are identified for $x/L \in$ [-3 – 2], therefore a higher value of the smoothing parameter is used in this interval to capture the pressure disturbance generated by the boat with a good level of confidence. The rest of the curve is made of spurious oscillations around the hydrostatic value, which are eliminated using a lower value of the smoothing parameter.

Following this procedure, an overview of the pressure signatures at the seafloor is presented in Figure 3.11 for the same values of Fr_h used for comparison above. In each plot, the pressure along x/L is shown in the semi-plane $y/L \in [-5 - 0]$. At the critical speed, the largest values of θ are attained at the centerline, y/L = 0, with



Figure 3.11: Nondimensional bottom pressure signatures for $Fr_h \in [1.00 - 4.00]$.



Figure 3.12: Nondimensional bottom pressure signatures at the boat centerline as a function of the depth Froude number.

peaks $\theta_{\text{max, Fr1}} \approx 0.009$ and $\theta_{\text{min, Fr1}} \approx -0.015$ located at the bow and behind the stern, respectively. The pressure disturbances spread laterally reaching another local peak at y/L = -1.0 in correspondence of x/L = -3.5, gradually weakening and disappearing



Figure 3.13: Plots of nondimensional bottom pressure, θ , (solid curve) and wave elevation, η , (dashed curve) at different x/L positions for Fr_h = 3.00. Scales are relative to nondimensional pressure. Wave elevation is scaled for comparison.

towards the edges of the domain. For depth Froude numbers beyond the critical threshold, peaks in the bottom pressure along the boat centerline are characterized by preponderant values of the high pressure, $\theta_{\text{max, Fr2-Fr4}} \approx 0.0065 - 0.008$, against lowest pressures peaking around the common value $\theta_{\text{min, Fr2-Fr4}} \approx -0.00475$. For these three cases, pressure disturbances at lateral locations are represented mainly by local depressions that decay rapidly at y/L = -1.0 and farther from the hull.

To better assess the dependence of the pressure signals from the depth Froude number, the centerline signatures are aggregated in Figure 3.12. Two meaningful observations can be made regarding the location and intensity of the pressure peaks. For increasing Froude numbers, the position of θ_{\min} shifts backwards of the hull, from $x/L \approx -0.8$ at Fr_h = 1.00 to $x/L \approx -2$ at Fr_h = 4.00. A similar phenomenon is noticed for the position of θ_{\max} from Fr_h = 1.00 to Fr_h = 2.00, whereas the high pressure disk remains around amidships at larger speeds. In terms of pressure magnitude, a quadratic relationship between θ and Fr_h is evident for the boat in supercritical regime, with pressure minima and maxima attaining nearly the same nondimensional value. Conversely, a much larger value of θ_{\min} is seen when the hull navigates at the critical speed, approximately three times the magnitude of θ_{\min} , Fr2-Fr4. A possible explanation for this phenomenon is related to the surface waves generated by the boat at the critical Froude number, where the hull approaches the hump of the bow wave, perturbing the free-surface and therefore the pressure at the seafloor more significantly. In this special case, the hydrodynamic drag on the hull is also seen to attain a local maximum due to the high value of the wave-making resistance component, contributing to an overall higher drag (Yun and Bliault, 2012).

In Figure 3.13, the relationship between surface waves and bottom pressure at $Fr_h = 3.00$ is traced by superimposing snapshots of both quantities at the same x/L positions. For $x/L \in [-0.5 - 0.5]$, delimiting the region between the bow and the stern, an increase and decrease of θ along the diameter of the high pressure disk is noticed, whereas an hollow is identified at the free-surface, in correspondence of the stern. Moving behind x/L = -0.5, the free-surface hollow stretches along y/L = 0, matched by a similar pressure profile, and as soon as the rooster tail begins to emerge at the surface, a small pressure hump begins to form at the boat centerline, accordingly. The creation of further crests and troughs of θ farther away from the stern follows that of η , qualitatively.

3.6.3 Comparison with RANS solutions and assessment of the low-pressure wake angle

In this section, a comparison of the SPH results with simulations data from the RANS solver CFDSHIP-Iowa (Tafuni et al., 2016) is provided and the dependence of the low-pressure wake angle on the depth Froude number is assessed. Bottom pressure contours obtained with both methods are shown in Figures 3.14 and 3.15 for $Fr_L = 2.00$. The axes limits chosen for representing the data correspond to the actual length and width of the computational domain used in the RANS simulations, much smaller than the SPH domain due to the presence of open boundaries. A good agreement is noted for the shape and position of the pressure disturbances, whereas some discrepancies are observed for the magnitudes of θ in Figure 3.14, approximately 40% smaller than predictions by SPH. This difference is attributed to the grid dissipation caused by the use of a non-uniform mesh in the RANS simulations that coarsens from the free-surface to the ocean bottom. This feature of the computational grid also affects the ability of the RANS solver to capture secondary disturbance patterns behind the principal pressure wake, given their smaller magnitude.



Figure 3.14: RANS bottom pressure contours, $Fr_L = 2.00$.



Figure 3.15: SPH bottom pressure contours, $Fr_L = 2.00$.

To obtain a quantitative relationship between the shape of the bottom pressure profiles and the Froude parameter, the half-angle, β , between the direction of motion and the low-pressure wedge arm in the negative y/L semi-plane is measured for both Fr_h and Fr_L in [1 - 4]. The results are shown in Figure 3.16 using the depth Froude number as the independent variable. For each simulation case, the angle β is measured consistently, starting from the point of minimum pressure at the boat centerline and best-fitting a line through the direction of the low pressure strip. The angle measurements from SPH simulations exhibit an inverse relationship with the depth Froude number in the same manner as the surface wake angle modeled by Rabaud and Moisy (2013). To quantify this dependence, a nonlinear least squares error fit of the measured data is performed with a 1/Fr_h fit model that yields

$$\beta(\mathrm{Fr_h}) = \frac{2.3}{\sqrt{2\pi} \,\mathrm{Fr_h}} - 0.07$$
 (3.1)

with a coefficient of determination $R^2 = 0.9976$. For the depth Froude number around the unity, the value of β is approximately 0.87 radians, whereas it narrows to roughly



Figure 3.16: Low-pressure wake half-angle (radians) as a function of the depth Froude number.

Table 3.2: Comparison of the low pressure wake half-angle, β , obtained from RANS solutions and SPH simulations.

β (rad) Fr _L	RANS	SPH	Difference RANS–SPH
1.00	0.5695	0.6435	-12.20%
2.00	0.2294	0.2776	-19.01%
3.00	0.1420	0.1684	-17.01%
4.00	0.1052	0.1303	-21.31%

0.38 and 0.16 radians for $Fr_h = 2.00$ and $Fr_h = 4.00$, respectively. The dependence of β from Fr_h in Equation (3.1) indicates the correlation between surface waves and bottom pressure in the range of speed considered, where both quantities scale as Fr_h^{-1} . In Table 3.2, values of β calculated from RANS simulations are compared with those from SPH and discrepancies are quantified in percent form. Overall, a good level of agreement is observed between the two methods, although the RANS estimates suggest a systematic narrower angle, around 10–20% smaller than SPH predictions.

Chapter 4

Harmonic oscillations of a rigid lamina in a viscous fluid with and without a free surface

In this section, Smoothed Particle Hydrodynamics is implemented to study the motion of a thin rigid lamina undergoing large harmonic oscillations in a viscous fluid with and without a free surface. For the latter, the flow physics in the proximity of the lamina is resolved and contours of non-dimensional velocity, vorticity and pressure are presented for selected oscillation regimes. The computation of the hydrodynamic load due to the fluid-structure interaction is carried out using Fourier decomposition to express the total fluid force in terms of a non-dimensional complex-valued hydrodynamic function, whose real and imaginary parts identify added mass and damping coefficients, respectively. For small oscillations, the hydrodynamic force reflects the harmonic nature of the displacement, whereas multiple harmonics are observed as both the amplitude and frequency of oscillation increase. In the range of the chosen parameters, a novel formulation of hydrodynamic function incorporating added mass and damping coefficients is proposed for large amplitude oscillations. Results of the simulations are validated against numerical and experimental works available in the literature in addition to theoretical predictions for the limit case of zero-amplitude oscillations. To treat the free surface case, the Boundary Element Method (BEM) is chosen, with the free surface modeled through a stress-free boundary condition. A Stokes flow assumption is adopted and the non-linear convective terms are neglected. The problem is formulated in Fourier and Laplace transform spaces, with results once again expressed in non-dimensional form through an expression for the hydrodynamic force that identifies added mass and damping coefficients. Several parametric studies are conducted to evaluate the effects of two non-dimensional parameters, namely the frequency of oscillation and the depth of submergence. Results are compared with the classical solution for oscillating laminae in an unbounded viscous fluid and validated using Smoothed Particle Hydrodynamics for small amplitude cases. Furthermore, a preliminary characterization of the hydrodynamic load in the case of finite-amplitude oscillations near a free surface is presented.

4.1 Literature review

The coupling of engineered structures with a surrounding viscous fluid represents a challenging fluid-structure interaction (FSI) problem. Increasing attention among researchers is being paid to FSI problems in many fields of engineering, from marine propulsion and vortex-induced vibrations (VIV) (Du et al., 2014; Nguyen et al., 2012), to biomimetic thrusters (Triantafyllou et al., 1993, 2000), micro-electro-mechanical systems (MEMS) (Kimber et al., 2009) and atomic force microscopy (AFM) (Basak et al., 2006; Sader, 1998). Specifically, slender structures, such as cylinders, plates, or beams coupled with a moving viscous fluid are widely used in engineering applications such as the above-mentioned. Analytical models for the solution of FSI problems are appealing due to the availability of closed-form solutions, and a great amount of work has been funneled towards this direction (e.g. Amabili, 1996a,b; Amabili and Kwak, 1996; Lamb, 1920; Tuck, 1969). However, the theoretical tractability of these problems is restricted to a few particular cases and the use of numerical and experimental techniques is required for solving more general cases, see for example Amabili (2000, 2001) and Amabili et al. (2002). The coupling of thin cantilever beams with a viscous oscillatory flow has entailed a great deal of research in the past decades (Aureli and Porfiri, 2010; Aureli et al., 2012; Bidkar et al., 2009; Sader, 1998; Tuck, 1969). A common approach to studying the vibrations of slender cantilevers in viscous fluids is to model the cross-section of the beam as a rectangular plate with negligible thickness. Thus the beam is considered infinitely thin in the direction of vibration and it is possible to bypass the complexity of the 3-D vibrational problem by focusing the attention on the two-dimensional flow field generated by transverse oscillations of the beam cross-section, provided that low vibration modes are considered. Detailed work on the flow variability along the axis of thin cantilevers is reported in Facci and Porfiri (2013), where the acceptability of the two-dimensional approach mentioned above is deemed correct for a wide range of width-to-length ratios. For beam cross-sections undergoing harmonic oscillations, the nature of the displacement and subsequent hydrodynamic response promote the use of a non-dimensional frequency parameter, β , (Sarpkaya, 1986) rather than the classical Reynolds number. Analytical works based on the linearization of the Navier-Stokes equations (Sader, 1998; Tuck, 1969) have shown the unique dependence of the hydrodynamic load on β for infinitely small oscillations. Results therein are presented in form of a complex-valued hydrodynamic function, $\Gamma(\beta)$, identifying added mass and hydrodynamic damping originated by the fluid surrounding the lamina. A detailed study performed by Aureli et al. (2012) has taken into account the non-linearities arising from vortex formation, shedding and advection in the case of finite-amplitude vibrations of thin cantilever beams. The hydrodynamic function, $\Theta(\beta, \text{KC})$, computed therein, incorporates a correction term, $\Delta(\beta, \text{KC})$, representing the dependence of the resultant fluid force on an amplitude parameter, represented by the Keulegan-Carpenter number (KC). Furthermore, several formulations have been proposed to consider the effect of parameters such as the presence of a solid wall or a free surface in the vicinity of the oscillating lamina (Grimaldi et al., 2012; Tafuni and Sahin, 2013), the influence of the lamina width-tothickness ratio (Phan et al., 2013), or coupled vibrations of two laminae (Intartaglia et al., 2014). In each case, a different hydrodynamic response is observed and a variety of hydrodynamic functions have been cast to predict fluid actions in the form of added mass and damping coefficients.

The application of Smoothed Particle Hydrodynamics (SPH) to fluid-structure interaction problems such as the above-mentioned has increased from the late 1990s onward. As seen in the previous chapter, it is possible to treat moving boundaries and free surfaces quite naturally with SPH, as flow variables are not retrieved on a grid but intrinsically retained by the particles. This feature is particularly appealing when simulating moderate to large fluid deformations due to a moving boundary and therefore justifies the choice of SPH in this particular study. To the best of the author knowledge, no previous attempts of using SPH for the solution of this problem can be found in the literature.

4.2 Governing equations

Large harmonic oscillations of a rigid lamina in a quiescent viscous fluid are considered. The nomenclature adopted to describe the FSI problem is summarized in Figure 4.1. Particularly, the analysis is focused on a thin rectangular plate, Ψ , with Band H representing the width and thickness, respectively, and $H \ll B$. The problem is formulated by attaching a Cartesian reference frame, O(x, y, z), to Ψ , such that



Figure 4.1: 3-D sketch of a thin cantilever beam with clamped base and free end (left) and projection of the cross-section Ψ onto its plane (right).

the y-axis is aligned with B, the z-axis is through H, the x-axis closes the right-hand system and the origin is coincident with the centroid of Ψ . The lamina oscillates in the vertical direction with displacement z(t) represented by a sinusoidal function of amplitude A and radian frequency $\omega = 2\pi f$ as

$$z(t) = A\sin(\omega t) \tag{4.1}$$

At each instant in time, the thin plate is surrounded by an incompressible viscous fluid, which is responsible for exerting hydrodynamic loading in response to harmonic oscillations. The continuity and Navier-Stokes momentum equations are employed to describe the motion of the fluid, i.e. Equations (2.14) and (2.15). Following Sarpkaya (1986), a non-dimensional frequency parameter is introduced in the form of

$$\beta = \frac{\rho \omega B^2}{2\pi \mu} \tag{4.2}$$

which can also be interpreted as an oscillatory Reynolds number. Moreover, a second non-dimensional parameter is introduced, namely the Keulegan-Carpenter number, KC, (Keulegan and Carpenter, 1958; Bidkar et al., 2009)

$$KC = \frac{2\pi A}{B} \tag{4.3}$$

For oscillations in an unbounded viscous fluid, the use of the non-dimensional pair (β, KC) is sufficient to model the total fluid force, F(t, y, z), experienced by the plate

(Aureli et al., 2012). To this extent, a hydrodynamic function, $\Theta(\beta, \text{KC})$, is introduced

$$\Theta(\beta, \mathrm{KC}) = \Theta_{add}(\beta, \mathrm{KC}) + i\Theta_{dam}(\beta, \mathrm{KC}) \tag{4.4}$$

where the real part is representative of an added mass coefficient, while the imaginary part is identified as a hydrodynamic damping coefficient (Sader, 1998; Aureli and Porfiri, 2010). The use of complex notation is due to the oscillatory nature of F(t, y, z), which is correlated to Equation (4.4) by

$$\hat{F}(\omega) = \frac{\pi^2}{2} \rho f^2 B^3 \Theta(\beta, \text{KC}) \text{ KC}$$
(4.5)

where $\hat{F}(\omega)$ is the phasor counterpart of F(t, y, z) obtained from the Fourier transform of the fluid force, see for example Sader (1998). As mentioned earlier, several formulations have been proposed for Equation (4.4) in order to predict the influence of oscillation parameters on the resultant fluid force. In this work, a mathematical model of hydrodynamic function is seek to describe the total fluid force acting on a thin rectangular plate undergoing oscillation strokes comparable to its width, *B*. Starting from the numerical model in Aureli et al. (2012)

$$\Theta(\beta, \mathrm{KC}) = \Gamma(\beta) - i \ 0.879 \ \beta^{0.75} \left(\frac{\mathrm{KC}}{2\pi}\right)^2 \tag{4.6}$$

formulated for $\beta \in [20 - 2000]$ and KC < 11.42 $\beta^{-0.625}$, and extend the amplitude range to values of KC outside the convergence inequality. In Equation (4.6), the term $\Gamma(\beta)$ refers to the hydrodynamic function derived by Sader (1998) for zeroamplitude oscillations, whereas the second term is representative of a correction due to finite-amplitude effects and involves only the imaginary part of $\Theta(\beta, \text{KC})$, i.e. the damping coefficient. The total fluid force obtained from numerical simulations herein is modeled using a truncated Fourier series and data from the fundamental harmonic are utilized to derive a new expression for the hydrodynamic function, similar to that in Equation (4.6) but valid in the range of the proposed parameters.

4.3 Results for a fluid of infinite extent

Results of the numerical analysis for an unbounded fluid are presented herein. As shown in Table 4.1, SPH simulations are conducted for values of the non-dimensional frequency in (200 - 2000). Conversely, the choice of the amplitude parameter is influenced by several factors, among which the domain size, a steady-state hydrodynamic force and the influence of WCSPH effects upon the solution quality.

To provide means of comparison with other works, the lowest KC numbers within the range of Aureli et al. (2012) and Bidkar et al. (2009) are selected. However, at each β , the value of the minimum acceptable KC is also constrained by the influence of spurious oscillations in the WCSPH density field that become significant at small amplitudes and degrade the quality of the solution. The parameter KC is gradually incremented until the influence of the domain boundaries becomes predominant, leading to the rapid decay of the solution quality. Therefore, an upper bound for the maximum admissible amplitude is also established based on a steady-state force solution over a minimum of three oscillation cycles. In Table 4.1, both lower and upper bounds are seen dependent on the frequency parameter into consideration: for small β , larger KC can be achieved, whereas smaller KC are difficult to manage; the opposite trend is observed as β increases.

4.3.1 Velocity and vorticity fields

In Figures 4.2 to 4.4, contours of fluid velocity and vorticity in the surroundings of the oscillating lamina are presented for a fixed frequency, $\beta = 976$, and three distinct KC values, namely KC = $\pi/10$, KC = $3\pi/5$ and KC = $6\pi/5$. In the range of the chosen parameters, these three cases are considered distinctive of small, moderate and large oscillation amplitudes, respectively. In each figure, the left panel represents the magnitude of the velocity vector, U, normalized with respect to the reference quantity, $U_{REF} = 2\pi f A$, whereas the right panel shows the x-component of the vorticity vector, ζ , normalized with respect to the reference quantity, $\zeta_{REF} = U_{REF}/B$. Time, t, is

β	KC
244	$\pi/5, 2\pi/5, 3\pi/5, 4\pi/5, \pi, 6\pi/5, 7\pi/5$
488	$\pi/5, 2\pi/5, 3\pi/5, 4\pi/5, \pi, 6\pi/5$
976	$\pi/10, \pi/5, 2\pi/5, 3\pi/5, 4\pi/5, \pi, 6\pi/5$
1952	$\pi/25, \pi/10, \pi/5, 2\pi/5, 3\pi/5, 4\pi/5$

 Table 4.1:
 Simulation parameters.

non-dimensionalized with respect to the oscillation period, T = 1/f. Each panel comprises five sub-figures representative of a half period of oscillation, starting at t/T = 2.5 with the lamina at z(t)/A = 0 and having negative z-velocity, $-U_{REF}$, and ending at t/T = 3, with the lamina at z(t)/A = 0 and with reversed velocity, U_{REF} .

In general, a common trend characterizes the velocity contours in that for all three cases velocity peaks are obtained when the plate crosses the y-axis at t/T = 2.5 and t/T = 3, whereas velocity minima are attained at t/T = 2.75. Though the support of the velocity field changes notably as one transitions from $\text{KC} = \pi/10$ to $\text{KC} = 3\pi/5$, and similarly to $KC = 6\pi/5$, it can still be argued that in a purely sinusoidal motion such as that in Equation (4.1), z(nT/2) and z(nT/2 + T/4), with $n = \pm 0, 1, 2, ..., \infty$, are indeed the maximal and minimal (zero) velocity configurations, respectively. Nonlinearities and vortex shedding are observed in all simulated cases, particularly when the lamina traverses large amplitudes. For example, comparing velocity scales of Figures 4.2 to 4.4, it can be noted that the rate of change of the velocity with respect to the oscillation amplitude increases with KC qualitatively. Assessment of non-linearities attributable to hydrodynamic actions can be made also by comparing Figure 4.2 to Figure 4.3 and 4.4 with the lamina in the zero-velocity configuration at t/T = 2.75. In the small amplitude case, the vast majority of the fluid surrounding the plate has a very small or null velocity, with areas of non-negligible velocity just around the tips, suggesting that in this case the hydrodynamic response to the displacement of the lamina is fast. Conversely, the fluid portion characterized by a non-zero velocity is much larger for $KC = 3\pi/5$, and this is even more evident for $KC = 6\pi/5$, where the fluid velocity is non-zero in an area of about the size of a circle with radius $\approx 2B$.

The vorticity evolution for $\beta = 976$ is illustrated in right panels of Figures 4.2 to 4.4, where the temporal identification of maxima and minima follows that of the velocity field qualitatively. It has been noted (Aureli et al., 2012) that KC influences vortex generation and shedding as, for increasing amplitudes, viscous effects responsible for the diffusion of the vorticity are contrasted by convection mechanisms generated by the displacement of the lamina. As a result, the lifespan of produced vortices is lengthened due to increasing transport in the fluid. As the Keulegan-Carpenter number is let beyond the convergence limit defined in Aureli et al. (2012), a similar flow behavior is identified. Vortical structures are produced near the edges of the plate every half-wave displacement and subsequently shed and advected into the fluid, generating a flow field that is highly dependent on KC. At KC = $\pi/10$, higher vorticity is observed only in the fluid portions around the edges of the lamina,


Figure 4.2: Velocity (left panels) and vorticity (right panels) contours for $\beta = 976$ and KC = $\pi/10$.



Figure 4.3: Velocity (left panels) and vorticity (right panels) contours for $\beta = 976$ and KC = $3\pi/5$.



Figure 4.4: Velocity (left panels) and vorticity (right panels) contours for $\beta = 976$ and KC = $6\pi/5$.

with most of the vortex dynamics taking place at the tips and the trailing region. In Figures 4.3 and 4.4, KC is representative of oscillation strokes comparable to the width of the lamina and the size of each pair of counter-rotating vortices generated at the tips of the plate is one or two orders of magnitude bigger than the previous case. Due to their high rotational inertia, these large vortical structures are dissipated at a much lower pace, allowing them to interact with other vortices, as well as with the oscillating plate. For instance, the vorticity contour in Figure 4.4 at t/T = 2.875denotes the presence of six areas of elevated vorticity, two behind and four ahead of the lamina as it travels upwards, and the corresponding velocity contour suggests two additional areas of weaker intensity positioned in the trailing region. Therefore, it can be inferred that when KC is large, vortices generated in a given oscillation cycle are shed and advected into the fluid for an amount of time that can span one or more periods of oscillations. Vortices shed at t/T = 2.5 interact with the lamina at $t/T \in (2.875, 3)$, and their loss of momentum due to the fluid-structure interaction can be observed at t/T = 3, where the two vortex cores have been broken. Since vorticity dynamics is the essential of non-linear damping mechanisms affecting the oscillations of slender structures in a viscous fluid, these phenomena are crucial in the characterization of the hydrodynamic loading, whose maxima are detected in correspondence of the vortex-structure interactions. Although SPH results in this analysis are based on a viscosity model that is intrinsically dissipative, vorticity transport phenomena do not suffer from this artificiality.

4.3.2 Pressure field

The evolution of the non-dimensional pressure, defined as $P(t)/(\rho f^2 B^2 \text{KC})$, is illustrated in Figures 4.5 and 4.6 for $\beta = 976$ and $\text{KC} = \pi/10$ and $6\pi/5$, respectively. Due to symmetry with respect to the z-axis, pressure contours are presented only for negative values of y and $t/T \in [2.5, 3]$. For $\text{KC} = \pi/10$, the pressure field is in good qualitative agreement with experimental results in Jalalisendi et al. (2014), where a Particle-Image-Velocimetry (PIV) study is carried out for similar values of the nondimensional pair (β , KC). At t/T = 2.5, the lamina has maximum velocity and it is surrounded by high pressure ahead of the leading side and low pressure behind the trailing side. As it decelerates towards the inversion point, part of the fluid in the trailing region starts impinging on the structure due to a change in relative acceleration and thus high pressure starts developing at t/T = 2.625, whereas low pressure



Figure 4.5: Evolution of the pressure field for $\beta = 976$ and KC = $\pi/10$.



Figure 4.6: Evolution of the pressure field for $\beta = 976$ and KC = $6\pi/5$.

is observed near the leading side. The lamina reaches z(t)/A = -1 at t/T = 2.75, where maxima in both high and low pressures are observed due to a large momentum exchange of the fluid with the plate. Successively, it travels upward, finalizing the third oscillation at t/T = 3, with lower pressure magnitudes on both sides.

A different scenario is observed for $\text{KC} = 6\pi/5$, where, in the initial descent, there are still two distinct areas of high and low pressures, although the latter is concentrated in a large circular region corresponding to a vortex core. The interaction of this vortical depression with the surrounding flow field yields coexisting areas of high and low pressure that, in turn, interact with the leading side of the plate, as seen in Figure 4.6 for t/T = 2.875 and t/T = 3. Such pressure gradients are identified as the leading mechanism characterizing the hydrodynamic actions on the lamina, due to the main contribution of pressure to the total fluid force. As for the velocity and vorticity fields, the scaling of the pressure with the Keulegan-Carpenter number can be assessed as a consequence of flow non-linearities driven by rising vortex shedding and convection.

4.3.3 Total hydrodynamic force

In this section, the extraction of the hydrodynamic force, F(t, y, z), from SPH simulations is carried out using results from DualSPHysics v3.0. Simulation data are



Figure 4.7: Hydrodynamic force, F(t), and displacement, z(t), as a function of the non-dimensional time for $\beta = 1952$. Forces are in Nm⁻¹. Displacement (blue line) is scaled to allow for comparison. (a) Time history of F(t) for KC = $\pi/25$ with domain size $3B \times 2B$ (red vertically-dashed line), $3B \times 3B$ (red dashed line), $4B \times 4B$ (red solid line), $5B \times 5B$ (black solid line). (b) Time history of F(t) for KC = $3\pi/5$ with domain size $5B \times 5B$ (red vertically-dashed line), $10B \times 10B$ (red dashed line), $15B \times 15B$ (red solid line), $20B \times 20B$ (black solid line).

then used to compute added mass and damping coefficients via MATLAB[®] scripts and other post-processing routines. F(t, y, z) is obtained from the SPH momentum equation by first computing the acceleration vector of each water particle in the first layer of fluid surrounding the lamina. This value is then multiplied by the fluid particle's mass and reversed in sign so that the resulting vector is sought as the force exerted by the water on the plate. Since force cancellation in the y direction is observed due to symmetry, only the z component of the acceleration is considered in the computation of the hydrodynamic force, referred to as F(t) in the remainder of this work.

The size of the computational domain affects the quality of the solution due to the presence of boundary effects and therefore it must be selected carefully in order to also balance the overall computational cost. In Figure 4.7 the time history of the fluid force for the third and fourth oscillation cycles is given for $\beta = 1952$ and two distinct amplitudes, $\text{KC} = \pi/25$ and $\text{KC} = 3\pi/5$, respectively. Given the restriction $N_f/A \ge 10$ on the particle resolution, the choice of the optimal domain size that minimizes the interference of the flow field with the solid boundaries is iterative, retaining an acceptable computation cost. The sensitivity of the fluid force can be observed as the shape of force patterns changes according to the domain size, which is varied as a function of the plate width, B. In the two examples in Figure 4.7, a domain of $5B \times 5B$ is selected for the small KC case on the left, and $20B \times 20B$ for the large KC case on the right, given the satisfactory convergence of the solution.

Following the approach introduced by Keulegan and Carpenter (1958) for oscil-



Figure 4.8: Coefficients of the fundamental harmonic, A_1, B_1 , as a function of the Keulegan-Carpenter number.

lating cylinders in a sinusoidal flow field, and elaborated in Bidkar et al. (2009) for flat plates oscillating in air, a fit of the fluid force with a Fourier series is carried out as follows

$$\frac{F(t)}{\rho f^2 B^3 \text{KC}^2} = A_0 + \sum_{n=1}^5 A_n \cos(2\pi n f t) + B_n \sin(2\pi n f t)$$
(4.7)

where F(t) is non-dimensionalized and terms in the Fourier series are retained up to the fifth harmonic, as higher harmonics are null. Results are illustrated in Figures 4.8 and 4.9, where the relationship between the Fourier coefficients and KC is illustrated for each harmonic and all four values of β . In Figure 4.8, plots of coefficients of the fundamental harmonic, $(-A_1, B_1)$ are presented, where A_1 is the out-of-phase component of F(t) accounting for the phase shift between the hydrodynamic force



Figure 4.9: Coefficients A_0 , A_2 , ..., A_5 and B_2 , ..., B_5 as a function of the Keulegan-Carpenter number for $\beta = 244$, $\beta = 488$, $\beta = 976$, and $\beta = 1952$. (a)-(d) absolute value of the cosine coefficients. (e)-(h) absolute value of the sine coefficients.

and the acceleration of the lamina, while B_1 represents the in-phase counterpart. As mentioned earlier, the former is associated with hydrodynamic damping, and therefore identifies a damping coefficient, whereas the latter defines an added mass coefficient. Results in Figure 4.8 are in qualitative agreement with Bidkar et al. (2009), where the Fourier coefficient identifying the damping therein does not vary much with KC, whereas an inverse proportionality is observed between KC and the added mass coefficient. The magnitude of even coefficients (A_2, A_4) and (B_2, B_4) in Figure 4.9 is generally very small and the mean force over an oscillation cycle can be seen negligible from coefficient A_0 . Thus it can be concluded that the fluid force preserves the half-wave symmetry of the displacement and even harmonics are neglected in the force computation. Coefficients of the fifth harmonic A_5, B_5 , are also negligible. Nevertheless, the arising influence of the third harmonic with increasing KC can be noticed from coefficients A_3 , B_3 in Figure 4.9 (b)-(d) and Figure 4.9 (f)-(h). Specifically, the magnitude of coefficients A_3, B_3 reaches values in the order of A_1, B_1 for high KC and $\beta \geq 488$. Although this effect is seen to grow fast with the frequency and amplitude of oscillations, it is not taken into consideration since its influence is still moderate in the chosen range of parameters. Therefore, the hydrodynamic function is modeled exclusively by considering coefficients of the first harmonic, using the phasor notation as in Equation (4.5) and rewriting Equation (4.7) in the frequency domain as

$$\hat{F}(\omega) = \rho f^2 B^3 \mathrm{KC}^2(B_1 + iA_1) \tag{4.8}$$

It is worth remarking that this approximation is made to assess the hydrodynamic load in Equation (4.5) using a simple mathematical function, $\Theta(\beta, \text{KC})$, whose validity is restricted to selected values of the non-dimensional pair (β, KC).

4.3.4 Hydrodynamic function

In order to obtain an expression for the hydrodynamic function using data from SPH simulations, Equation (4.5) is combined with Equation (4.8) and obtain

$$\Theta(\beta, \mathrm{KC}) = \frac{2}{\pi^2} \mathrm{KC} \left(B_1 + iA_1 \right) \tag{4.9}$$

where, similarly to Equation (4.6), $\Theta(\beta, \text{KC})$ incorporates the hydrodynamic function $\Gamma(\beta)$ characterizing infinitely small oscillations (Sader, 1998), and a correction



Figure 4.10: Real (top) and imaginary (bottom) parts of $\Delta(\beta, \text{KC})$.

term that is seek for large amplitudes. In Figure 4.10 the real and imaginary part of Equation (4.9) subtracted by $\Gamma(\beta)$ is plotted as a function of the Keulegan-Carpenter number. The result is $\Delta(\beta, \text{KC}) = \Theta(\beta, \text{KC}) - \Gamma(\beta)$. Consistently with Aureli et al. (2012), Re[$\Delta(\beta, \text{KC})$] is close to zero for most of the considered pairs (β, KC), suggesting that the added mass coefficient is well approximated by $\Gamma(\beta)$. At large KC values, discrepancies of the calculated Re[$\Delta(\beta, \text{KC})$] with respect to values predicted by $\Gamma(\beta)$ are observed in the range [0.4 – 0.8] and can be attributed to small errors in the computation of the hydrodynamic force via SPH. As will be shown later, a few of the pairs (β, KC) considered in this work are within the validity range of Equation (4.6) and direct comparison between the two predictions show a slight overestimation of both added mass and damping coefficients using SPH. On the other hand, it is noted that the variation of Re[$\Delta(\beta, \text{KC})$] with KC is marginal compared with that of Im[$\Delta(\beta, \text{KC})$] in the bottom part of Figure 4.10, especially at large amplitudes. Therefore, the attention is focused solely on changes in the hydrodynamic damping coefficient, following the approach of Aureli et al. (2012) to cast an equation



Figure 4.12: Linear correlation between numerical findings from SPH simulations (squares) and results from the fit applied to

Equation (4.9) (line).

Figure 4.11: Surface representation of the damping coefficient, $-\text{Im}[\Theta(\beta, \text{KC})]$, i.e. the imaginary part of Equation (4.9).

for the hydrodynamic function as

$$\Theta(\beta, \mathrm{KC}) = \Gamma(\beta) + \Delta(\beta, \mathrm{KC}) = \Gamma(\beta) + i \ a \ \beta^b \left(\frac{\mathrm{KC}}{2\pi}\right)^c \tag{4.10}$$

3.

 $\operatorname{Im}[\Delta(\beta,\operatorname{KC})]$

Coefficients a, b, c are computed via non-linear least-squares fitting with the MATLAB[®] Trust Region optimization algorithm. Numerical values are obtained as a = -0.821, b = 0.305 and c = 1.150, with a coefficient of determination $R^2 = 0.9822$. Therefore

$$\Theta(\beta, \text{KC}) = \Gamma(\beta) - i \ 0.821 \ \beta^{0.305} \left(\frac{\text{KC}}{2\pi}\right)^{1.150}$$
(4.11)

It is worth noting that Equation (4.11) is formulated to provide an estimation of the hydrodynamic force experienced by the oscillating plate for moderate-to-high frequency and oscillation strokes from few percents to lengths comparable with the lamina width. Nonetheless, the use of a power law such as that in Equation (4.10) imposes $\Delta(\beta, \text{KC}) \rightarrow 0$ as $\text{KC} \rightarrow 0$, allowing to recover the linear solution $\Gamma(\beta)$ for infinitely small oscillations.

A surface plot is considered in Figure 4.11, with the x-axis representing the nondimensional frequency, the y-axis containing the Keulegan-Carpenter number and values of $-\text{Im}[\Theta(\beta, \text{KC})]$ on the z-axis. The influence of large amplitudes on the hydrodynamic damping can be detected by looking at the monotonic increase of $-\text{Im}[\Theta(\beta, \text{KC})]$ with KC, regardless of the frequency. Through most of the considered amplitudes, the damping coefficient is also monotonically increasing with β , except for small amplitudes, where an inverse proportionality between the damping coefficient and β is observed, consistently with results in Sader (1998).

To assess the validity of the proposed correction, results from Equation (4.11)are compared with those of Equation (4.6), using values of β from this work and values of KC corresponding to the validity limit of Equation (4.6), beyond which this model is believed to give a better estimation. Results from the comparison are summarized in Table 4.2. Despite KC values in Table 4.2 being relatively modest with respect to the range of amplitudes analyzed in this work, a good agreement is still observed, with largest percent differences of 10.7% and 7.47% in the lowest and highest frequency case, respectively. As mentioned earlier, such small discrepancies can be attributed to SPH features such as the weak-compressibility or the choice of boundary conditions, and influence the results only at the lowest amplitude case. This can indeed be observed from the Fourier coefficients in Figure 4.9 (a) and Figure 4.9 (d), where the presence of higher harmonics is observed for the lowest KC, whereas, at the next KC, Fourier coefficients other than A_3 have already decayed to zero. On the other hand, differences as little as 1.78% and 4.29% are observed for mid-range frequencies, highlighting the overall agreement between the two models within the validity of Equation (4.6). In Figure 4.12 a graph of the linear correlation of SPH data in terms of $-\text{Im}[\Delta(\beta, \text{KC})]$ with the power law of Equation (4.11) is provided. Once again, a slight over-estimation of the damping coefficient predicted by SPH can be observed within unity, whereas for values of the abscissa greater than 1, numerical data fit the model fairly well.

β	KC	Equation (4.6)	Equation (4.11)	% Difference
244	0.3680	0.1860	0.1680	10.70
488	0.2384	0.1313	0.1260	4.29
976	0.1546	0.0929	0.0946	1.78
1952	0.1002	0.0656	0.0709	7.47

Table 4.2: Comparison of the proposed hydrodynamic function in Equation (4.11) with that of Aureli et al. (2012) in Equation (4.6).



Figure 4.13: Comparison of added mass (left) and damping (right) coefficients from present findings with other works in the literature.

4.3.5 Correlation with existing results

A comprehensive comparison of the results is presented in Figure 4.13, where numerical findings from SPH are superimposed with data from cited literature, rearranged in the form of $\operatorname{Re}[\Theta(\beta, \mathrm{KC})]$ (left) and $-\operatorname{Im}[\Theta(\beta, \mathrm{KC})]$ (right) as a function of $\text{KC}/2\pi$. Values of the added mass coefficient are in good agreement for $\text{KC}/2\pi \leq 0.1$, where SPH results overlap with most of the other data. In this range of amplitudes, a slight over-estimation is only observed for data reported in Falcucci et al. (2011). For KC/ $2\pi > 0.1$, however, values of Re[$\Theta(\beta, \text{KC})$] in this work stay within [1-2]and experiments in Singh (1979) and Graham (1980) show the same trend up to $KC/2\pi = 1$, beyond which a sharp decrease of the added mass coefficient is noted. Conversely, experimental findings in Aureli and Porfiri (2010) and Bidkar et al. (2009) suggest that $\operatorname{Re}[\Theta(\beta, \mathrm{KC})]$ should increase. In the plot on the right of Figure 4.13, the dependence of the damping coefficient, $-\text{Im}[\Theta(\beta, \text{KC})]$, on KC is observed. Unlike the added mass behavior, a common trend characterizes all datasets, as $-\text{Im}[\Theta(\beta, \text{KC})]$ increases monotonically with KC, regardless of the frequency or the solution method. Good agreement is observed between SPH data and results in Bidkar et al. (2009), with both datasets extending the numerical and experimental findings in Aureli et al. (2012), Falcucci et al. (2011) and Jalalisendi et al. (2014) to larger amplitudes. Similar behavior is exhibited by data from Aureli and Porfiri (2010), Graham (1980) and Singh (1979), although the increase of the damping coefficient with KC is not as fast.



(a) 3-D sketch of a vibrating cantilever beam. (b) 2-D representation of the thin crosssection submerged in a viscous fluid with a free surface.

Figure 4.14: Nomenclature for the vibrational problem.

4.4 Results for a fluid with a free surface

4.4.1 Boundary integral formulation

An infinitely thin plate representative of a cantilever beam cross-section of width, b, and undergoing zero-amplitude oscillations in a viscous fluid with a free surface is studied. A Cartesian coordinates system O(x, y, z) is defined, where x is along the cantilever axis, y is through b, and z is in the direction of motion, as shown in Figure 4.14a. The effect of varying the depth of submergence is investigated via the non-dimensional parameter $H = h_0/b$, where h_0 is the distance of the lamina from the undisturbed free surface, as in Figure 4.14b.

A boundary integral formulation is implemented, in which the solid boundary and the flow domain are discretized along the boundary surfaces rather than the volume in three-dimensional space, and along the lines for two-dimensional cases. The method is well-established (Tuck, 1969) and is used for several problems of computational physics. A viscous incompressible Newtonian fluid moving at low Reynolds numbers is considered, with

$$Re = \frac{\omega b^2}{4\nu} \tag{4.12}$$

where ν is the kinematic viscosity and ω is the radian frequency, $\omega = 2\pi f$. Under

these conditions, the flow physics can be modeled using the unsteady Stokes flow theory, where the inertial effects are negligible with respect to the viscous terms. The Navier-Stokes momentum and continuity equations are therefore written as

$$\nabla \cdot \mathbf{u} = 0 \tag{4.13}$$

$$\frac{\partial \mathbf{u}}{\partial t} = -\frac{1}{\rho} \nabla P + \mathbf{f} + \nu \nabla^2 \mathbf{u}$$
(4.14)

where $\mathbf{u} = v(y, z, t)\mathbf{\hat{j}} + w(y, z, t)\mathbf{\hat{k}}$ is the velocity vector, P(y, z, t) denotes the pressure and $\mathbf{f} = 0\mathbf{\hat{j}} + g\mathbf{\hat{k}}$ is the gravitational force vector. The displacement vector is defined as $\mathbf{r} = y\mathbf{\hat{j}} + z\mathbf{\hat{j}}$ and express the gravitational force as a gradient of the potential energy. With this notation, Equation (4.14) becomes

$$\frac{\partial \mathbf{u}}{\partial t} = -\frac{1}{\rho} \nabla P + \nabla (gz) + \nu \nabla^2 \mathbf{u}$$
(4.15)

and incorporating the gradient of the potential energy in the pressure term

$$\frac{\partial \mathbf{u}}{\partial t} = -\frac{1}{\rho} \nabla P' + \nu \nabla^2 \mathbf{u}$$
(4.16)

with $P'=P-\rho gz.$ The stream function, $\Psi(y,z,t)$, is introduced, satisfying the following relations

$$v(y,z,t) = \frac{\partial \Psi(y,z,t)}{\partial z}$$
(4.17)

$$w(y, z, t) = -\frac{\partial \Psi(y, z, t)}{\partial y}$$
(4.18)

In the frequency domain, the boundary integral formulation for the stream function in unsteady Stokes flow conditions is given by

$$\Psi(y', z', \omega) = \int_{C} [\Psi(y, z, \omega)G_n(y, z; y', z') - \Psi_n(y, z, \omega)\Omega(y, z; y', z') - \zeta(y, z, \omega)\xi_n(y, z; y', z') + \mu^{-1}P'(y, z, \omega)\xi_l(y, z; y', z')]dl$$
(4.19)

The subscript n indicates the normal derivative operator applied to the boundary C of the fluid domain, whereas the subscript l indicates the tangential derivative along the boundary. Normal and tangential directions are depicted in Figure 4.14b, where the integral boundaries for the lamina, (C^{b+}, C^{b-}) , and the free surface, C^{fs} are indicated.

Terms in the contour integral of Equation (4.19) are defined as follows

$$G = \frac{1}{2\pi} \log R \tag{4.20}$$

$$\Omega = -\frac{1}{2\pi} K_0(\sigma R) \tag{4.21}$$

$$\xi = -\frac{1}{2\pi\sigma^2} [\log R + K_0(\sigma R)]$$
(4.22)

$$\zeta = -\nabla^2 \Psi \tag{4.23}$$

where

$$R = \sqrt{(y - y')^2 + (z - z')^2}$$
(4.24)

$$\sigma^2 = -i\omega/\nu \tag{4.25}$$

and K_0 is the modified Bessel function of the second kind. Equations (4.20) to (4.22) represent two-dimensional Green's functions in free space, whereas Equation (4.23) is the *x*-component of the vorticity vector. Applying Equation (4.19) to the configuration in Figure 4.14b yields

$$\Psi(y',z',\omega) = \int_{-\infty}^{\infty} \left[-\Psi^{fs}(y,0,\omega)G_z(y,0;y',z') + \Psi^{fs}_z(y,0,\omega)\Omega(y,0;y',z') - \zeta^{fs}(y,0,\omega)\xi_z(y,0;y',z') - \mu^{-1}[P^{fs}(y,0,\omega) - gz^{fs}]\xi_y(y,0;y',z')] \, dy + (4.26) \right]$$

$$\int_{-b/2}^{b/2} \left[\Delta\zeta^b(y,h_0,\omega)\xi_z(y,h_0;y',z') - \mu^{-1}\Delta P^b(y,h_0,\omega)\xi_y(y,h_0;y',z')\right] \, dy$$

where the superscript f^s refers to free-surface quantities and $\Delta \zeta^b$, ΔP^b are the vorticity and pressure jumps across the beam, respectively. No-slip boundary conditions are imposed as

$$v(y', h_0, \omega) = 0; \quad w(y', h_0, \omega) = W_0$$
(4.27)

with W_0 representing the maximum value of the one-dimensional oscillation velocity. The zero-stress condition at the free surface is given by

$$\Psi_{z'z'}(y',0,\omega) - \Psi_{y'y'}(y',0,\omega) = 0 \tag{4.28}$$

$$\Psi_{z'y'}(y',0,\omega) = \Psi_{y'z'}(y',0,\omega) = 0 \tag{4.29}$$

Furthermore, the relative pressure on the free surface is considered null

$$P^{fs}(y,0,\omega) = 0 (4.30)$$

and define a reference for the gravitational potential as

$$gz^{fs} = gH_g \tag{4.31}$$

where H_g is a constant value found iteratively for the problem under investigation. After differentiating Equation (4.26) and applying boundary conditions in Equations (4.27) to (4.30), a non-dimensional system of matrix equations is obtained with $\bar{\Psi}^{fs}, \bar{\Psi}_z^{fs}, \bar{\Delta}\zeta^{fs}$ and $\bar{\Delta P}^b$ being the unknown vectors in non-dimensional form. A solution to this system of equations is presented in the next section and validated against numerical simulations performed via Smoothed Particle Hydrodynamics.

4.4.2 Added mass and damping for infinitely small oscillations

To study the hydrodynamics of the proposed problem, the total fluid force per unit length is represented in the frequency domain as



Figure 4.15: Non-dimensional hydrodynamic force as a function of the non-dimensional depth of submergence, H.

$$\hat{F}(\omega) = \frac{\pi}{4} \rho \; \omega^2 b^2 \Lambda(\omega) A(\omega) \tag{4.32}$$

or, in dimensionless form, using the Reynolds number as

$$\hat{F}(\omega) = -i\pi \text{Re}\Lambda(\omega)$$
(4.33)

where $A(\omega)$ is the infinitesimal amplitude and $\Lambda(\omega)$ is a non-dimensional hydrodynamic function, whose real and imaginary parts are proportional to the inertial and viscous loading, respectively. Following Tuck (1969), $\Lambda(\omega)$ can be expressed in complex notation as

$$\Lambda(\omega) = k + ik' \tag{4.34}$$

where k defines an added mass coefficient and k' is the damping coefficient. In Figure 4.15, $\overline{\hat{F}}(\omega)$ is shown for Re = 1 and several values of H. The magnitude of the fluid force increases sharply as the depth of submergence is in the range of the lamina width, b, or smaller. Conversely, as H increases, the hydrodynamic load exhibits an asymptotic behavior, approaching the limit case $H \to \infty$ in Tuck (1969).

In Figure 4.16, the computed added mass and damping coefficients are presented for selected values of H and a broad range of Reynolds numbers. A double logarithmic



Figure 4.16: Total hydrodynamic force per unit length expressed as added mass and damping coefficients as a function of the Reynolds number.

scale is used to allow for an easier comparison of the results with the limit case $H \to \infty$, herein represented with a black dashed curve. The presence of the free surface influences the hydrodynamic force, especially for low-to-moderate values of the Reynolds number. At fixed Re, the added mass, acting in phase with the acceleration of the lamina, varies with H, increasing as the lamina approaches the free surface. A similar behavior is observed for the damping coefficient, acting out of phase with the acceleration. As one can observe from Figure 4.16b, varying the depth of submergence corresponds to a substantial change among values of the damping coefficient in the low-to-moderate Reynolds regimes, whereas, as Re increases, k' is less affected by the depth of submergence. These results demonstrate that, at relatively low values of Re, the presence of a free surface near the oscillating plate is responsible for an increase in viscous dissipative effects. Asymptotic behavior of both added mass and damping coefficient is observed for values of the Reynolds number near 10^3 , where k and k' tend to overlap with the theoretical prediction in Tuck (1969), regardless of the depth of submergence.

To further validate the results, Smoothed Particle Hydrodynamics simulations are carried out for Re = 785 and varying H in [0.25-2]. The choice of the Reynolds number is mainly dictated by the intensity of spurious oscillations in the Weakly-Compressible SPH (WCSPH) pressure and density fields, which degrade the quality of the solution at small frequencies and amplitudes of oscillation. The thickness of the lamina as well as the oscillation amplitude are set to 0.01*b*, as both must be finite in the simulation framework. This value is chosen such that the lamina is fairly one-dimensional and the influence of finite-amplitude effects is small. Simulations are



Figure 4.17: Numerical results from BEM (upward triangles) and SPH (dots).

run for approximately five oscillation cycles, and discard the first two or three cycles due to non-steady behavior. The total SPH hydrodynamic force is fitted with the first harmonic of its Fourier series approximation. Computed Fourier coefficients are then rearranged in terms of k and k', and presented in Figures 4.17a and 4.17b as a function of H. In both graphs, results from the boundary integral formulation are superimposed with SPH simulations.

Although some discrepancies are observed between the two methods, the overall behavior of the two coefficients seems to follow the same physical trend, especially for the damping coefficient. Both methods can in fact predict an increase of k' for H < 1, while variations in k' at larger depths are not substantial. However, SPH simulations appear to overestimate the damping coefficient, especially at small values of the submergence depth. This is attributed to finite-amplitude effects, for which corrections to the hydrodynamic function are provided in Aureli et al. (2012) for the unbounded fluid case. The added mass, k, which is strictly related to the behavior of k', exhibits an asymptotic trend with increasing depths for both approaches. For small H, however, BEM results are fairly constant while SPH simulations show an increase of the added mass coefficient as well.



Figure 4.18: Contours of velocity magnitude for $\beta = 500$, KC = 0.628 and $\chi = 2.50$.

Effects of finite amplitudes: a preliminary study

4.4.3

As a first attempt to understand the physics of the problem described in the previous section for finite-amplitude oscillations, a preliminary set of SPH simulations is carried out with a finite value of KC. Other non-dimensional parameters governing this problem are the frequency, β , and depth of submergence, χ , corresponding to the H used in Section 4.4. The geometry of the lamina is the same used in Section 4.2. Analysis of the flow physics around the lamina is conducted from the third cycle of oscillation, regarding the first two as settling-in periods.

Figures 4.18 and 4.19 show the evolution of the velocity magnitude along one period of oscillation for $\beta = 500$, KC = 0.628 and two values of the depth, $\chi = 2.50$ and 0.50, respectively. In Figure 4.18, nonzero values of the velocity magnitude are observed just around the oscillating plate, while the remaining fluid is characterized by



Figure 4.19: Contours of velocity magnitude for $\beta = 500$, KC = 0.628, and $\chi = 0.50$.

a much smaller velocity, gradually reaching zero as the fluid moves further away from the lamina. In agreement with findings in Section 4.3.1, the maximum value of the velocity magnitude is observed when the lamina approaches z(t)/A = 0 at t/T = 2, t/T = 2.5 and t/T = 3, whereas minima are observed for the maximum amplitude configuration, i.e. t/T = 2.25 and t/T = 2.75.

Figure 4.19 shows the third cycle of oscillation for $\chi = 0.50$, with significant differences with respect to the previous case. Highly non-harmonic velocity profiles are observed, with velocity contours varying from cycle to cycle, mainly due to the formation and breaking of waves at the free surface. While for $\chi = 2.50$ the support of the velocity field is confined in the vicinity of the lamina, this is not the case for small depths of submergence, where the motion of the fluid continues further away in the fluid domain. Qualitative assessment of non-linearities due to the presence of a free



Figure 4.20: Time history of the fluid force for $\beta = 500$ and several values of the depth.

surface can be made by the temporal identification the maxima and minima in the velocity field. Due to the presence of waves, t/T = 2.25 and t/T = 2.75 are no longer identifying minima for the velocity field, as opposite to the case of large χ . Similar conclusions can be drawn for velocity maxima.

Hydrodynamic force extraction via DualSPHysics v2.1 is presented next for selected values of the triad (β , KC, χ). The total fluid force per unit width is nondimensionalized with the reference quantity, $F_{REF} = \rho \omega^2 b^3$, and a plot over two periods of oscillations is presented. In Figure 4.20 the time history of the hydrodynamic force is shown for $\beta = 500$, two amplitude values, KC = 0.0628 and KC = 0.628, and χ in the range [0.50-3.00]. It is noted that in both cases the hydrodynamic load for small depths of submergence has a marked phase shift with respect to higher χ values, denoting an increase of the out-of-phase component of the fluid force with the depth parameter. This suggests that the hydrodynamic damping increases as the lamina approaches the free surface, coherently with the coefficient k' in Figure 4.16b. For greater depths, the force curves tend to collapse onto each other to a fair extent, emphasizing a negligible effect of the χ parameter already at values of $\chi = 1.75$ and greater.

Chapter 5

Variable resolution SPH

The extensive application of the SPH method to the problems described in the previous chapters has enabled the author to identify some of the key SPH shortcomings with the present state of the code and to choose directions for his research work. Results in this and the following chapter are obtained after improving the code with variable particle resolution and new types of boundary conditions, two of the major challenges of the SPH methodology which are subject of considerable ongoing research. For the former topic, this chapter is focused on testing the splitting and coalescing algorithm originally devised in Vacondio et al. (2013) and currently under implementation in DualSPHysics by the main developers. While the use of adaptive grids has proven successful in other CFD techniques, the vast majority of SPH simulations are presently solved using uniformly sized particles. This is inefficient from a computational point of view and imposes several limitations to the use of SPH, some of which have been mentioned in Chapter 3. From preliminary results, it is corroborated that the ability of using a large number of particles only in specific areas of the domain where high resolution is desired yields higher accuracy while maintaining reasonable computation costs.

5.1 The particle splitting and coalescing technique

The particle splitting and coalescing technique chosen for the implementation of variable resolution in DualSPHysics is based on a SPH discretization scheme that employs particles with non-uniform mass and smoothing length as in Vacondio et al. (2013). The algorithm has been designed to ensure conservation of mass, linear and angular momentum in presence of variable h. Moreover, changes in the local density and velocity distributions due to the refinement and derefinement of particles are dealt



Figure 5.1: Optimal particle splitting pattern with current code implementation (Vacondio et al., 2015).

with by minimizing the density estimate error and smoothing out discontinuities at the coarse/refined interfaces.

The process of splitting is activated by the detection of coarse particles in a userdefined area of the computational domain where higher resolution is required. Figure 5.1 shows two of the possible configurations for the splitting of the mother particle (red), with smoothing length h, into daughter particles (blue), with smoothing length αh , where α is a scalar parameter in [0 - 1] named smoothing ratio. ε is simply defined as the relative distance between mother and daughter particles, also varying between 0 and 1. The hexagonal pattern on the left panel of Figure 5.1 has been identified as the optimal choice for two-dimensional problems (Feldman and Bonet, 2007), whereas Vacondio et al. (2015) report better efficiency and negligible errors in the density estimate for 3-D cases using the icosahedron pattern shown on the right. Regardless of the specific geometry chosen for the refinement, the use a fixed pattern is beneficial in that it reduces the degrees of freedom of the problem: the

Table 5.1: Smoothing ratio, α , and relative distance, ε , of daughter particles found by Vacondio et al. (2015).

	2-D	3-D
Pattern	Hexagonal	Icosahedronal
Parameter α	0.9	0.89
Parameter ε	0.4	0.4

	Mother particle, i	Daughter particles , $k = 1, \ldots, M$
Mass	m_i	$\sum_k m_k$
Kinetic energy	$0.5 \ m_i \mathbf{u}_i \cdot \mathbf{u}_i$	$0.5 \sum_k m_k \mathbf{u}_k \cdot \mathbf{u}_k$
Linear momentum	$m_i \mathbf{u}_i$	$\sum_k m_k \mathbf{u}_k$
Angular momentum	$\mathbf{x}_i \times m_i \mathbf{u}_i$	$\sum_k \mathbf{x}_k \times m_k \mathbf{u}_k$

 Table 5.2: Properties of mother and daughter particles (Feldman and Bonet, 2007).

number of daughter particles and their position are now fixed, whilst their mass, velocity and smoothing length are still needed. A minimization problem is solved to find the mass value of the new particles, where the objective function to minimize is the error between the refined and original local density field. In this process, the set of coefficients (α, ε) is set arbitrarily and the optimal pairs for each case is shown in Table 5.1, with further details on the optimization process in Vacondio et al. (2012) and Vacondio et al. (2013). Moreover, fundamental properties of mother and daughter particles are summarized in Table 5.2: while daughter particles are characterized by a smaller mass than m_i , their velocity must be equal to that of the mother particle in order to preserve both the total momentum and kinetic energy, as pointed out in Feldman and Bonet (2007).

The coalescing process is triggered by pairs of particles, closest to each other, that are located in a user-defined area where high resolution is no longer needed. The position of the new particle created by the merge of the two smaller ones is determined as a weighed average

$$\mathbf{x}_{\text{new}} = \frac{m_1 \mathbf{x}_1 + m_2 \mathbf{x}_2}{m_1 + m_2} \tag{5.1}$$

This procedure is repeated at successive time steps if the mass of the new particle is still below the coalescing mass threshold defined by the user. Computation of the velocity vector, \mathbf{u}_{new} , follows that of the position and this approach guarantees conservation of mass and momentum as the particles merge together. The last quantity to compute is the new smoothing length of the coarse particle, h_{new} . Similarly as before, this is done by monitoring the error in the density estimate before and after the merge of the two particles. If a Wendland kernel is chosen, the value for the new smoothing length is given by (Vacondio et al., 2015)

$$h_{\rm new} = h_{\rm new}(m_1, m_2, W_{\rm new,1}, W_{\rm new,2}) = \left(\frac{1}{\theta_W} \frac{m_1 + m_2}{m_1 W_{\rm new,1} + m_2 W_{\rm new,2}}\right)^{1/d}$$
(5.2)

with $W_{\text{new},1} = W_{\text{new},1}(h_1)$ and $W_{\text{new},2} = W_{\text{new},2}(h_2)$. The coefficient θ_W is 7/4 π in 2-D (d = 2) and 21/16 π in 3-D (d = 3). Both refined and coarse particles evolve according to a slightly modified set of SPH equations. The continuity equation becomes

$$\frac{d\rho_k}{dt} = \sum_{l=1}^{N} m_l \mathbf{u}_{k,l} \cdot \nabla W_l(\mathbf{x}_k, h_l) + 2\delta h_i \sum_{l=1}^{N} m_l \bar{c}_{k,l} (\kappa_k - 1) \frac{1}{\mathbf{x}_{k,l} + \eta^2} \cdot \nabla W_l(\mathbf{x}_k, h_l)$$
(5.3)

where the only major difference with respect to Equation (2.37) is in the exchange of h_k with h_l for the kernel support. This has been seen to improve the interpolation accuracy in the splitting procedure. Consistently, the momentum equation is derived from Equation (5.3) in a variational manner

$$\frac{d\mathbf{u}_{k}}{dt} = \sum_{l=1}^{N} \frac{m_{l}}{\rho_{l}\rho_{k}} \Big(P_{k} \cdot \nabla W_{l}(\mathbf{x}_{k}, h_{l}) + P_{l} \cdot \nabla W_{k}(\mathbf{x}_{l}, h_{k}) \Big) + \sum_{l=1}^{N} \frac{m_{l}}{\rho_{l}\rho_{k}} \Big(T_{k,l} \cdot \nabla W_{l}(\mathbf{x}_{k}, h_{l}) \Big) + \mathbf{g} \Big)$$
(5.4)

5.2 Preliminary results

5.2.1 Dam break

The first study presented herein concerns the 2-D free surface flow caused by a dam break. This is considered one of the classical SPH benchmark cases and, as such, it is chosen for investigating the behavior of the splitting and coalescing algorithm.

The numerical set-up is illustrated in Figure 5.2. A column of water with height H = 1 ft is enclosed in a basin with dimensions $4/3H \times 4H$. The absence of a lateral wall blocking the water column causes the fluid to collapse under the action of gravity, mimicking the failure of a dam wall. Higher resolution is imposed in two refinement areas of the computational domain, both located at the bottom right corner of the basin, where strong hydrodynamic forces are expected due to the high momentum of the water approaching this area. In the first refinement area, particles are split by imposing a refined mass range of $(0.25m_{\text{init}} < m_{\text{ref}} < 0.5m_{\text{init}})$, where m_{init} is the mass calculated with the initial particle spacing, Δx , in the coarse region. The



Figure 5.2: Nomenclature for the 2-D dam break problem.

resolution is further increased in the second refinement area, with a new mass range of $(0.1m_{\text{init}} < m_{\text{ref}} < 0.25m_{\text{init}})$. Thus particles entering the first refinement area are split up to 25% of m_{init} and further refined up to 10% of m_{init} in the successive region. Standard SPH equations and parameters are used, with a low 0.01 value for the artificial viscosity coefficient. This is done to isolate every possible numerical artifice that not is directly connected to the use of the variable resolution algorithm. Simulations are run for the following three cases:

- A case with no splitting/coalescing and initial spacing, Δx , equal to $3.333 \times 10^{-4} H$
- A case with splitting/coalescing and initial spacing, Δx , equal to $3.333 \times 10^{-4} H$
- A case with no splitting/coalescing and initial spacing, Δx , equal to $8.333 \times 10^{-5} H$

The flow evolution at several time instants is shown in Figures 5.3 to 5.10 for the velocity field and Figures 5.11 to 5.18 for the pressure field. The agreement between the flow in the refined regions (mid panel) and the flow in the same regions for the high-resolution case (bottom panel) is evident in all snapshots, whereas some discrepancies are noted and expected for the coarse case (top panel).

Particularly, the variable resolution algorithm allows to capture flow details and the prediction of the free-surface location is more accurate than results obtained without



Figure 5.3: Velocity field at time t = 0 s.



Figure 5.4: Velocity field at time t = 0.15 s.



Figure 5.5: Velocity field at time t = 0.40 s.



Figure 5.6: Velocity field at time t = 0.70 s.



Figure 5.7: Velocity field at time t = 0.99 s.



Figure 5.8: Velocity field at time t = 1.01 s.



Figure 5.9: Velocity field at time t = 1.05 s.



Figure 5.10: Velocity field at time t = 1.11 s.


Figure 5.11: Pressure field at time t = 0 s.



Figure 5.12: Pressure field at time t = 0.15 s.



Figure 5.13: Pressure field at time t = 0.40 s.



Figure 5.14: Pressure field at time t = 0.70 s.



Figure 5.15: Pressure field at time t = 0.99 s.



Figure 5.16: Pressure field at time t = 1.01 s.



Figure 5.17: Pressure field at time t = 1.05 s.



Figure 5.18: Pressure field at time t = 1.11 s.



Figure 5.19: Close-up on the velocity field around the basin right corner for both coarse resolution (top) and two levels of splitting (bottom).

splitting. For example, the plunging wave generated after that the water impacts the right wall of the basin is depicted in Figures 5.7, 5.8, 5.15 and 5.16 during first contact with the free-surface beneath. The moment of impact is anticipated in the coarse case at t = 0.99 s, while for both splitting and high-resolution cases the time of impact is t = 1.01 s. As the water flows back towards the left wall, fluid particles exit the refined areas, triggering the coalescing process. The accuracy of the solution starts to deteriorate, as can be seen by observing the splash evolution in Figures 5.8 to 5.10 and Figures 5.16 to 5.18.

A close-up of the flow around the bottom right corner of the basin is presented in Figure 5.19, highlighting the greater level of detail offered in the refined areas. In Table 5.3, the computational runtimes are 180, 311 and 1328 minutes for the coarse case with no splitting, coarse case with variable resolution and high-resolution case, respectively. Simulations are run on a single eVGA NVIDIA[®] GeForce Titan GPU.

	Coarse resolution	Variable resolution	Full high-resolution
Runtime [minutes]	180	311	1328
# particles	107,000	284,730	1,688,000
# split particles	0	2,074,800	0
# coalesced particles	0	1,681,943	0

Table 5.3: Simulation details for the dam break problem.



Figure 5.20: Nomenclature for the 2-D sloshing problem.

Although the overhead of the splitting procedure increases dramatically from CPU to GPU computing, simulation times are still far smaller than using a single 8- or 12-core CPU. An overall speed-up of over 4 times is achieved when using variable resolution to refine areas of interest instead of uniform high resolution everywhere.

5.2.2 Water sloshing under external tank excitation

The second test case to assess the efficacy of the variable resolution algorithm implemented in DualSPHysics is a 2-D sloshing case. Water sloshing is another benchmark problem very common in SPH (Souto Iglesias et al., 2004; Shao et al., 2012; Wang et al., 2013a) because it involves highly non-linear free-surface flows for which SPH is a well-suited choice. In this particular case, water sloshing in a rigid tank undergoing external horizontal excitation is simulated using the same parameters as in Faltinsen et al. (2000) for the validation of the results. The water tank in Figure 5.20 contains initially quiescent water with a height of H = 0.6 m and is subjected to a cosinusoidal motion along the horizontal direction given by

$$x(t) = x_0 + A\cos\left(\frac{2\pi t}{T}\right) \tag{5.5}$$

with A = 0.032 m, T = 1.5 s, and x(t) representing the instantaneous position of any

point of the tank with initial position, $x_0(t)$. Two levels of splitting are once again defined to enable high-resolution flow near the free-surface using similar mass ranges as in the dam break case in Section 5.2.1. Specifically, particles entering the first refinement area are split into daughter particles with mass at least equal to 50% and not smaller than 20% of the mother particle mass, while this value decreases to 20% and 5% for the second refinement area, respectively. Similarly to the previous test case, only the artificial viscosity is implemented to fully observe the behavior of the variable resolution algorithm, therefore the simulations can be considered inviscid. Three cases are run as follows:

- Splitting/coalescing deactivated and initial spacing, Δx , equal to $1.667 \times 10^{-2} H$
- Splitting/coalescing activated and initial spacing, Δx , equal to $1.667 \times 10^{-2} H$
- Splitting/coalescing deactivated and initial spacing, Δx , equal to $4.167 \times 10^{-3} H$

It can be noted that an overall higher value of Δx has been employed for this particular problem as opposed to the dam break case in Section 5.2.1, even though the length scale of the two cases is approximately the same. The main reason behind a larger inter-particle distance is the presence of two far bigger refinement areas for the sloshing problem, where roughly 1.5 million particles are split and coalesced during the entire simulation.

Figure 5.21 includes snapshots of velocity contours for the tank in several horizontal configurations that span approximately one period of oscillation. As the tank moves towards the left so does the liquid, which then starts to rise due to the blockage of the left wall. A local peak in free-surface elevation is reached on the left side while the tank has already begun its right stroke, starting to drag the water in the opposite direction. This fluid behavior is periodic in time with an observed frequency that matches the fundamental frequency of excitation, i.e. 1/T. However, free-surface peaks are not at all the same in successive oscillations due to fluid inertia, and alternate periods of in-sync and out-of-sync are noted between the movement of the water and the tank.

To better interpret these findings and collect information about the instantaneous water height, a numerical probe is implemented at 5 cm from the left wall, similarly to the experimental probe used in Faltinsen et al. (2000). Specifically, a vertical row of SPH particles is introduced at the given horizontal position and moved rigidly with the tank, according to Equation (5.5). These new particles are not seen by SPH fluid particles. However, flow variables can be extrapolated at the probe position



(a) t = 4 s. (b) t = 4.125 s. (c) t = 4.25 s.



(d) t = 4.375 s. (e) t = 4.625 s. (f) t = 4.875 s.



Figure 5.21: Snapshots of velocity magnitude.



Figure 5.22: Time history of the free-surface elevation at $x(t) = 0.05 + A \cos\left(\frac{2\pi t}{T}\right)$ during the first ~ 9 oscillation cycles.

using kernel interpolations around the probe particles and therefore providing flow information at their location. Results from this procedure are shown in Figure 5.22, where data for the free-surface elevation measured by the probe is presented for several oscillation cycles and superimposed with experimental findings in Faltinsen et al. (2000). Figure 5.22 encloses the first ~ 9 oscillation cycles, corresponding to one period between minima for both wave crests and troughs. While in all three simulated cases the fundamental sloshing frequency is retrieved, an overall better agreement is noted between higher resolution runs and experimental data, especially noticeable by investigating the peaks in free-surface height. Coarse SPH data exhibits some noise in these regions leading to difficulties in the identification of crests and troughs. This is indeed due to poor particle resolution, which causes the kernel interpolation to be less reliable especially at free-surface locations. Smoother output is observed for SPH with splitting and coalescing (red dash-dotted curve) and the fully refined case (green vertically-dashed curve). For these two cases, a good overall agreement with experimental data is observed, although numerical curves are characterized by a negative phase shift, approximately constant in time. Two possible causes for this can be the absence of physical viscous stresses in the SPH momentum equation and a plausible influence of the initial conditions. While both of these are interesting cues that may lead to the improvement of the numerical results, no further investigation is conducted as they fall out of the scope of this section, which is to assess the efficacy of the splitting and coalescing algorithm against uniformly distributed particles.

Chapter 6

Open boundary conditions in DualSPHysics

A novel algorithm for treating open boundaries has been devised by the author to broaden the applicability of SPH to real-world engineering problems characterized by domains of large extent. In classical Eulerian models, this problem is easily addressed by using grid points to discretize the boundary of the computational domain and assigning permeability properties that mimic the flow physics at the boundaries, allowing to reduce the domain extent considerably. However, in Lagrangian methods such as SPH, particles must be inserted in/removed from the domain while preserving physical quantities and numerical conditions such as stability and consistency. The algorithm is presented in this chapter together with an extensive validation study for both two- and three-dimensional problems.

6.1 Implementation

The challenging topic of open boundary conditions in SPH is tackled by following the approach introduced in Lastiwka et al. (2009) for aerodynamics application and subsequently adapted in Federico et al. (2012) for hydrodynamics problems. As seen earlier, the two types of particles generally used for fluid simulations in SPH are fluid and solid boundary particles. The former are governed by the family of SPH conservation equations, whereas different techniques are available to mimic the fluidsolid interactions, whether through boundary particles within the SPH framework (Crespo et al., 2007; Maciá et al., 2011) or by coupling SPH with other methods (Hieber and Koumoutsakos, 2008; Yang et al., 2012). To deal with open boundaries in DualSPHysics, two additional types of particles are introduced, namely "inflow"



Figure 6.1: Sketch of a SPH channel flow with the implemented I/O model.

and "outflow" particles. These are placed in buffer areas located alongside the inlets and outlets of the domain, where the insertion and removal of fluid particles occurs. To limit the complications involved, the work is focused only on straight inlet and outlet shapes.

The working principles of the model are illustrated with an example in Figure 6.1, where a SPH adaptation of a general 2-D channel flow is depicted. The inlet is located on the left of the domain, with a buffer breadth equal to the kernel support (2h in this particular example) plus an arbitrarily small constant, $\varepsilon > 0$. This choice guarantees full kernel support during SPH interpolations among fluid particles within the influence domain of inflow particles. Moreover, the use of the constant ε ensures that inflow particles are strictly inside the inflow buffer in the initial configuration. Conversely, the choice of exactly 2h ($\varepsilon = 0$) can be confusing as some inflow particles would overlap the inlet threshold, thus being fluid particles at the same time. As an inflow particle crosses the inlet threshold, a new fluid particle is created at its location, \mathbf{x}_{new} , while the position of the inflow particle is reinitialized at $\mathbf{x}_{\text{new}} \pm (2h + \varepsilon)$. On the other hand, when a fluid particle goes through the outlet threshold, it is flagged as an outflow particle. Several options are available in this case. With the current state of the code, the properties of an outflow particle are frozen and stored in a separate array which is purged at the next time step. This is chosen to limit the computational effort and therefore only a truly outlet condition is presently available.



Figure 6.2: Contiguous array notation for the insertion/removal process of particles in DualSPHysics.

However, similarly to the treatment of inflow particles, the algorithm structure allows to keep the outflow particles within the buffer breadth for successive time steps and prescribe or extrapolate flow properties from the domain interior to achieve specific outflow conditions.

To further clarify the process of particle insertion and removal, the evolution of particle arrays during certain simulation steps is represented in Figure 6.2. The contiguous array notation adopted with solid, s_k , fluid, f_k , and inflow particles, i_k , corresponds to the exact order used to generate the particle array fed to the interpolation stage, where fluid-fluid, fluid-inflow, fluid-solid, solid-fluid, solid-inflow, inflow-fluid and inflow-solid interactions take place. The difference with respect to the formulation in Federico et al. (2012) is that this model allows the transfer of flow information from the domain interior to the boundary, for example using inflow-fluid and inflow-solid interactions. This is important when simulating subsonic flows (Moretti, 1969; Lastiwka et al., 2009). An inflow-inflow interaction is also possible, but not considered in this work.

As mentioned earlier, the choice of a separate array is preferred for outflow particles, o_k , that do not contribute to interactions with any of the other SPH elements. As simulation time advances, the arrays in Figure 6.2 evolve as follows:

- (a) At time t, the initial arrays configuration is shown. The size of the left array corresponds to the number of all particles enclosed in the domain. Some memory is also allocated for the outflow array on the right, initially empty.
- (b) At time $t + k_1 \Delta t$, with k_1 arbitrary, an inflow particle, i_1 , crosses the inlet threshold, therefore leading to the creation of a new fluid particle, f_6 , at the same position. The particle array is resized as follows:
 - 1. Dynamic memory is used to store a copy of the particle array prior to modification.
 - 2. The new array size is computed and compared to the current memory size reserved for the particle array, with new memory allocated if needed.
 - 3. Inflow particles are shifted at the end of the array. The position of particle i_1 is reinitialized in the buffer, i.e. $\mathbf{x}_{i_1} = \mathbf{x}_{i_1} \pm (2h + \varepsilon)$
 - 4. Information of the new fluid particle(s) is retrieved from the array copy made in step one.
 - 5. Dynamic memory is purged.
- (c) At time $t + k_2 \Delta t$, with k_2 arbitrary, a fluid particle, f_4 , crosses the outlet threshold, therefore leading to the cancellation of a fluid particle and the creation of an outflow particle, o_1 ,. The particle array is resized again, whereas the outflow array has now one element.
- (d) At time $t+k_3\Delta t$, with k_3 arbitrary, both inflow particles, i_1, i_2 , and fluid particle, f_3 , cross the permeable boundaries, leading to steps (b) and (c) simultaneously.

The scheme uses the same logic in three dimensions, with 3-D buffer regions occupying the outer domain area with a breadth of $\pm(2h + \varepsilon)$ in the main flow direction. One of the advantages of this approach is that the size of the buffers is always constant and generally quite small with respect to other arrays. Thus no substantial cost is introduced when computing particle pairwise interactions, one of the most expensive process during simulations with DualSPHysics (Crespo et al., 2015). However, some overhead is introduced because of array resizing other memory operations. Some remarks about the algorithm efficiency are provided for the test cases presented in Section 6.3.2.



Figure 6.3: Schematics of the 2-D jet impingement problem (Lokman Hosain, 2013).

6.2 2-D test cases

6.2.1 Liquid jet impinging on a flat plate

The free-surface flow created by a liquid jet impinging on a solid surface is studied in this section as a first test case for the newly implemented boundary conditions. This is a relevant topic in studies involving fast liquid cooling (Naphon and Wongwises, 2010) or hydraulic jet cleaning (Wang et al., 2013b) and it is a fairly common benchmark case in the SPH community due to the presence of a free-surface. Simulations are presented for a two-dimensional water jet impinging on a rigid plate at a right angle, using the same input data as in Koukouvinis and Anagnostopoulos (2013) for comparison of the results. A sketch of the investigated problem is shown in Figure 6.3, with three main flow areas identified nearby the plate: a stagnation zone, where most of the flow kinetic energy is suddenly converted in pressure, and two symmetric regions of radial flow obtained after flow acceleration away from the stagnation point. The jet diameter, d, and distance from the plate, H, are fixed at 30 mm and 120 mm, respectively, whereas flow conditions at the inlet are assigned as $\rho = 1,000 \text{ Kg m}^{-3}$ and three different jet velocities, $U_{\infty} = [20, 30, 40] \text{ m/s}$. Due to the high Reynolds numbers involved, e.g. $Re \in [10^5 - 10^6]$, the boundary layer from the pipe walls is neglected and a free-slip condition with a uniform velocity distribution



Figure 6.4: Time snapshots of velocity contours for a jet speed $U_{\infty} = 20$ m/s.

is assigned at the inlet. Following this reasoning, no viscosity model other than the artificial viscosity is implemented, leading to inviscid jet impingement as simulated also by Koukouvinis and Anagnostopoulos (2013). The particle spacing is fixed to $\Delta x = 0.01667D$, giving 60 fluid particles along the jet diameter and approximately 32,000 particles on average for each time step.

Contours of velocity magnitude for a jet speed $U_{\infty} = 20$ m/s are plotted in Figure 6.4 at different time instants. Figure 6.4(a) shows the initial condition, with the dynamic boundary particles discretizing the plate on the bottom and the inlet buffer region at the top of the domain. Panels (b)-(e) describe the transient flow, starting from the moment of impact between the fluid and the plate in Figure 6.4(b). Finally, Figure 6.4(f) illustrates the steady flow over the plate after 100 ms of simulated physical time, approximately four times the simulation length in Koukouvinis and Anagnostopoulos (2013). In this last snapshot, it can be seen how the stagnation zone is properly captured by observing the impinging point right beneath the jet at the plate center, where the velocity vanishes. Subsequent acceleration due to the high pressure in this region causes the flow to exit radially, entering the radial flow zones at the left and right of the stagnation area, respectively.



Figure 6.5: Normalized pressure for $U_{\infty} = 20$ m/s.



Figure 6.6: Normalized pressure for $U_{\infty} = 30$ m/s.







Figure 6.8: Comparison of the pressure coefficient for $U_{\infty} = 20$ m/s.

In Figures 6.5 to 6.7, pressure coefficient contours are presented for all three simulated cases, with $C_p = 2P/\rho U_{\infty}^2$. As expected, the fluid pressure scales as the square of the free-stream speed and the contours present striking similarities altogether. Both velocity and pressure fields are in very good agreement with results in Koukouvinis and Anagnostopoulos (2013), and in qualitative agreement with similar simulations in Lokman Hosain (2013), obtained using ANSYS[®] FLUENT[®]. For $U_{\infty} = 20$ m/s, curves of instantaneous C_p along the plate are reported in Figure 6.8 for SPH results obtained herein (dots), SPH-ALE results with preconditioning in Koukouvinis and Anagnostopoulos (2013) (x's) and the analytical results (solid curve) reported in the same paper and calculated using the implicit relations in Taylor (1966). SPH data from this work exhibits a slightly more scattered pressure distribution around the stagnation area, which is most likely due to unphysical repulsive forces generated by the dynamic particles discretizing the plate. Despite this small numerical discrepancy, a reasonable match with the other results is still noted.



Figure 6.9: Numerical domain for the 2-D channel flow past a cylinder.

6.2.2 Channel flow past a cylinder

Fluid flow past a cylinder is a classical problem of CFD for which many numerical and experimental results are available in the literature, see for example (Calhoun, 2002; Coutanceau and Bouard, 1977; Dennis and Chang, 1970; Fornberg, 1980; Vacondio et al., 2013), therefore it is chosen as the second test case for the newly implemented boundary conditions. A sketch of the two-dimensional domain is presented in Figure 6.9, with the cylinder diameter, $D = 10^{-1}$ m, used to nondimensionalize all other lengths. A $50D \times 20D$ computational space is considered, with the cylinder located at the origin of a 2-D Cartesian reference system, O(x, y). Dynamic boundary particles with null velocity are employed for discretizing the cylinder and the remaining outer edges of the domain. In particular, some dynamic particles are also used to discretize two small vertical segments on the top-left and bottom-left of the domain, with the aim of generating a moderate background pressure which is known for increasing the accuracy of simulations involving flow past objects with SPH, see for example Marrone et al. (2013). The fluid enters the domain through the inlet located at a distance 8D from the cylinder, with prescribed density, $\rho = 1,000$ kg m⁻³, and velocity, $U_{\infty} = 1 \text{ m/s}$. For the latter, a rectangular and uniform profile is chosen. This is acceptable since the inlet is placed at 3h from the wall, therefore inflow particles do not fall within the influence domain of the dynamic particles initially. The outlet is located on the left side of the domain and fluid particles that cross this threshold are treated as outflow particles. The particle spacing is fixed to $\Delta x = 0.05D$, giving 20 boundary particles along the cylinder diameter and approximately 400,000 particles in full every time step. Moreover, particle shifting is activated as this correction is deemed particularly important in ensuring uniform particle distributions (Vacondio et al., 2013). The Reynolds number, $\text{Re} = U_{\infty}D \nu^{-1}$, is varied by changing the value of ν while keeping U_{∞} and D constant. Following the approach of Calhoun (2002), simulation results are calculated for three different regimes: steady (Re = 20), transitional (Re = 50) and unsteady flows (Re = 100, 200 and 500). Results are validated with similar works of Calhoun (2002); Vacondio et al. (2013) and the cited literature therein.

Steady solution: Re = 20

Contours of nondimensional velocity magnitude and vorticity for the steady solution are displayed in Figure 6.10. Results are in very good agreement with findings in Calhoun (2002); Fornberg (1980), with a steady smooth wake behind the cylinder and symmetric flow about the x-axis. The velocity field nearby the stagnation point is shown in Figure 6.11, where the arrow size and tip represent the magnitude and direction of the velocity vector at several points across the domain. This depiction will be used momentarily to identify changes in the velocity sign for the detection of recirculation areas. Streamlines are also given in Figure 6.12, showing a well-ordered flow with two vortices filling the wake behind the cylinder and quite symmetric in shape. Using the vectorial notation of Figure 6.11 in the wake area, a recirculation length of 0.92D and separation angle of ~ 41° are measured, in close agreement with values of Calhoun (2002); Coutanceau and Bouard (1977); Dennis and Chang (1970); Fornberg (1980) and SPH simulation with variable resolution in Vacondio et al. (2013).

Transitional regime: Re = 50

For Re = 50, the flow is transitioning to the unsteady regime, but is still in the steady and attached configuration. Contours in Figure 6.13 show a longer wake as opposed to the previous case, with the symmetry starting to break along the *x*-axis. Furthermore, the support of the vorticity field increases notably along the main flow direction. To better study the transition, streamlines are investigated in Figure 6.14. The two vortices in the recirculation area have grown considerably and, while still approximately equal in shape, they are followed by a mildly wavy wake, revealing the



Figure 6.10: Close-up contours of nondimensional velocity (top) and vorticity (bottom) for Re = 20.



Figure 6.11: Vectorial representation of velocity field around the stagnation region.



Figure 6.12: Streamlines for Re = 20.



Figure 6.13: Close-up contours of nondimensional velocity (top) and vorticity (bottom) for Re = 50.

imminent onset of flow instability. The measured length of the recirculation bubble is equal to 2.25*D*, whereas the separation angle is now ~ 53°. Once again, very close agreement is seen with the results in Calhoun (2002); Coutanceau and Bouard (1977); Dennis and Chang (1970); Fornberg (1980). For this particular case, the influence of the channel walls is also assessed by comparing the results with the work of Singha and Sinhamahapatra (2010), where the dependence of the solution from the channel breadth is studied using the Finite Volume Method. Results therein claim negligible wall effects for channel breadths equal or greater than 8*D*, a value far smaller than the one chosen in this work and for which both recirculation length and separation angle are independent of the wall distance. SPH simulation in this thesis corroborate



these findings as strong similarities are seen between the SPH flow field and that in Singha and Sinhamahapatra (2010) with breadth 8D.

Unsteady solution: Re = 100, 200 and 500

The last regime considered for this test case is the fully unsteady flow, for which simulations at Re = 100, 200 and 500 are presented. For all three cases, an oscillatory wake behind the cylinder is observed and expected, with the formation of the classical von Karman street. Figures 6.15 to 6.17 highlight the presence of periodic counterrotating vortices shed behind the cylinder. In Figures 6.18 to 6.20 streamlines relative to all three unsteady cases are also shown, where striking flow similarities can be observed with the Re = 100 case simulated by Vacondio et al. (2013) using variable resolution SPH. Further looking at vorticity contours, it can be seen how the frequency of shed vortices is dependent of the Reynolds number since, as Re increases, a larger number of vortex cores is spotted in the same domain area. To assess the goodness of the captured shedding frequency, f, Table 6.1 reports calculated Strouhal numbers, St = fDU_{∞}^{-1} , for SPH simulations herein and other literature works. A very close agreement with the cited literature can be noted, indicating that the physics of this problem is well-captured by SPH with the present open boundary implementation.

Strouhal number (St)	Re = 100	Re = 200	Re = 500
SPH (present)	0.174	0.205	0.235
SPH (Vacondio et al., 2013)	0.175	-	-
Calhoun (2002)	0.175	0.202	-
Liu et al. (1998)	0.164	0.192	-
Cao and Tamura (2009)	-	0.187	0.210

Table 6.1: Strouhal number for Re = 100, 200 and 500.



Figure 6.15: Close-up contours of nondimensional velocity (top) and vorticity (bottom) for Re = 100.



Figure 6.16: Close-up contours of nondimensional velocity (top) and vorticity (bottom) for Re = 200.



Figure 6.17: Close-up contours of nondimensional velocity (top) and vorticity (bottom) for Re = 500.







Figure 6.19: Streamlines for Re = 200.



Figure 6.20: Streamlines for Re = 500.



Figure 6.21: Pressure coefficient along the plate.

6.3 3-D test cases

6.3.1 Liquid jet impinging on a flat and inclined plate

The problem proposed in Section 6.2.1 is extended to three dimensions by simply extruding the 2-D set-up in Figure 6.3 in the third orthogonal direction. Herein only one velocity, $U_{\infty} = 20$ m/s, is considered and results are provided for the plate in cross-flow and at 45 degrees with respect to the flow direction. For simplicity of implementation, a two-dimensional squared cross-section is chosen for the jet, with same flow conditions as for the 2-D case. The particle spacing is doubled to $\Delta x =$ 0.0333D, giving 900 fluid particles in the jet cross-section and approximately 350,000 particles on average during each time step. The contour plot in Figure 6.21 depicts the pressure coefficient, C_p , along the plate surface. Despite the use of δ -SPH to reduce



Figure 6.22: 3-D contours of normalized velocity.



Figure 6.23: Free-surface profile along the positive XZ-plane for $U_{\infty} = 20$ m/s.

the oscillations affecting the pressure, some noise is still present and are attributed to the use of dynamic boundary particles for the discretization of the plate. It is worth noting, however, that the use of filtering for pressure is critical since the method would yield completely unrealistic results otherwise. The qualitative shape of the pressure field is satisfactory, with the circular area of high pressure presenting strong similarities with contours in Koukouvinis and Anagnostopoulos (2013) for a round jet of same dimensions. The major difference for the square jet impingement is the presence of a four-point star-shaped pressure area around the circular one, with the tips located along the four main directions of outflow. This mild pressure region is not seen in the circular jet because the outflow is purely radial due to the absence of edges. Velocity contours in three-dimensions are presented in Figure 6.22, suggesting a symmetric flow about the two planes of symmetry of the jet, XZ and YZ, respectively. Figure 6.23 shows the thickness of the fluid film along the XZ-centerplane for the round jet of same diameter studied numerically in Koukouvinis and Anagnostopoulos (2013) and experimentally in Kvicinsky (2002). It can be seen that the shape of the jet cross-section influences the water layer thickness downstream, especially in the region of transition from stagnation to radial flow, with a film thickness up to 40%higher for the square jet. The discrepancy decreases as the fluid leaves the highpressure zone. Similarities between square and round jet of same dimensions hold for the plate at 45° as well. Velocity and pressure distributions are displayed at the jet YZ-centerplane in Figures 6.24 and 6.25, with a flow field that looks similar to SPH and SPH-ALE contours in Koukouvinis and Anagnostopoulos (2013). Using SPH alone, the distribution of particles discretizing the free-surface is slightly more sparse in some areas because of kernel truncation. Nevertheless, the flow is properly captured with overall satisfactory results.



Figure 6.24: Velocity contours for the inclined plate case.



Figure 6.25: Pressure contours for the inclined plate case.


Figure 6.26: 3-D visualization of the wake behind the cube for Re = 30 (top) and Re = 300 (bottom).

6.3.2 Channel flow past a cube

A three-dimensional version of the channel flow past a cylinder presented in Section 6.2.2 is addressed here. Similarly to the extension approach chosen for the impinging jet problem, the computational space is extruded in the third direction, with same domain dimensions, simulation parameters and boundary conditions as in the two-dimensional case. The object in cross-flow is shaped as a cube with side L = 0.2 m. Particle spacing is fixed to L/8, with 512 particles discretizing the cube and approximately 1.4×10^6 particles on average at each time step.

The wake formed behind the cube is visualized in Figure 6.26 for Re = 30 (top) and Re = 300 (bottom). In the first case, the flow downstream looks smooth and streamlined, characterized by a recirculation region nearby the cube where the velocity vanishes, followed by an area of increasing velocity as the fluid gains kinetic energy. The higher Reynolds number case presents different features, with several warps in the wake suggesting unsteady flow. Furthermore, vorticity contours are presented in Figures 6.27 to 6.29 for the steady case and Figures 6.30 to 6.32 for the unsteady case. Each figure includes three panels representing planes cutting through the computational space in three different directions. The top panels depict the flow along the three centerplanes of the cube, i.e. XY, XZ and YZ, whereas the mid and bottom panels are shifted away from each centerplane by an orthogonal distance of -3/2L and 3/2L, respectively.



Figure 6.27: Vorticity contours along the (XY), (XY - 1.5L), (XY + 1.5L) centerplanes for Re = 30.



Figure 6.28: Vorticity contours along the (XZ), (XZ - 1.5L), (XZ + 1.5L) centerplanes for Re = 30.



Figure 6.29: Vorticity contours along the (YZ), (YZ - 1.5L), (YZ + 1.5L) centerplanes for Re = 30.



Figure 6.30: Vorticity contours along the (XY), (XY - 1.5L), (XY + 1.5L) centerplanes for Re = 300.



Figure 6.31: Vorticity contours along the (XZ), (XZ - 1.5L), (XZ + 1.5L) centerplanes for Re = 300.



Figure 6.32: Vorticity contours along the (YZ), (YZ - 1.5L), (YZ + 1.5L) centerplanes for Re = 300.

Process	Runtime [h]
Computation of forces	290
Data sorting	3.15
Pre-computation of forces	2.95
I/O generation	0.04

Table 6.2: Runtime specifications for 10 [s] of simulated physical time on a 12-core CPU.

For Re = 30, Figures 6.27 and 6.28 highlight flow symmetry along the streamwise direction for both the XY and XZ planes. In the top panels, symmetric flow areas are separated by the zero-contour level clearly visible in correspondence of the cube x-axis. Two additional contour levels of null vorticity can be spotted between the cylinder wake and the boundary layer growing on both sides of the channel walls. In the case of steady flow, this interaction suggests that, depending on the Reynolds number, viscous dissipation can strongly influence the breadth of the recirculation area behind the cube as the point of contact between the wake and the boundary layer could move further upstream. However, in this case, this effect is deemed negligible since the contact is estimated several characteristic lengths past the cube. High areas of vorticity nearby the cube appear in the vicinity of the stagnation point, whose top view is shown in the bottom panel of Figure 6.29. These big vortical regions surround the side walls as a result of the blockage effect of the cube frontal wall. Mid and bottom panels of Figures 6.27 and 6.28 provide a top view of these areas.

At the higher Reynolds number, Re = 300, symmetry is lost and the oscillatory nature of unsteady flows is noticed. The zero-level contour in the top panels of Figures 6.30 and 6.31 has now warped and exhibits the oscillatory behavior characteristic of vortex shedding. Nevertheless, the visualization of vortex cores is not explicitly noted. This could be due to the low aspect ratio chosen, which causes greater formation lengths for the vortices (Dutta et al., 2008), therefore requiring a longer computational domain in the streamwise direction. In addition, as highlighted in Singha and Sinhamahapatra (2010), the presence of lateral walls can be the cause of delayed vortex shedding.

Table 6.2 summarizes simulation runtimes, together with the CPU time required by the three most demanding processes in DualSPHysics and the newly designed algorithm. Given the relatively small number of particles in the buffer regions, the CPU effort for introducing buffer areas is negligible, although it is worth noting that a very small portion of the other expensive processes is also dedicated to buffer particles.

Chapter 7 Conclusions and future works

Numerical results of free-surface flows and a variety of other hydrodynamics problems have been presented in this dissertation using the Smoothed Particle Hydrodynamics (SPH) method. Simulations have been carried out using a state of the art open-source code named DualSPHysics (Crespo et al., 2015; Gómez-Gesteira et al., 2012a,b), allowing the author to gain useful insights about this computational methodology. Specifically, SPH shortcomings have been identified after extensive simulation work and this has led to the development and testing of numerical techniques to deal with two major challenges in SPH: adaptivity and boundary conditions.

One of the free-surface problems addressed in this thesis is the solution of the wave elevation and bottom pressure generated by a planing hull in finite-depth water. Numerical results are obtained for a variety of depth and length Froude numbers to provide a comprehensive breadth of flow conditions. Analysis of the simulation data shows a clear correlation between surface waves and pressure disturbances at the ocean bottom. At the free-surface, the boat produces a wake mainly composed of divergent waves, with a wake half-angle that decreases with increasing Froude number in agreement with other findings in the literature. The pressure field at the ocean bottom follows a similar trend, with two main disturbance areas: a high-pressure disk underneath amidships and a V-shaped wedge of low pressure that extends for several boat lengths behind the hull. At the critical Froude, secondary areas of pressure disturbance become more visible in an alternate pattern. A functional relation between the low-pressure wedge angle and the depth Froude number is presented, reflecting the 1/Fr dependence of the surface wake angle seen in other works. In terms of intensity of the pressure signals, a quadratic relationship between the hydrodynamic pressure and the Froude number is evident at supercritical regimes, whereas larger pressures are observed when the hull navigates at the critical Froude number. Valid ideas for future works on this subject include studying the effect of the hull shape, the influence of waves and their dispersion relation (shallow-intermediate-deep water) and the use of open boundaries instead of dynamic boundary particles to discretize the lateral edges of the domain.

A second study in this dissertation concerns the prediction of the hydrodynamic load on a thin rigid lamina that oscillates harmonically in an viscous fluid with and without a free surface. The numerical analysis is carried out using two control parameters that describe the oscillation amplitude and frequency. Non-linearities arising from the effect of finite oscillation strokes are identified with the formation of vortical structures that interact with the lamina and the surrounding flow field for one or more oscillation cycles. As a result, forces exerted by the fluid are strongly affected by vortex formation, shedding and advection. The transport of vorticity constitutes the underlying mechanism that drives the hydrodynamic damping, which is modeled by a complex hydrodynamic function that also incorporates an added mass coefficient. For the case of an unbounded fluid, a simple power model is proposed to account for the effect of large amplitudes upon the hydrodynamic actions on the oscillating plate. In the range of considered parameters, the added mass coefficient remains approximately constant whereas the damping coefficient varies simultaneously with both the frequency and amplitude parameters. Simulation data from this work is compared with several other numerical and experimental applications in the literature, suggesting the suitability of the obtained results. In the presence of a freesurface, a boundary integral technique is adopted to estimate the hydrodynamic load for infinitely small amplitudes and a subsequent preliminary study is carried out with SPH to investigate the influence of finite amplitudes. For the former, the attention is focused on Stokes flows due to the negligible effects of the oscillation amplitude and low Reynolds numbers considered. A third control parameter is introduced to account for the effect of the depth of submergence. It is noted that the presence of the free surface strongly affects the flow physics and numerical results indicate that the hydrodynamic load increases with the decrease of the submergence depth. The effect of viscous dissipation due to the presence of a free surface is observed especially at very low Reynolds numbers. Similarly to the unbounded case, simulation results for finite-amplitude oscillations in a liquid with a free-surface also corroborate the dependence of the fluid force on the size of the oscillation stroke. In particular, a strong influence of the oscillation amplitude on the damping coefficient is observed, suggesting the possibility to extrapolate new coefficients for the hydrodynamic function. To the best of the author knowledge, this represents the first SPH solution of the hydrodynamics of a thin lamina undergoing harmonic oscillations in a viscous fluid with and without a free surface in the range of selected parameters. Directions for future work include finding a quantitative measure of the influence of the submergence depth on the total fluid force and casting this result in a novel hydrodynamic function that includes both small and large oscillation amplitudes. To this extent, the use of variable resolution SPH could yield higher accuracy of the computational results, especially for cases of small oscillations in which a large number of particles is needed to capture vortex shedding properly.

Development and testing of variable resolution SPH and open boundary conditions are two additional topics investigated in this thesis. They both represent challenges of the SPH method and are current subjects of extensive research studies to broaden the applicability of SPH to complex engineering problems. A variable resolution SPH scheme is presently under implementation in DualSPHysics and has been introduced and described in this manuscript. Two free-surface flows, namely a dam-break and a sloshing problem, have been studied in detail to demonstrate how variable resolution SPH could potentially yield higher accuracy in targeted areas of the computational domain. In both cases, a gain in quality of the results and computational time has been noticed, despite the algorithm still undergoing improvements. With regard to the topic of boundary conditions, a novel algorithm has been developed by the author to simulate certain inflow and outflow conditions with DualSPHysics. While this is a relatively easy task in grid-based methods, careful attention must be devoted when using mesh-less methods such as SPH, as particles must be inserted in/removed from the domain while preserving physical quantities and numerical conditions such as stability and consistency. The algorithm consists of introducing two new types of particles, namely inflow and outflow particles, in buffer areas located in correspondence of domain inlets and outlets. The buffer breadth is chosen to slightly exceed the size of the kernel support in order to avoid any particle inconsistency at the boundaries. Several flow quantities can be either imposed or extrapolated from the domain interior for inflow particles, whereas only a truly outlet condition is used at the moment for the outflow region. The algorithm has been validated for the following twoand three-dimensional cases:

• A 2-D liquid jet impinging on a flat plate, simulated for three different jet velocities. For all cases, the stagnation area is well captured and no sign of instabilities due to the addition and removal of fluid particles has been observed. Flow velocity and pressure profiles obtained with DualSPHysics are in good agreement with analogous works in the literature.

- A 2-D channel flow past a cylinder, simulated for five values of the Reynolds number, i.e. Re = [20, 50, 100, 200, 500]. Numerical outcomes are validated against simulations from a variety of other works in the cited literature, including a variable resolution SPH method. A close match is observed for each Reynolds number in terms of velocity and vorticity contours, length of the recirculation area, angle of separation and, for unsteady cases, the Strouhal number.
- The case of the impinging jet is extended to three dimensions with the choice of a square cross-sectioned jet impacting the plate at 45 and 90 degrees, respectively. Only one value of the velocity is investigated for this particular case. The pressure coefficient on the plate and the free-surface profile along plane cuts are presented, showing striking similarities with a circular jet of same speed and dimensions simulated with an hybrid SPH-ALE method and ANSYS[®] FLUENT[®].
- An extension of the channel flow past a cylinder is also investigated by extruding the set-up used for the 2-D case in the third dimension. A cubical shape is chosen for the object in cross flow and two Reynolds numbers are considered, namely Re = 30 and Re = 300. Results are organized in terms of vorticity contours in the three centerplanes cutting along the cube. A 3-D representation of the velocity field in the wake behind the cube is also provided. The flow at the low Reynolds number is smooth and streamlined as expected, with a longitudinal symmetry noticed as the solution reaches steady-state conditions. Conversely, an oscillatory wake is observed at Re = 300, highlighting the unsteady nature of the flow at this regime seen also in the 2-D case.

The efficiency of the implemented I/O algorithm is later discussed and it is inferred that the process controlling the insertion and removal of particles has negligible computational costs.

A cue for the improvement of the open boundary formulation presented herein is represented by exploiting particles inside the outflow buffer to achieve desired outlet conditions, such as the case of an hydraulic jump in open channel flows or other problems requiring specific conditions at the outlet. Other directions for future work include the use of multiple inlets, which can be readily implemented with the present algorithm structure, the design of other inlet/outlet geometries (e.g. curved boundaries), the treatment of back flows in the presence of I/O conditions and an investigation on the particle consistency of the presented algorithm.

List of publications

Tafuni, A., Sahin, I., Domínguez, J.M., Crespo, A.J.C., Vacondio, R.. A SPH open boundary condition formulation and implementation in DualSPHysics, journal article in preparation.

Tafuni, A., Sahin, I., 2016. Effect of depth on the bottom pressure signatures of a high-speed vessel, abstract submitted to: 31st Symposium on Naval Hydrodynamics.

Tafuni, A., Shrestha, B., Ma, B., Sahin, I., Aureli, M., 2016. Vortex shedding and symmetry breaking in plates oscillating in viscous fluids: a computational and experimental study, abstract submitted to: 31st Symposium on Naval Hydrodynamics.

Tafuni, A., Sahin, I., Hyman, M., 2016. Numerical investigation of wave elevation and bottom pressure generated by a planing boat in finite-depth water, under review in: Applied Ocean Research.

Tafuni, A., Sahin, I., 2015. Non-linear hydrodynamics of thin laminae undergoing large harmonic oscillations in a viscous fluid. Journal of Fluids and Structures 52, 101–117. doi:10.1016/j.jfluidstructs.2014.10.004.

Di Ilio, G., Sahin, I., Tafuni, A., 2014. Unsteady Stokes flow for a vibrating cantilever under a free surface, in: Proceedings of the ASME 2014 International Mechanical Engineering Congress and Exposition 2014, Montreal, Quebec, Canada. doi:10.1115/IMECE2014-36929.

Tafuni, A., Sahin, I., 2014. Seafloor pressure signatures of a high-speed boat in shallow water with SPH, in: Proceedings of the ASME 33rd International Conference on Ocean, Offshore and Arctic Engineering, San Francisco, CA, USA. doi:10.1115/OMAE2014-24080. Tafuni, A., Sahin, I., 2013. Hydrodynamic loads on vibrating cantilevers under a free surface in viscous fluids with SPH, in: Proceedings of the ASME 2013 International Mechanical Engineering Congress and Exposition 2013, San Diego, CA, USA. doi:10.1115/IMECE2013-63792.

Bibliography

- Akimoto, H., 2013. Numerical simulation of the flow around a planing body by MPS method. Ocean Engineering 64, 72–79. doi:10.1016/j.oceaneng.2013.02.015.
- Amabili, M., 1996a. Effect of finite fluid depth on the hydroelastic vibrations of circular and annular plates. Journal of Sound and Vibration 193, 909–925. doi:10. 1006/jsvi.1996.0322.
- Amabili, M., 1996b. Free vibration of partially filled, horizontal cylindrical shells. Journal of Sound and Vibration 191, 757–780. doi:10.1006/jsvi.1996.0154.
- Amabili, M., 2000. Eigenvalue problems for vibrating structures coupled with quiescent fluids with free surface. Journal of Sound and Vibration 231, 79–97. doi:10.1006/jsvi.1999.2678.
- Amabili, M., 2001. Vibrations of circular plates resting on a sloshing liquid: solution of the fully coupled problem. Journal of Sound and Vibration 245, 261–283. doi:10. 1006/jsvi.2000.3560.
- Amabili, M., Garziera, R., Negri, A., 2002. Experimental study on large-amplitude vibrations of water-filled circular cylindrical shells. Journal of Fluids and Structures 16, 213–227. doi:10.1006/jfls.2001.0412.
- Amabili, M., Kwak, M.K., 1996. Free vibrations of circular plates coupled with liquids: reversing the Lamb problem. Journal of Fluids and Structures 10, 743– 761. doi:10.1006/jfls.1996.0051.
- Aureli, M., Basaran, M.E., Porfiri, M., 2012. Nonlinear finite amplitude vibrations of sharp-edged beams in viscous fluids. Journal of Sound and Vibration 331, 1624– 1654. doi:10.1016/j.jsv.2011.12.007.

- Aureli, M., Porfiri, M., 2010. Low frequency and large amplitude oscillations of cantilevers in viscous fluids. Applied Physics Letters 96, 164102. doi:10.1063/1. 3405720.
- Barnell, A., Noblesse, F., 1986. Far-field features of the Kelvin wake. Technical Report. David W. Taylor Naval Ship Research and Development Center. doi:10. 5962/bhl.title.48274.
- Barnes, J.C., 1998. New techniques to predict ship vulnerability to pressure mines along shipping routes. Technical Report. Defence Science and Technology Organization Canberra (Australia).
- Basak, S., Raman, A., Garimella, S.V., 2006. Hydrodynamic loading of microcantilevers vibrating in viscous fluids. Journal of Applied Physics 99, 114906. doi:10.1063/1.2202232.
- Benassai, G., Piscopo, V., Scamardella, A., 2013. Field study on waves produced by HSC for coastal management. Ocean & Coastal Management 82, 138–145. doi:10.1016/j.ocecoaman.2013.06.003.
- Bentson, J., Vradis, G., 1990. Two-stage pressure correction technique for the incompressible Navier-Stokes equations. AIAA Journal 28, 1155–1156. doi:10.2514/3. 25185.
- Bidkar, R., Kimber, M., Raman, A., Bajaj, A.K., Garimella, S.V., 2009. Nonlinear aerodynamic damping of sharp-edged flexible beams oscillating at low Keulegan-Carpenter numbers. Journal of Fluid Mechanics 634, 269–289. doi:10.1017/ S0022112009007228.
- Bielański, J., 2012. Signature of hydrodynamic pressure field. Hydroacoustics 15, 13–20.
- Bielański, J., 2014. The hydrodynamic pressure field of the ship Zodiak, measurements and calculations. Hydroacoustics 17, 17–28.
- Calhoun, D., 2002. A cartesian grid method for solving the two-dimensional streamfunction-vorticity equations in irregular regions. Journal of Computational Physics 176, 231–275. doi:10.1006/jcph.2001.6970.
- Cao, S., Tamura, Y., 2009. Reynolds number dependence of the velocity shear effects on flow around a circular cylinder, in: Proceedings of the 5th European & African Conference on Wind Engineering, EACWE, Florence, Italy. pp. 1–12.

- Carrica, P.M., Huang, J., Noack, R., Smith, B., Stern, F., 2010. Large-scale DES computations of the forward speed diffraction and pitch and heave problems for a surface combatant. Computer & Fluids 39, 1095–1111. doi:10.1016/j.compfluid. 2010.02.002.
- Chen, Y., Kang, Q., Cai, Q., Zhang, D., 2011. Lattice Boltzmann method on quadtree grids. Physical Review E 026707. doi:10.1103/PhysRevE.83.026707.
- Chorin, A.J., 1967. A numerical method for solving incompressible viscous flow problems. Journal of Computational Physics 135, 118–125. doi:10.1006/jcph. 1997.5716.
- Chorin, A.J., 1968. Numerical solution of the Navier-Stokes equations. Mathematics of Computation 22, 745–762. doi:10.1090/S0025-5718-1968-0242392-2.
- Colagrossi, A., 2003. A meshless Lagrangian method for free–surface and interface flows with fragmentation. Ph.D. thesis. University of Rome La Sapienza.
- Colagrossi, A., Antuono, M., Le Touzé, D., 2009. Theoretical considerations on the free-surface role in the smoothed-particle-hydrodynamics model. Physical Review E 79, 056701. doi:10.1103/PhysRevE.79.056701.
- Coutanceau, M., Bouard, R., 1977. Experimental determination of the main features of the viscous flow in the wake of a circular cylinder in uniform translation. Part 2. Unsteady flow. Journal of Fluid Mechanics 79, 257–272. doi:10.1017/ S0022112077000147.
- Crespo, A.J.C., 2008. Application of the Smoothed Particle Hydrodynamics model SPHysics to free-surface hydrodynamics. Ph.D. thesis. University of Vigo.
- Crespo, A.J.C., Domínguez, J.M., Rogers, B.D., Gómez-Gesteira, M., Longshaw, S., Canelas, R., Vacondio, R., Barreiro, A., García-Feal, O., 2015. DualSPHysics: Open-source parallel CFD solver based on Smoothed Particle Hydrodynamics (SPH). Computer Physics Communications 187, 204–216. doi:10.1016/j.cpc. 2014.10.004.
- Crespo, A.J.C., Gómez-Gesteira, M., Dalrymple, R.A., 2007. Boundary conditions generated by dynamic particles in SPH methods. Computers, Materials, & Continua 5, 173–184. doi:10.3970/cmc.2007.005.173.

- Cummins, S.J., Rudman, M., 1999. An SPH projection method. Journal of Computational Physics 152, 584–607. doi:10.1006/jcph.1999.6246.
- Dalrymple, R.A., Knio, O., 2001. SPH modelling of water waves. Coastal Dynamics 1, 779–787. doi:10.1061/40566(260)80.
- Dalrymple, R.A., Rogers, B.D., 2006. Numerical modeling of water waves with the SPH method. Coastal Engineering '53, 141–147. doi:10.1016/j.coastaleng. 2005.10.004.
- Darmon, A., Benzaquen, M., Raphaël, E., 2014. Kelvin wake pattern at large Froude numbers. Journal of Fluid Mechanics 738, (R3). doi:10.1017/jfm.2013.607.
- Dehnen, W., Aly, H., 2012. Improving convergence in smoothed particle hydrodynamics simulations without pairing instability. Monthly Notices of the Royal Astronomical Society 425, 1068–1082. doi:10.1111/j.1365-2966.2012.21439.x.
- Deng, H., Zhang, Z., Liu, J., Gu, J., 2014. Nonlinear effects on hydrodynamic pressure field caused by ship moving at supercritical speed in shallow water. Ocean Engineering 82, 144–149. doi:10.1016/j.oceaneng.2014.03.003.
- Dennis, S.C.R., Chang, G.Z., 1970. Numerical solutions for steady flow past a circular cylinder at Reynolds number up to 100. Journal of Fluid Mechanics 42, 471–489. doi:10.1017/S0022112070001428.
- Dilts, G.A., 1999. Moving-least-squares-particle hydrodynamics—i. Consistency and stability. International Journal for Numerical Methods in Engineering 44, 1115– 1155. doi:10.1002/(SICI)1097-0207(19990320)44:8<1115::AID-NME547>3.0. C0;2-L.
- Du, L., Jing, X., Sun, X., 2014. Modes of vortex formation and transition to threedimensionality in the wake of a freely vibrating cylinder. Journal of Fluids and Structures 49, 554–573. doi:10.1016/j.jfluidstructs.2014.05.012.
- Dutta, S., Panigrahi, P., Muralidhar, K., 2008. Experimental investigation of flow past a square cylinder at an angle of incidence. Journal of Engineering Mechanics 134, 788–803. doi:10.1061/(ASCE)0733-9399(2008)134:9(788).
- Facci, A.L., Porfiri, M., 2013. Analysis of three-dimensional effects in oscillating cantilevers immersed in viscous fluids. Journal of Fluids and Structures 38, 205– 222. doi:10.1016/j.jfluidstructs.2012.11.006.

- Falcucci, G., Aureli, M., Ubertini, S., Porfiri, M., 2011. Transverse harmonic oscillations of laminae in viscous fluids: a lattice Boltzmann study. Phylosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences 369, 2456–2466. doi:10.1098/rsta.2011.0062.
- Faltinsen, O.M., 2010. Hydrodynamics of High-Speed Marine Vehicles. Cambridge University Press.
- Faltinsen, O.M., Rognebakke, O.F., Lukovsky, I.A., Timokha, A.N., 2000. Multidimensional modal analysis of nonlinear sloshing in a rectangular tank with finite water depth. Journal of Fluid Mechanics 407, 201–234. doi:10.1017/ S0022112099007569.
- Fang, M.C., Yang, R.Y., Shugan, I.V., 2011. Kelvin ship wake in the wind waves field and on the finite sea depth. Journal of Mechanics 27, 71–77. doi:10.1017/jmech. 2011.9.
- Federico, I., Marrone, S., Colagrossi, A., Aristodemo, F., Antuono, M., 2012. Simulating 2D open-channel flows through an SPH model. European Journal of Mechanics B/Fluids 34, 35–46. doi:10.1016/j.euromechflu.2012.02.002.
- Feldman, J., Bonet, J., 2007. Dynamic refinement and boundary contact forces in SPH with applications in fluid flow problems. International Journal for Numerical Methods in Engineering 72, 295–324. doi:10.1002/nme.2010.
- Ferrari, A., Dumbser, M., Toro, E.F., Armanini, A., 2009. A new 3D parallel SPH scheme for free surface flows. Computers and Fluids 38, 1203–1217. doi:10.1016/ j.compfluid.2008.11.012.
- Fornberg, B., 1980. A numerical study of steady viscous flow past a circular cylinder. Journal of Fluid Mechanics 98, 819–855. doi:10.1017/S0022112080000419.
- Gingold, R.A., Monaghan, J.J., 1977. Smoothed Particle Hydrodynamics: theory and application to non-spherical stars. Monthly Notices of the Royal Astronomical Society 181, 375–389. doi:10.1093/mnras/181.3.375.
- Graham, J.M.R., 1980. The forces on sharp-edged cylinders in oscillatory flow at low Keulegan-Carpenter numbers. Journal of Fluid Mechanics 97, 331–346. doi:10. 1017/S0022112080002595.

- Grimaldi, E., Porfiri, M., Soria, L., 2012. Finite amplitude vibrations of a sharp-edged beam immersed in a viscous fluid near a solid surface. Journal of Applied Physics 112, 104907. doi:10.1063/1.4765029.
- Gómez-Gesteira, M., Crespo, A.J.C., Rogers, B.D., Dalrymple, R.A., Dominguez, J.M., Barreiro, A., 2012a. SPHysics – development of a free-surface fluid solver – Part 2: Efficiency and test cases. Computers & Geosciences 48, 300–307. doi:10. 1016/j.cageo.2012.02.028.
- Gómez-Gesteira, M., Dalrymple, R.A., 2004. Using a three-dimensional Smoothed Particle Hydrodynamics method for wave impact on a tall structure. Journal of Waterway, Port, Coastal, and Ocean Engineering 130, 63–69. doi:10.1061/(ASCE) 0733-950X(2004)130:2(63).
- Gómez-Gesteira, M., Rogers, B.D., Crespo, A.J.C., Dalrymple, R.A., Narayanaswamy, M., Dominguez, J.M., 2012b. SPHysics – development of a free-surface fluid solver – Part 1: Theory and formulations. Computers & Geosciences 48, 289–299. doi:10.1016/j.cageo.2012.02.029.
- He, J., Zhang, C., Zhu, Y. Wu, H., Yang, C., Noblesse, F., Gu, X., Li, W., 2015. Comparison of three simple models of Kelvin's ship wake. European Journal of Mechanics - B/Fluids 49 A, 12–19. doi:10.1016/j.euromechflu.2014.07.006.
- Hieber, S.E., Koumoutsakos, P., 2008. An immersed boundary method for smoothed particle hydrodynamics of self-propelled swimmers. Journal of Computational Physics 227, 8636–8654. doi:10.1016/j.jcp.2008.06.017.
- Intartaglia, C., Soria, L., Porfiri, M., 2014. Hydrodynamic coupling of two sharpedged beams vibrating in a viscous fluid. Proceedings of the Royal Society of London: Mathematical, Physical & Engineering Sciences Series A 470, 20130397. doi:10.1098/rspa.2013.0397.
- Issa, R., 2004. Numerical assessment of the Smoothed Particle Hydrodynamics gridless method for incompressible flows and its extension to turbulent flows. Ph.D. thesis. University of Manchester Institute of Science and Technology.
- Jalalisendi, M., Panciroli, R., Cha, Y., Porfiri, M., 2014. A particle image velocimetry study of the flow physics generated by a thin lamina oscillating in a viscous fluid. Journal of Applied Physics 115, 054901. doi:10.1063/1.4863721.

- Keulegan, G.H., Carpenter, L.H., 1958. Forces on cylinders and plates in an oscillating fluid. Journal of Research of the National Bureau of Standards 60, 423–440. doi:10. 6028/jres.060.043.
- Kimber, M., Lonergan, R., Garimella, S.V., 2009. Experimental study of aerodynamic damping in arrays of vibrating cantilevers. Journal of Fluids and Structures 25, 1334–1347. doi:10.1016/j.jfluidstructs.2009.07.003.
- Koukouvinis, P.K., Anagnostopoulos, J.S., 2013. An improved MUSCL treatment for the SPH-ALE method: comparison with the standard SPH method for the jet impingement case. International Journal for Numerical Methods in Fluids 71, 1152–1177. doi:10.1002/fld.3706.
- Kvicinsky, S., 2002. Méthode d'analyse des écoulements 3D à surface libre: application aux turbines Pelton. Ph.D. thesis. Ecole Polytechnique Federale de Lausanne, Lausanne, Switzerland. doi:10.5075/epfl-thesis-2526.
- Lamb, H., 1920. On the vibrations of an elastic plate in contact with water. Proceedings of the Royal Society of London: Mathematical, Physical & Engineering Sciences Series A 98, 205–216. doi:10.1098/rspa.1920.0064.
- Lastiwka, M., Basa, M., Quinlan, N.J., 2009. Permeable and non-reflecting boundary conditions in SPH. International Journal for Numerical Methods in Fluids 61, 709–724. doi:10.1002/fld.1971.
- Lazauskas, L., 2007. The hydrodynamic resistance, wave wakes and bottom pressure signatures of a 5900t displacement air warfare destroyer. Technical Report. Department of Applied Mathematics, The University of Adelaide, South Australia (Australia).
- Lee, E.S., Moulinec, C., Xu, R., Violeau, D., Laurence, D., Stansby, P., 2008. Comparisons of weakly compressible and truly incompressible algorithms for the SPH mesh free particle method. Journal of Computational Physics 227, 8417–8436. doi:10.1016/j.jcp.2008.06.005.
- van Leer, B., Lee, W.T., Roe, P., 1991. Characteristic time-stepping or local preconditioning of the Euler equations, in: Proceedings of the 10th Computational Fluid Dynamics Conference, Honolulu, HI, USA. pp. 260–282. doi:10.2514/6.1991-1552.

- Liu, C., Zheng, X., Sung, C.H., 1998. Preconditioned multigrid methods for unsteady incompressible flows. Journal of Computational Physics 139, 35–57. doi:10.1006/ jcph.1997.5859.
- Liu, G.R., Liu, M.B., 2003. Smoothed Particle Hydrodynamics: a meshfree particle method. World Scientific Publishing.
- Liu, M.B., Liu, G.R., 2010. Smoothed Particle Hydrodynamics (SPH): an overview and recent developments. Archives of Computational Methods in Engineering 17, 25–76. doi:10.1007/s11831-010-9040-7.
- Liu, M.B., Liu, G.R., Lam, K.Y., 2003. Constructing smoothing functions in smoothed particle hydrodynamics with applications. Journal of Computational and Applied Mathematics 155, 263–284. doi:10.1016/S0377-0427(02)00869-5.
- Liu, W.K., Jun, S., Zhang, Y.F., 1995. Reproducing kernel particle methods. International Journal for Numerical Methods in Fluids 20, 1081–1106. doi:10.1002/ fld.1650200824.
- Lo, E.Y.M., Shao, S., 2002. Simulation of near-shore solitary wave mechanics by an incompressible SPH method. Applied Ocean Research 24, 275–286. doi:10.1016/ S0141-1187(03)00002-6.
- Lokman Hosain, M.D., 2013. CFD simulation of jet cooling and implementation of flow solvers in GPU. Master's thesis. Royal Institute of Technology, Stockholm, Sweden.
- Lucy, L.B., 1977. A numerical approach to the testing of the fission hypothesis. Astronomical Journal 82, 1013–1024. doi:10.1086/112164.
- Maciá, F., Antuono, M., M., G.L., Colagrossi, A., 2011. Theoretical analysis of the no-slip boundary condition enforcement in SPH methods. Progress of Theoretical Physics 125, 1091–1121. doi:10.1143/PTP.125.1091.
- Marrone, S., Colagrossi, A., Antuono, M., Colicchio, G., Graziani, G., 2013. An accurate SPH modeling of viscous flows around bodies at low and moderate reynolds numbers. Journal of Computational Physics 245, 456–475. doi:10.1016/j.jcp. 2013.03.011.
- Molteni, D., Colagrossi, A., 2009. A simple procedure to improve the pressure evaluation in hydrodynamic context using the SPH. Computer Physics Communications 180, 861–872. doi:10.1016/j.cpc.2008.12.004.

- Monaghan, J.J., 1985. Particle methods for hydrodynamics. Computer Physics Reports 3, 71–124. doi:10.1016/0167-7977(85)90010-3.
- Monaghan, J.J., 1992. Smoothed particle hydrodynamics. Annual Review of Astronomy and Astrophysics 30, 543–574. doi:10.1146/annurev.aa.30.090192.002551.
- Monaghan, J.J., 1994. Simulating free surface flows with SPH. Journal of Computational Physics 110, 399–406. doi:10.1006/jcph.1994.1034.
- Monaghan, J.J., 1996. Gravity currents and solitary waves. Physica D: Nonlinear Phenomena 98, 523–533. doi:10.1016/0167-2789(96)00110-8.
- Monaghan, J.J., 2000. SPH without a tensile instability. Journal of Computational Physics 159, 290–311. doi:10.1006/jcph.2000.6439.
- Monaghan, J.J., Gingold, R.A., 1983. Shock simulation by the particle method SPH. Journal of Computational Physics 52, 374–389. doi:10.1016/0021-9991(83) 90036-0.
- Monaghan, J.J., Kos, A., 1999. Solitary waves on a cretan beach. Journal of Waterway, Port, Coastal, and Ocean Engineering 125, 145–155. doi:10.1061/(ASCE) 0733-950X(1999)125:3(145).
- Moretti, G., 1969. Importance of boundary conditions in the numerical treatment of hyperbolic equations. Physics of Fluids 12, 13–20. doi:10.1063/1.1692426.
- Morris, J.P., Monaghan, J.J., 1997. A switch to reduce SPH viscosity. Journal of Computational Physics 136, 41–50. doi:10.1006/jcph.1997.5690.
- Naphon, P., Wongwises, S., 2010. Investigation on the jet liquid impingement heat transfer for the central processing unit of personal computers. International Communications in Heat and Mass Transfer 37, 822-826. doi:10.1016/ j.icheatmasstransfer.2010.05.004.
- Nguyen, T., Koide, M., Yamada, S., Takahashi, T., Shirakashi, M., 2012. Influence of mass and damping ratios on VIVs of a cylinder with a downstream counterpart in cruciform arrangement. Journal of Fluids and Structures 28, 40–55. doi:10.1016/ j.jfluidstructs.2011.10.006.
- Nguyen, T.C., Hyman, M., 2010. A hybrid method for computing the bottom pressure field of a planing boat, in: Proceedings of MTS/IEEE OCEANS 2010, Seattle, WA, USA. pp. 1–6. doi:10.1109/OCEANS.2010.5664070.

- Noack, R.W., 2006. SUGGAR: A general capability for moving body overset grid assembly, in: 17th AIAA Computational Fluid Dynamics Conference, Toronto, Canada. doi:10.2514/6.2005-5117.
- Noblesse, F., 1986. Analytical approximation for steady ship waves at low Froude numbers. Technical Report. David W. Taylor Naval Ship Research and Development Center. doi:10.5962/bhl.title.48287.
- Noblesse, F., He, J., Zhu, Y., Hong, L., Zhang, C., Zhu, R., Yang, C., 2014a. Why can ship wakes appear narrower than Kelvin's angle? European Journal of Mechanics B/Fluids 46, 164–171. doi:10.1016/j.euromechflu.2014.03.012.
- Noblesse, F., Hendrix, D., 1991. Near-field nonlinearities and short far-field ship waves, in: Proceedings of the 18th Symposium on Naval Hydrodynamics, Washington, DC, USA, pp. 465–476.
- Noblesse, F., Zhu, R., Hong, L., Zhang, C., He, J., Zhu, Y., 2014b. Interference effects on farfield ship waves at high Froude numbers, in: 29th International Workshop on Water Waves and Floating Bodies, Osaka, Japan.
- Patankar, S.V., Spalding, D.B., 1972. A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. International Journal of Heat and Mass Transfer 15, 1787–1806. doi:10.1016/0017-9310(72)90054-3.
- Phan, C.N., Aureli, M., Porfiri, M., 2013. Finite amplitude vibrations of cantilevers of rectangular cross sections in viscous fluids. Journal of Fluids and Structures 40, 52–69. doi:10.1016/j.jfluidstructs.2013.03.013.
- Price, D.J., 2012. Smoothed particle hydrodynamics and magnetohydrodynamics. Journal of Computational Physics 231, 759–794. doi:10.1016/j.jcp.2010.12.011.
- Rabaud, M., Moisy, F., 2013. Ship wakes: Kelvin or Mach angle? Physical Review Letters 110, 214503. doi:10.1103/PhysRevLett.110.214503.
- Rabaud, M., Moisy, F., 2014. Narrow ship wakes and wave drag for planing hulls. Ocean Engineering 90, 34–38. doi:10.1016/j.oceaneng.2014.06.039.
- Sader, J.E., 1998. Frequency response of cantilever beams immersed in viscous fluids with applications to the atomic force microscope. Journal of Applied Physics 84, 64–76. doi:10.1063/1.368002.

- Sahin, I., Hyman, M., 1993. Numerical calculation for the flow of submerged bodies under a free surface. Ocean Engineering 20, 339–347. doi:10.1016/0029-8018(93) 90028-G.
- Sahin, I., Hyman, M.C., 2001. Simulation of three-dimensional finite-depth wave phenomenon for moving pressure distributions. Ocean Engineering 28, 1621–1630. doi:10.1016/S0029-8018(01)00025-7.
- Sahin, I., Hyman, M.C., Nguyen, T.C., 1994. Three-dimensional flow around a submerged body in finite-depth water. Applied Mathematical Modelling 18, 611–619. doi:10.1016/0307-904X(94)90319-0.
- Sarpkaya, T., 1986. Force on a circular cylinder in viscous oscillatory flow at low Keulegan-Carpenter numbers. Journal of Fluid Mechanics 165, 61–71. doi:10. 1017/S0022112086002999.
- Savitsky, D., 1964. Hydrodynamic design of planing hulls. Marine Technology 1, 71–95.
- Scullen, D.C., Tuck, E.O., 2011. Free-surface elevation due to moving pressure distributions in three dimensions. Journal of Engineering Mathematics 70, 29–42. doi:10.1007/s10665-010-9428-2.
- Shao, J.R., Li, H.Q., Liu, G.R., Liu, M.B., 2012. An improved SPH method for modeling liquid sloshing dynamics. Computers and Structures 100–101, 18–26. doi:10.1016/j.compstruc.2012.02.005.
- Shao, S., Lo, E.Y.M., 2003. Incompressible SPH method for simulating Newtonian and non-Newtonian flows with a free surface. Advances in Water Resources 26, 787–800. doi:10.1016/S0309-1708(03)00030-7.
- Shepard, D., 1968. A two-dimensional interpolation function for irregularly-spaced data, in: Proceedings of the 1968 23rd ACM national conference, New York, NY, pp. 517–524. doi:10.1145/800186.810616.
- Singh, S., 1979. Forces on bodies in oscillatory flow. Ph.D. thesis. University of London.
- Singha, S., Sinhamahapatra, K.P., 2010. Flow past a circular cylinder between parallel walls at low Reynolds numbers. Ocean Engineering 37, 757–769. doi:10.1016/j. oceaneng.2010.02.012.

- Souto Iglesias, A., Pérez Rojas, L., Zamora Rodríguez, R., 2004. Simulation of anti-roll tanks and sloshing type problems with smoothed particle hydrodynamics. Ocean Engineering 31, 1169–1192. doi:10.1016/j.oceaneng.2003.09.002.
- Springel, V., 2010. Smoothed Particle Hydrodynamics in astrophysics. Annual Review of Astronomy and Astrophysics 48, 391–430. doi:10.1146/ annurev-astro-081309-130914.
- Springel, V., Dullemond, C.P., 2011. Lectures on Numerical Fluid Dynamics: Smoothed Particle Hydrodynamics. URL: http://www.ita.uni-heidelberg.de/ ~dullemond/lectures/num_fluid_2011/Chapter_12.pdf.
- Swegle, J.W., Hicks, D.L., Attaway, S.W., 1995. Smoothed Particle Hydrodynamics stability analysis. Journal of Computational Physics 116, 123–134. doi:10.1006/ jcph.1995.1010.
- Tafuni, A., Sahin, I., 2013. Hydrodynamic loads on vibrating cantilevers under a free surface in viscous fluids with SPH, in: Proceedings of the ASME 2013 International Mechanical Engineering Congress and Exposition (IMECE 2013), San Diego, CA, USA. p. V07BT08A013. doi:10.1115/IMECE2013-63792.
- Tafuni, A., Sahin, I., 2014. Seafloor pressure signatures of a high-speed boat in shallow water with SPH, in: ASME 2014 33rd International Conference on Ocean, Offshore and Arctic Engineering, Volume 8B: Ocean Engineering, San Francisco, CA, USA. doi:10.1115/OMAE2014-24080.
- Tafuni, A., Sahin, I., Hyman, M., 2016. Numerical investigation of wave elevation and bottom pressure generated by a planing boat in finite-depth water,. under review in: Applied Ocean Research.
- Taylor, G., 1966. Oblique impact of a jet on a plane surface. Philosophical Transactions for the Royal Society of London, Series A , Mathematical and Physical Sciences 260, 96–100. doi:10.1098/rsta.1966.0034.
- Thomson, W., 1887a. On ship waves. Transactions, Institution of Mechanical Engineers, London, 409–433.
- Thomson, W., 1887b. On the waves produced by a single impulse in water of any depth, or in a dispersive medium. Proceedings of the Royal Society of London 42, 80–83.

- Triantafyllou, G.S., Triantafyllou, M.S., Grosenbaugh, M.A., 1993. Optimal thrust development in oscillating foils with application to fish propulsion. Journal of Fluids and Structures 7, 205–224. doi:10.1006/jfls.1993.1012.
- Triantafyllou, M.S., Triantafyllou, G.S., Yue, D.K.P., 2000. Hydrodynamics of fishlike swimming. Annual Review of Fluid Mechanics 32, 33–53. doi:10.1146/annurev. fluid.32.1.33.
- Tuck, E.O., 1969. Calculation of unsteady flows due to small motions of cylinders in a viscous fluid. Journal of Engineering Mathematics 3, 29–44. doi:10.1007/ BF01540828.
- Tuck, E.O., Lazauskas, L., 1998. Optimum hull spacing of a family of multihulls. Schiffstechnik 45, 180–195.
- Tuck, E.O., Scullen, D.C., Lazauskas, L., 2002. Wave patterns and minimum wave resistance for High-Speed Vessels, in: 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan.
- Tunaley, J.K.E., 2014a. Ship wakes in shallow waters. Technical Report. London Research and Development Corporation, Ottawa, Canada.
- Tunaley, J.K.E., 2014b. Wakes from go-fast and small planing boats. Technical Report. London Research and Development Corporation, Ottawa, Canada.
- Turkel, E., 1999. Preconditioning techniques in Computational Fluid Dynamics. Annual Review of Fluid Mechanics 31, 385–416. doi:10.1146/annurev.fluid.31.1. 385.
- Vacondio, R., Crespo, A.J.C., Gómez-Gesteira, M., Rogers, B.D., 2015. DualSPHysics with adaptivity: towards the simulation of real engineering problems with variable resolution, in: Proceedings of the 10th International SPHERIC Workshop, Parma, Italy. pp. 171–177.
- Vacondio, R., Rogers, B.D., Stansby, P.K., 2012. Accurate particle splitting for smoothed particle hydrodynamics in shallow water with shock capturing. International Journal for Numerical Methods in Fluids 69, 1377–1410. doi:10.1002/fld. 2646.
- Vacondio, R., Rogers, B.D., Stansby, P.K., Mignosa, P., Feldman, J., 2013. Variable resolution for SPH: a dynamic particle coalescing and splitting scheme. Computer

Methods in Applied Mechanics and Engineering 256, 132–148. doi:10.1016/j.cma. 2012.12.014.

Violeau, D., 2012. Fluid mechanics and the SPH method. Oxford University Press.

- Wang, J.G., Karim, M.R., Lin, P.Z., 2007. Analysis of seabed instability using element free Galerkin method. Ocean Engineering 34, 247–260. doi:10.1016/j.oceaneng. 2006.01.004.
- Wang, L., Wang, Z., Li, Y., 2013a. A SPH simulation on large-amplitude sloshing for fluids in a two-dimensional tank. Earthquake Engineering and Engineering Vibration 12, 135–142. doi:10.1007/s11803-013-0157-1.
- Wang, T., Stevens, L.J., Tan, J.S.C., Davidson, J.F., Wilson, D.I., 2013b. Flow patterns and draining films created by horizontal and inclined coherent water jets impinging on vertical walls. Chemical Engineering Science 102, 585–601. doi:10. 1016/j.ces.2013.08.054.
- Wendland, H., 1995. Piecewise polynomial, positive definite and compactly supported radial functions of minimal degree. Advances in Computational Mathematics 4, 389–396. doi:10.1007/BF02123482.
- Whittaker, T.J.T., Doyle, R., Elsässer, B., 2001. An experimental investigation of the physical characteristics of fast ferry wash, in: 2nd International EuroConference on High Performance Marine Vehicles, Hamburg, Germany, pp. 480–491.
- Yang, Q., Jones, V., McCue, L., 2012. Free-surface flow interactions with deformable structures using an SPH-FEM model. Ocean Engineering 55, 136-147. doi:10. 1016/j.oceaneng.2012.06.031.
- Yun, L., Bliault, A., 2012. High Performance Marine Vessels. Springer Science & Business Media.
- Zhu, Q., Hernquist, L., Li, Y., 2015. Numerical convergence in smoothed Particle Hydrodynamics. The Astrophysical Journal 800, 1–13. doi:10.1088/0004-637X/ 800/1/6.