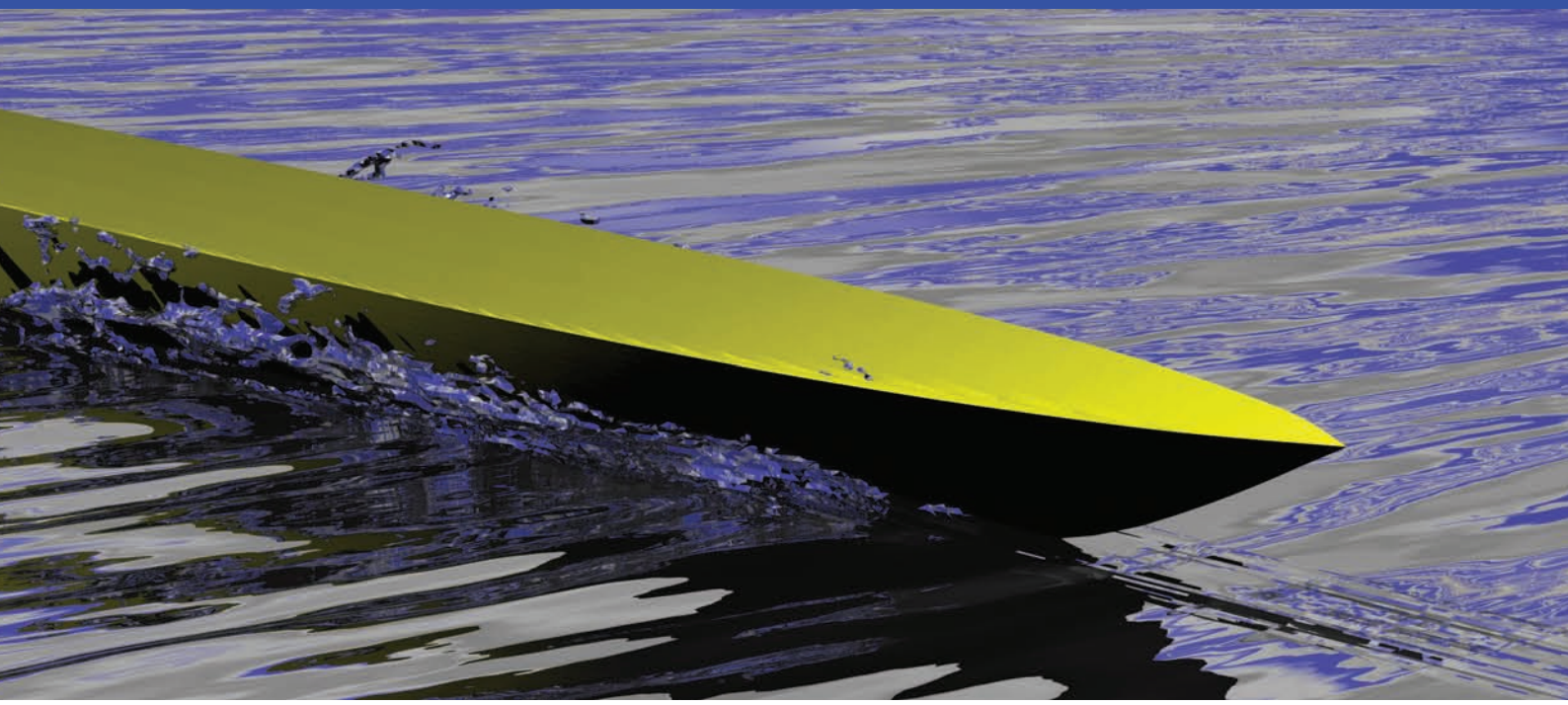
4th SPHERIC workshop



Ecole Centrale de Nantes, France

May 26-29, 2009

Proceedings of the

4th SPHERIC Workshop

May 26 - 29, 2009
École Centrale De Nantes
Nantes, France

Edited by
D. Le Touzé (Centrale Nantes)

Organized by:

Hydrodynamics & Ocean Engineering Group,
Fluid Mechanics Laboratory
Ecole Centrale Nantes, Nantes, France

"The Local Organization Committee 4th SPHERIC Workshop 2009" shall not be responsible for statements or opinions advanced in papers or printed in this volume.
In order to make these proceedings as economically and rapidly as possible, the author's papers have been prepared for final reproduction and printing without any reduction, correction, etc …
Therefore the authors are fully responsible for all the information contained in their papers.
The printing process has been performed in a standard way for all the papers submitted.

Sponsored by:

Published by:

4th SPHERIC Workshop Local Organization Committee

Printed in France (May 2009) by
Goubault Imprimeur
8, rue Thessalie 44240 La Chapelle sur Erdre
Tél. 02 51 12 75 75 - Fax. 02 51 12 75 76

Foreword

The Fluid Mechanics Laboratory of Ecole Centrale Nantes is very pleased to host the 4th international workshop of the SPHERIC community on its campus, which will be the major international event of 2009 dedicated to the SPH (Smoothed Particle Hydrodynamics) method and its applications.

The SPH European Research Interest Community (SPHERIC) aims at fostering the spread of this simulation method within Europe and abroad. Special Interest Group of the wider community ERCOFTAC, SPHERIC serves as a framework for co-operation between research groups working on the subject, and as a platform for the information exchange from science to industry through the assessment of the method validity and applicability in various engineering domains. The SPHERIC community now counts more than 60 academic and industrial members of 21 countries, making its initial clustering objective a real success.

This fourth edition of the SPHERIC workshop is confirming the constant growth of interest in SPH worldwide, with 67 contributions received and more than 100 attendees, that is 30% more than for the third edition. A wide range of topics are covered by the contributions gathered in the present proceedings, from numerical models to engineering applications, from physics to High-Performance Computing. And as every year since 2005, these contributions are of higher and higher quality.

I am therefore very glad to welcome you all in Nantes on behalf of our research group, and I am sure that this year again, the workshop will lead to very fruitful and constructive exchanges in a friendly atmosphere.

David Le Touzé
Chairman of the 4th SPHERIC workshop

Committees

SCIENTIFIC COMMITTEE (SPHERIC STEERING COMMITTEE)

- Damien Violeau (Electricité de France, LNHE, Chatou, France) - Chairman
- Dr. Nathan Quinlan (National Univ. of Ireland, Galway) - Secretary
- Dr. Ben Rogers (Univ. of Manchester, UK) - Webmaster
- Prof. Moncho Gómez-Gesteira (Univ. de Vigo, Spain)
- Dr. Stefano Sibilla (Univ. di Pavia, Italy)
- Dr. Jean-Christophe Marongiu (VA TECH Hydro ANDRITZ, Switzerland)
- Dr. Paul Groenenboom (ESI BV, The Netherlands)
- Dr. David Graham (Univ. of Plymouth, UK)
- Dr. Andrea Colagrossi (INSEAN, Italy)
- Dr. David Le Touzé (Ecole Centrale de Nantes, France)
- Dr. John Biddiscombe (CSCS, Switzerland)
- Prof. Peter Berczik (Ruprecht-Karls-Universität Heidelberg, Germany)

LOCAL ORGANIZING COMMITTEE (ECOLE CENTRALE DE NANTES)

- Dr. David Le Touzé - Chairman
- Prof. Pierre Ferrant
- Dr. Gérard Delhommeau
- Dr. Bertrand Alessandrini
- Dr. Lionel Gentaz
- Dr. Nicolas Grenier
- Matthieu de Leffe
- Anne Lagatu
- Thuy Diem Phung

Contents

Session 4B: Viscosity

Session 5A: Fluids & Structure 1

Session 5B: SDPD & Sedimentation

II. DAY 2: THURSDAY, MAY 28TH, 2009

Session 6: Boundary conditions 1

Session 7: Viscosity & Turbulence

Session 8: SPH-related particle methods

Session 9A: Boundary conditions 2

Session 9B: Sloshing

Session 10A: Free-surface & Parallel computing

Session 10B: ISPH

III. DAY 3: FRIDAY, MAY 29TH, 2009

Session 11: Fluids & Structure 2

Session 12: Fluids & Structure 3 & Stabilization models

Session 13: Free-surface 2

Poster Session

Hydrodynamical instabilities in compressible fluids using SPH

Børve, S.
Norwegian Defence Research Establishment
Kjeller, Norway
e-mail: Steinar.Borve@ffi.no

Price, D. J.
School of Mathematical Sciences
Monash University
Melbourne, Australia

***Abstract*—In 2007, the results from a comparison study of grid based codes and SPH codes on multiphase interactions in compressible fluids were presented [1]. The main conclusion from this study was that whilst the grid codes are able to describe important dynamical instabilities, such as Kelvin-Helmholtz (KH) and Richtmyer-Meshkov (RM), these processes are poorly described by existing SPH codes. It was later shown [17] how the inclusion of artificial conductivity could bring major improvements to the SPH description of the KH instability. The aim of the current work is to compare different SPH formulations and different approaches to adding artificial dissipation on a selection of problems involving hydrodynamic instabilities. We use two widely different, state-of-the-art SPH codes in this study: One code is based on the Regularized SPH (RSPH) method [4] while the other code represents a more conventional approach to SPH [17] with the full "grad-h" formulation [12].**

I. Introduction

Despite the fact that SPH has found its use in an increasingly wide range of problems, including both incompressible and compressible fluids, there are still questions being raised concerning quite fundamental aspects of the method. Among the more important issues still being investigated, is the question of how well SPH can describe hydrodynamic instabilities and multi-phase flows which involve sharp density interfaces. A recent study pointed towards fundamental deficiencies in the SPH treatment of compressible instabilities [1]. In response to this, it was documented how the use of artificial conductivity could greatly improve SPH descriptions of the Kelvin-Helmholtz (KH) instability [17]. A comparison of RSPH with two different Eulerian solvers found in the commercial software package AUTODYN showed the former method to treat Richtmyer-Meshkov instabilities related to high-explosives simulations quite well. In addition to these recent results, several SPH models developed specifically for multi-phase flows have surfaced over the past 10 years [6], [8], [19].

The aim of the current work is to study more closely how 3 different SPH formulations handle two-dimensional hydrodynamic instabilities in compressible fluids. In addition to the standard formulation (as used in e.g. [4]), we evaluate the multiphase formulations of Ritchie & Thomas (RiT) [19] and Hu & Adams (HuA) [8]. The RiT formulation was developed with astrophysical applications in mind and was to some extent evaluated already in [17]. It effectively uses a smoothed pressure and finds the density by

$$\rho_a = \frac{\sum_b m_b u_b W_{ab}}{u_a}, \tag{1}$$

where m_a and u_a are the mass and thermal energy of particle a. The time derivative of u_a is correspondingly found (in the absence of dissipation) as

$$\frac{du_a}{dt} = \frac{\gamma - 1}{\rho_a} \sum_b m_b u_b \boldsymbol{v}_{ab} \cdot \nabla_a W_{ab}, \tag{2}$$

where γ is the ratio of specific heats and $\boldsymbol{v}_{ab} = \boldsymbol{v}_b - \boldsymbol{v}_a$ is the velocity difference between particles a and b.

The HuA formulation was proposed for weakly incompressible flow problems where large shifts in density is reflected in a corresponding shift in particle mass. With this formulation, the density is found using the number density estimate as

$$\rho_a = m_a \sum_b W_{ab} = m_a \sigma_a. \tag{3}$$

The time derivatives of $\boldsymbol{v}_a$ and u_a are similarly found as

$$\frac{d\boldsymbol{v}_a}{dt} = -\frac{1}{m_a} \sum_b \left(\frac{P_b}{\sigma_b^2} + \frac{P_a}{\sigma_a^2} \right) \nabla_a W_{ab} \tag{4}$$

and

$$\frac{du_a}{dt} = \frac{P_a}{\rho_a \sigma_a} \sum_b \boldsymbol{v}_{ab} \cdot \nabla_a W_{ab}. \tag{5}$$

(The energy equation was obviously not included in the original work since it dealt with a simpler equation of state [8].) In the case of compressible flow, density is not a fixed material property. Therefore, it is normally recommended to use equal-mass particles in simulations, in which case the HuA formulation becomes identical to the standard formulation. When particle regularization is used, variations in density will always result in similar variations in the particle masses. For this reason, we include the HuA formulation in this discussion.

In addition to the 3 different formulations, we also consider 2 options for adding dissipation. It was argued in [17] that the failure of standard SPH to simulate the KH instability came from the lack of a dissipation source associated with discontinuities in temperature. It was proposed to introduce an artificial conductivity term acting alongside the commonly

A conservative SPH method for interfacial flows with surfactant dynamics

S. Adami
Institute of Aerodynamics
Technische Universität München
Munich, Germany
stefan.adami@aer.mw.tum.de

X.Y. Hu
Institute of Aerodynamics
Technische Universität München
Munich, Germany

N.A. Adams
Institute of Aerodynamics
Technische Universität München
Munich, Germany

***Abstract*—In this work, a Lagrangian particle method is proposed for the simulation of multiphase flows with surfactant. The model is based on the multi-phase smoothed particle hydrodynamics (SPH) framework of Hu and Adams [1]. Surface-active agents (surfactants) are incorporated into our method by a scalar quantity describing the local concentration of molecules in the bulk phase and on the interface. The surfactant dynamics are written in conservative form, thus global mass of surfactant is conserved exactly. The transport model of the surfactant accounts for advection and diffusion. Within our method, we can simulate insoluble surfactant on an arbitrary interface geometry as well as interfacial transport such as adsorption or desorption. The flow-field dynamics and the surfactant dynamics are coupled through a constitutive equation, which relates the local surfactant concentration to the local surface-tension coefficient. Hence, the surface-tension model includes capillary and Marangoni forces. The present numerical method is validated by comparison with analytic solutions for diffusion and for surfactant dynamics. More complex simulations of an oscillating bubble, the bubble deformation in a shear flow, and of a Marangoni-force driven bubble show the capabilities of our method to simulate interfacial flows with surfactants.**

I. Method overview

Here, we present only a short overview of the smoothed particle hydrodynamics (SPH) method we have developed to simulate multi-phase flows with surfactant. For more detailed information the reader is referred to Adami et al. [2].

A. Governing equations

The isothermal Navier-Stokes equations are solved on a moving Lagrangian frame and surface tension effects are included via the continuum-surface-tension model (CSF). As we model incompressible flow with the weakly-compressible approach, a stiff equation of state (EOS) is used to relate the pressure to the density, i.e.

$$p = p_0 \left(\frac{\rho}{\rho_0} \right)^{\gamma} + b \ , \tag{1}$$

with $\gamma = 7$, the reference pressure p_0, the reference density ρ_0 and a parameter b. These parameters and the artificial speed of sound are chosen following a scale analysis presented by Morris et al. [3] which determines the threshold of the admissible density variation.

The evolution of surfactant on the interface is governed by an advection-diffusion equation with a source term accounting for the surfactant transport between the bulk and the phase interface, e.g. adsorption and desorption. After integration over the domain, the variation of the total mass m_s of the interfacial surfactant is given by

$$\frac{dm_s}{dt} = \int_V \nabla_s \cdot \mathbf{D_s} \nabla_s \Gamma \delta_\Sigma \ dV + \int_V \dot{S_\Gamma} \delta_\Sigma \ dV. \tag{2}$$

Here, Γ, $\mathbf{D_s}$, $\dot{S_\Gamma}$ and δ_Σ are the interfacial surfactant concentration, the diffusion coefficient matrix (in case of isotropic diffusion $\mathbf{D_s} = D_s \cdot \mathbf{I}$), the source term and the surface delta function, respectively.

Assuming that each surfactant molecule can move freely in the bulk phase, the transport of surfactant can be described by the advection-diffusion equation. In conservative form, the rate of change of the total surfactant mass in the liquid M_s is obtained by

$$\frac{dM_s}{dt} = \int_V D_\infty \nabla^2 C \ dV - \int_V \dot{S_\Gamma} \delta_\Sigma \ dV \tag{3}$$

with the bulk diffusion coefficient D_∞ and the volumetric surfactant concentration in the liquid C. The second term on the right side of eq. (3) is equal to the second term on the right hand side of eq. (2), hence ensures global mass conservation.

To close our model, we relate the interfacial surfactant concentration Γ to the surface-tension coefficient α by a constitutive equation $\alpha = f(\Gamma)$ following the model of Otis et al. [4]. Note that the use of other relations, such as the Frumkin isotherm or the Langmuir model [5]–[8], is straightforward.

B. Numerical model

The governing equations are discretized using the multi-phase SPH method of Hu and Adams [1]. Each particle represents a Lagrangian element of fluid, carrying all local phase properties. With updating the positions of the particles, this method accounts for advection as the governing equations are formulated in terms of material derivatives.

Using the color function c to distinguish different phases in the system, particles with a non-vanishing color-function gradient ∇c approximate the singularity at an interface as a

A SPH multiphase formulation with a surface tension model applied to oil-water separation

N. Grenier, D. Le Touzé
Laboratoire de Mécanique des Fluides
CNRS UMR 6598
École Centrale de Nantes
1 rue de la Noë, Nantes, France
nicolas.grenier@ec-nantes.fr

A. Colagrossi
INSEAN, The Italian Ship Model Basin
Via di Vallerano 139,,Roma, Italy
CeSOS, Centre of Excellence for Ship
and Ocean Structures, NTNU
Trondheim,Norway
a.colagrossi@insean.it

M. Antuono
INSEAN, The Italian Ship Model Basin
Via di Vallerano 139, Roma, Italy

***Abstract*—A SPH model [1] addressing the specific issue of accurately evaluating the quantities which are discontinuous across the interface is used to simulate multiphase flows together with the presence of a free surface (meaning here an interface without accounting for the (air) phase above it). This work is an extension of the formulation discussed in [2] and it is based on the variational approach introduced by [3].**

Physical models have been added in the formulation to account for viscous and surface tension effects. Classical tests for which reference solutions are available in literature are used to validate the latter models, such as the Poiseuille flow or an oscillating droplet without gravity. More complex validation case is then simulated such as an air bubble rising by gravity in a water column at rest. Finally, the model capabilities are illustrated on the case of a simplified oil-water separator.

I. Introduction

Multiphase flows play a significant role in numerous engineering applications characterized by strong dynamics of the flow making the SPH scheme a valuable candidate as simulation method, e.g. flows involved in mixing/separation devices, engines, propellers with cavitation, etc. Even for free-surface water flows of strong dynamics (i.e. including jets, sprays, impacts, free-surface reconnections, etc.) usually simulated using one-phase SPH models, the air phase can have a large influence on the water flow evolution and on subsequent loads on structures.

Although the classical SPH formulation succeeds in correctly simulating one-phase flows, the presence of an interface and the physical conditions associated make a stable two-phase formulation more difficult to derive. The main issue is the estimation of the ratio between pressure gradient and density in the momentum equation in the region near the interface, where the density is discontinuous when crossing the interface. Actually, the SPH scheme relying on a smoothing, accuracy is lost when a particle has its compact support which intersects the interface, namely the density of the other phase spuriously influences the evaluation of acceleration of the concerned particle.

In the present work a SPH model for simulating interface flows is reminded [1]. This model is an extension of the formulation discussed in Colagrossi & Landrini [2] and it is based on the variational approach introduced by Bonet & Lok [3]. This SPH formulation presented shows strong similarities with one proposed by Hu & Adams [8] to study multiphase flows. The main difference between these two models is exposed in [1].

The formulation is validated on test cases for which reference solutions are available in literature. After classical tests as the Poiseuille flow and the one of a droplet oscillating without gravity, more complex validation cases are simulated such as an air bubble rising by gravity in a water column at rest. The last problem studied in the present work is the case of a collection of oil bubbles evolving in a water column at rest.

II. Physical model

The model presented in the present work uses the Navier-Stokes equations for a set of viscous newtonian fluids. The sketch in Fig. 1 shows a fluid domain Ω composed by different fluids $\mathcal{A}, \mathcal{B}, \ldots$. The boundaries of the domain Ω are constituted by a free-surface $\partial\Omega_F$ and by solid boundaries $\partial\Omega_B$.

The conservation of the momentum in Ω is written in lagrangian formalism as

$$\frac{\mathrm{D}\mathbf{u}}{\mathrm{D}t} = -\frac{\nabla p}{\rho} + \mathbf{F}_V + \mathbf{F}_S + \mathbf{F}_B \tag{1}$$

where $\mathbf{u}$, p and ρ are respectively, the velocity, the pressure and the density fields, while $\mathbf{F}_V$, $\mathbf{F}_S$, $\mathbf{F}_B$ represent the viscous, the surface tension and the external body forces (here the force field $\rho\mathbf{F}_B$ is the gravity force $\rho\mathbf{g}$).

The spatial position of the generic material point $\mathbf{X}$,at time t will be indicated through $\mathbf{x}(t)$, in other words

$$\mathbf{x}(t) = \phi(\mathbf{X}, t) \tag{2}$$

where ϕ is the map which link the lagrangian coordinates $\mathbf{X}$ with the physical ones $\mathbf{x}$.

Weakly compressible fluids are considered. Under this assumption the pressure field can be directly linked to the density

Comparison and Evaluation of Multi-phase and Surface Tension models

Benedict D. Rogers
School of Mechanical, Aerospace & Civil Engineering
University of Manchester
Manchester, United Kingdom
benedict.rogers@manchester.ac.uk

Julien Leduc[1,2], Jean-Christophe Marongiu[1,2], Francis Leboeuf[1]
[1]Laborartory of Fluid Mechanics and Acoustics (LMFA),
Ecole Centrale de Lyon, Université de Lyon
Ecully, France
[2]ANDRITZ Hydro, Vevey, Switzerland
julien.leduc@andritz-hydro.com, Jean-Christophe.Marongiu@andritz-hydro.com,
Francis.Leboeuf@ec-lyon.fr

***Abstract* — During the last decade several different models have been proposed to simulate multi-phase flows with SPH. The aims of these models differed and thus their characteristics also differe. Furthermore, several different formulations for surface tension have also been proposed. In this paper we show advantages and difficulties arising from some of these models to simulate multi-phase flow with and without surface tension. We compare the performance of three compressible two-phase models: the variationally consistent approach of Colagrossi & Landrini (2003) [4], a volume-fraction formulation (Grenier et al. (2008) [7] and a new approach using a Arbitrary Lagrangian-Euler (Leduc et al. (2009) [10]. The tests were chosen to deal with low (static bubble) or high kinematic cases. We present results for several two-dimensional cases including the frequency of oscillation of a gravity wave compared with an analytical result, the collapse of a 2-D water column and to test different surface tension formulations we examine the pressure profiles and parasitic velocities for the Young-Laplace problem for a fluid cylinder. The preliminary results show that while the results from the different formulations are similar, differences exist in the results that are sufficient to be significant.**

I. INTRODUCTION

The physical processes of potentially highly violent free-surface hydrodynamics such as occurs during wave breaking at beaches or in high-head driven jet-powered pelton turbines in hydro electricity generation are highly complex multi-phase multiscale problems involving highly nonlinear deformation of the fluids. To date, mono-phase SPH simulations have produced remarkable results considering that only the water phase is simulated and the second air phase is entirely neglected and is considered as an absence of water particles (Dalrymple & Rogers[6], Marongiu et al. [11]). However, many of the problems being studied require the inclusion of the both the water and air phases since the presence of the air can have important secondary effects such as during wave impact with the entrapped pockets, the dissipation of breaking waves with the topologically-induced gyre and an improved predictive capability for the jet impact and spray in the pelton turbine leading to efficiency gains. The simulation of accurate and robust multi-phase physics represents the next logical step in the development of free-surface SPH, but as will be briefly outlined in this paper, this area has received development research so far. The motivation behind the present work is an assessment of the state-of-the-art for multi-phase simulations with the eventual aim of simulating the highly nonlinear physical processes mentioned above.

An overview of the current published work on multi-phase SPH reveals that this area of research is sparse and has not been thoroughly investigated. For air-water simulations, the density difference is 1:1000 and therefore represents a significant challenge. For compressible flows involving air-water mixtures several different approaches appear the most attractive:

(i) Nugent-Posch [14] approach augmented by Colagrossi & Landrini [4], using different equations of state for each fluid with some changes to the summations

(ii) Hu & Adams [8], remove troublesome density outside the summation, include surface tension

(iii) Volume Fraction approach (Grenier et al.[7], Chanteperdrix [3])

Within each paper, it is not clear that any one formulation is better adapted for different kinds of test cases. In the work presented herein, we have chosen to present our results from the Colagrossi & Landrini formulation [4], the Arbitrary Lagrange Euler (ALE) volume fraction formulation of Grenier et al. [7] and a new formulation using an acoustic Riemann solver [10] also being presented in this workshop by Leduc et al. The Hu & Adams [8] compressible multi-phase formulation is not covered here and so remains to be fully investigated.

Other issues that are problemetic include how to formulate viscous effects across the interface between two phases such that both linear and angular momenta are conserved (Hu & Adams [8]). This could have profound influence on the

Formulating surface tension with reproducing divergence approximation for multi-phase SPH

X. Y. Hu, S. Adami and N. A. Adams
Lehrstuhl für Aerodynamik, Technische Universität München
85748 Garching, Germany

***Abstract*—In this work, we propose a new surface-tension method for multi-phase SPH. To obtain stable and accurate surface curvature, a new reproducing divergence approximation without requiring matrix inversion is derived. To reflect the reality of asymmetric distribution of surface force, a new density-weighted color gradient formulation is used. Numerical results show that, while the new formulation has comparable accuracy as our previous formulation (Hu and Adams 2006), it can achieve much faster computation for problems with large density ratio.**

I. Introduction

Surface-tension effect is important for many multi-phase flow phenomena, especially, when the interested flow has sufficient small characteristic length scales. Comparing to other traditional methods, SPH (Monaghan 2005) method has many benefits when modeling multi-phase flow because the material interface can be represented conveniently with its fully Lagrangian formulation.

Generally, there are two approaches to model the surface-tension effect in SPH: one is based on microscopic inter-phase attractive potential; the other is based on macroscopic surface-tension model. Though the approach using inter-phase attractive potential is easy to implement, one of the difficulties is that the resulting surface tension need to be calibrated with numerical simulation. Furthermore, with given parameters, the surface tension is resolution dependent and does not converge to a fixed value with increasing resolution. On the contrast, the approach using surface-tension model always gives the prescribed surface tension which converges with increasing resolution. To implement the surface-tension model in SPH, usually, the continuous surface force (CSF) method is used. In this method, a color function is used to describe different phases, and the interface is defined as a finite transitional band in which the color gradient does not vanish.

The early formulation of the surface-tension model, which uses a smoothed color function, has difficulties on predicting the surface curvature, which is the divergence of the unit interface normal direction. One is because the standard SPH approximation of divergence needs full support of the kernel function, which cannot be satisfied by the finite transitional band. The other is because the small magnitude of color gradient near the edge of the finite transitional band may have an erroneous direction. In our previous formulation (Hu and Adams 2006), a sharp, other than smoothed, color function with discontinuity is used directly. The calculation of surface curvature is totaly circumvented by introducing a surface-stress tensor which is only dependent on the color gradient. Furthermore, since the magnitude of the surface tensor is proportional to the magnitude of the color gradient, the contribution of a small color gradient vanishes hence does not introduce numerical difficulty.

In reality, the surface force may not be distributed beside the interface evenly. One example is that, beside the air-water interface, the surface force is dominantly acting on the water side. Up to now, the surface force modeled by SPH is assumed being evenly distributed beside the interface. Not only nonphysical, this assumption can also introduce numerical difficulties. For example, in an air-water interfacial flow, the surface force on the air side can introduce an acceleration about 1000 times of that on the water side! Consequentially, the stiffness of the equation of motion is increased dramatically and the step-size for time integration is strongly limited.

In this work, we revisit the formulation of surface curvature based on a sharp color function. To obtain stable and accurate surface curvature without full support of the kernel function, a new reproducing divergence approximation is derived. Unlike previous formulations which calculate the divergence from reproducing gradient approximations, the new approximation does not require a matrix inversion. Furthermore, we have not found notable effects caused by the errors due to the small color gradient. To reflect the reality of asymmetric distributed surface forces, a new density-weighted color gradient formulation is used. Several numerical tests on static water drop, oscillating drop, drop deformation and splitting in shear flow are carried out to show the potential of the present method. The results show that, while the new formulation has comparable accuracy as our previous formulation (Hu and Adams 2006), it can achieve much faster computation for problems with large density ratio.

II. Governing equations

The isothermal Navier-Stokes equations are solved on a moving Lagrangian frame

$$\frac{d\rho}{dt} = -\rho\nabla\cdot\mathbf{v}\ , \tag{1}$$

$$\frac{d\mathbf{v}}{dt} = \mathbf{g}+\frac{1}{\rho}\left[-\nabla p+\mathbf{F}^{(\nu)}+\mathbf{F}^{(s)}\right]\ , \tag{2}$$

SPH Van der Waals model for Multiphase Flows

Alexandre M. Tartakovsky
Computational Mathematics
Pacific Northwest National Laboratory
Richland, USA
alexandre.tartakovsky@pnl.gov

Kim Ferris
Computational Mathematics
Pacific Northwest National Laboratory
Richland, USA
Kim.Ferris@pnl.gov

Paul Meakin
Center for Advanced Modeling and Simulation
Idaho National Laboratory
Idaho Falls, USA
Paul.Meakin@inl.gov

***Abstract*—A Lagrangian particle model for multiphase fluid flow, based on smoothed particle hydrodynamics (SPH), was developed and used to simulate the flow of a multiphase fluid consisting of bubbles of a non-wetting fluid surrounded by a wetting fluid. In SPH simulations fluids are represented by a set of particles that are used as discretization points to solve the Navier-Stokes fluid dynamics equations. In the multiphase SPH model, the van der Waals equation of state is used to close the system of flow equations. The combination of the momentum conservation equation with the van der Waals equation of state results in a particle equation of motion in which the total force acting on each particle consists of many-body repulsive and viscous forces, two-body (particle-particle) attractive forces, and body forces such as gravitational forces. Based on similarities between SPH and molecular dynamics, the behavior of the fluids away from interfaces is governed by purely hydrodynamic (Navier-Stokes) forces, and the surface tension at fluid-fluid interfaces is imposed through component dependent attractive forces. Similarly, the wetting behavior of the fluids is controlled by attractive interactions between the fluid particles and stationary particles that represent the solid phase. Comparison with analytical solutions for relatively simple flows demonstrates the accuracy of the SPH model.**

I. Introduction

Multiphase flows are encountered in many important engineering, petroleum and environmental applications. Numerical continuum-scale modeling is complicated by the strong nonlinearity of the Navier-Stokes equations describing multiphase flow. The fluid-fluid-solid interactions present additional challenges because they are a result of molecular interactions that cannot be adequately described by the standard (no-slip) boundary conditions commonly used in conjunction with the Navier-Stokes equations to simulate fluid dynamics. Simple and efficient multi-scale numerical methods, able to reproduce both the continuum bulk behavior of the fluids and complex fluid-fluid-solid interactions, are needed to simulate multiphase flows on scales of practical interest. Particle methods such as Smoothed Particle Hydrodynamics (SPH) [1]–[3] and Dissipative Particle Dynamics (DPD) [4], [5] have been extensively used to model complex binary fluids and the separation of fluid mixtures into phases with different compositions. In this paper we present a new multiscale Lagrangian particle method, based on smoothed particle hydrodynamics, for the flow of immiscible fluids.

The isothermal flow of two immiscible fluids, A and B, is governed by a combination of linear momentum conservation equations and continuity equations subject to the Young-Laplace condition for the pressure change at curved interfaces separating the two fluid phases:

$$P_A - P_B = \kappa\sigma, \tag{1}$$

where P_I, $I = A, B$, is the pressure of fluid I, κ is the curvature of the interface separating fluids A and B, and σ is the surface tension.

At the fluid-fluid-solid interface, a boundary condition prescribing the contact angle formed by the fluid-fluid interface and the solid surface is commonly used. Under thermodynamic equilibrium conditions, simple energy considerations lead to the expression

$$\cos(\theta) = (E_{SA} - E_{SB})/E_{AB}, \tag{2}$$

for the static contact angle θ, where E_{SA} is the specific interfacial energy (energy per unit area) between the solid and fluid A, E_{SB} is the specific interfacial energy between the solid and fluid B, and E_{AB} is the specific interfacial energy between fluid A and fluid B. In the presence of flow, the contact angle depends on the direction of flow (whether fluid A is displacing fluid B or fluid B is displacing fluid A), the velocity of the fluid-fluid interface at the solid surface and the direction of motion of the fluid-fluid-solid contact line if the surface is physically or chemically anisotropic. A variety of empirical models for the contact angle [e.g [6], [7]] have been proposed, but most of them work only for a limited range of parameters [[8] and references therein] and wetting behaviors.

A more serious complication in modeling multiphase fluid flow arises due to the fact that the classical no-slip boundary condition at fluid-fluid-solid interfaces implies that the strain rate diverges at the contact line [9]. In practice, this singularity does not exist because of the molecular (particle) nature of real fluids. One consequence of the molecular nature of fluids is a microscopic slip length (the ratio between the slip velocity and the strain rate at the interface). In large scale continuum computational fluid dynamics applications, the systems size is much greater than the slip length and no slip boundary conditions at solid-fluid interfaces are accurate and well established. However, non-slip boundary conditions cannot be used at the contact line.

A number of meso-scale numerical particle-based techniques such as lattice Boltzmann methods and dissipative particle dynamics have been used to simulate multiphase fluid

Specific pre/post treatments for 3-D SPH applications through massive HPC simulations

G.Oger, C. Leroy, E. Jacquin
HydrOcean
Nantes, France
guillaume.oger@hydrocean.fr

D. Le Touzé, B. Alessandrini
Laboratoire de Mécanique des Fluides
Ecole Centrale Nantes
Nantes, France

***Abstract*— The solver SPH-Flow is developed under a consortium currently leads by Ecole Centrale Nantes and HydrOcean, and involving academic and industrial partners. SPH-flow is a 3-D parallel SPH solver, able to compute fluid and multiphysic simulations involving structures, fluid structure interactions, multiphasics or thermics … on complex geometries and topologies. The current version of this solver is mainly dedicated to the simulation of high dynamic phenomena, or so complex topologies that classical meshed based solvers quickly fail. An important effort has been done in order to improve the SPH model towards more accuracy and robustness, and has been developed with the objective to reach very high HPC performances on thousands of processors.**

The recent developments of SPH-flow performed at HydrOcean are dedicated to the improvement of the functionalities that are needed to perform industrial applications: pre and post treatments, massive and efficient parallelisation, ability to perform computation on complex geometries and topologies.

Concerning the initialization of the calculation, the main problem concerning pre-treatment is related to the automatic filling of any particle tank including immersed geometries. The current meshing software are extremely efficient for mesh based solver for which grid criteria are far from SPH requirement such as inter-particle distance. In order to overcome this difficulty, this paper presents a particle seeding method named "Bubbles". Based on Van der Waals particle interactions, this algorithm proved its capabilities in terms of regularity of the initial particle space distribution.

A second specificity of SPH compared with classical meshed based methods concerns the difficulty to visualize the results based on million of particles at each time step. Classical visualization tools are able to efficiently deal with heavy meshed-based datasets, but are not well adapted to particle rendering. In order to overcome this difficulty, a specific Marching Cube algorithm has been developed, allowing converting scattered data in meshed surfaces. Recent improvements on the efficiency of the parallelization algorithms are also highlighted, allowing to reach efficiency of about 90%. Then, the validations and applications on very complex 3D geometries are presented, showing the beginning of the maturity of SPH solver for industrial applications.

I. INTRODUCTION

The extension of SPH from 2D to 3D problems can be achieved rather rapidly in terms of formulation. However, such extension leads to new needs in terms of computational resources and pre/post treatments capabilities. First of all, the scattered data aspect of this method makes the initialization of the fluid domain possibly difficult, namely when complex tanks and immerse body geometries are involved. A possible solution is to use a particle topology obtained through an unstructured meshing software. But such a technique is problematic due to the needs of SPH for equally spaced particle topologies, that current meshing software do not use as a criteria (they mainly use skeeness, volume …). In this paper, the *Bubble* algorithm is presented as an alternative for 2D and 3D applications involving complex geometries, due to its ability to distribute particles in the computational domain with a very regular space distribution. This presentation is followed by a global description the SPH-flow software, through its various implemented formulations, clearly oriented towards multi-species and multi-physic solutions through massive HPC simulations. A discussion concerning 2D and 3D boundary treatments, with a special attention paid on the ability of imposing ghost particle boundary conditions for any complex geometries is provided. Another specificity of 3D SPH calculations concerns the global computational costs. Indeed, because of the large amount of operations required associated with the imposition of small time steps, the parallelization of the 3D SPH model sounds unavoidable. In this paper, the parallelization efficiency of SPH-flow is presented and discussed, through large improvements recently obtained. Finally, after having overcome the difficulties inherent to fluid domain initialisation, boundary conditions and calculation acceleration through parallelization, a last difficulty would concern the visualization of 3D SPH data, which often result in incomprehensible messy particle movements, for which

A fast algorithm for free-surface particles detection in 2D and 3D SPH methods

Marrone S., Colagrossi A.
INSEAN, The Italian Ship Model Basin
Rome, Italy
a.colagrossi@insean.it

Le Touzé D.
Fluid Mechanics Laboratory
Ecole Centrale Nantes/CNRS
Nantes, France
David.LeTouze@ec-nantes.fr

Graziani G.
Department of Mechanics and Aeronautics
University La Sapienza
Rome, Italy
g.graziani@uniroma1.it

***Abstract*—The present paper aims at proposing a fast algorithm permitting to detect the free-surface in particle simulations, and next to define a level-set function on a cartesian grid. The latter allows for analyzing in detail the free surface shape of three-dimensional simulations as well as internal features of the flow, by using standard visualization tools. The algorithms proposed for detecting the free-surface particles are described in both two and three dimensions, and are validated on simple and complex flow simulations. Then, the derivation of the level-set function is detailed and the usefulness of the proposed method to post-process and analyze complex flows are illustrated on realistic 2D and 3D examples.**

I. Introduction

In recent years the SPH method has been successfully applied to problems involving free-surface flows with fragmentation. In order to analyze flows with complex free surface patterns (fragmentations, air entrapment, etc.) and to face a larger range of problems it is useful to know which particles belong to the free surface. The detection of the free surface, indeed, allows giving suitable boundary conditions along it (surface tension, isothermal condition, etc) and, consequently, to deal with different physical phenomena and flow behaviors. Dilts [1] developed an algorithm for the free-surface tracking that can detect free-surface particles in a robust and reliable way and is applicable to any meshless method. However it is quite difficult to implement, especially in its extension to three-dimensional simulations [2].

In this work a novel algorithm for the free-surface detection is presented. Such a scheme, based on the properties of the SPH kernel, is easy to implement both in two and three dimensions, and computationally cheap (CPU time required is an order of magnitude lower than particle interaction calculation). The accuracy of the method is comparable to the method proposed by Dilts; it is possible, indeed, to catch small cavities of radius up to h (h being the smoothing length) and fluid elements with dimension even smaller than h (like jets and drops).

Thanks to these valuable features the proposed algorithm can be used at each time-step of the simulations, without an appreciable increase of the CPU-time, and allows enforcing specific boundary conditions along the free surface.

Moreover, the free-surface detection permits to strongly improve the post-processing phase, particularly of three-dimensional simulations with complex flow features. In fact, if one uses merely a SPH output, flow analysis is problematic since data are known on scattered points, and it is difficult to obtain contour plots, slices and iso-surfaces. Such an analysis could be performed in a straightforward way if data were interpolated on a regular grid. This operation can be carried out with a Moving-Least Square (MLS) interpolator that can exactly interpolate a linear field on a regular grid from scattered points. However, in the presence of a free surface, it is a bit more complicated since the interpolation will lead to extrapolated values on nodes out of the fluid domain, at least as far as a distance $\mathcal{O}(h)$ from the free surface. In such cases, if particles of the free surface are known, it is possible to define a distance function among the grid nodes from the free surface and, in this way, to distinguish between nodes inside and outside the fluid domain.

Such an interpolation of field values on a regular grid can be also useful to introduce a "remeshing" technique. In fact, a noisy particle distribution associated with the fluid evolution leads to large inaccuracies [4]. This problem can be overcome by a periodic resetting of the particle positions onto a regularized set of nodes.

In the present paper the algorithm for free-surface detection is described and validated in sections 2 and 3, for both two- and three-dimensional cases. To assess the accuracy of the algorithm the validation has first been performed using simple geometries, and then on complex flow cases. Finally, in section 4, the procedure for the interpolation on a regular grid is described and post-processing results are shown to illustrate the proposed algorithm capabilities.

II. 2D algorithm

A. Algorithm details

The algorithm is composed of two steps: in the first one the properties of the renormalization matrix, defined by Randles and Libersky [6], are used to find particles next to the free surface. This first step strongly decreases the number of particles that will be processed in the second step. In the second step the algorithm, by means of geometric properties,

HIGH PERFORMANCE OF SPH CODES: Best approaches for efficient parallelisation on GPU computing

A. J. C. Crespo
School of Mechanical, Aerospace and Civil Engineering,
University of Manchester
Manchester, UK
Alejandro.Crespo@manchester.ac.uk

E. Parkinson
Department of Hydraulic Research
ANDRITZ Hydro
Vevey, Switzerland
Etienne.Parkinson@andritz-hydro.com

J. C. Marongiu
Laboratory of Fluid Mechanics and Acoustics
Ecole Centrale de Lyon, University of Lyon
Ecully, France
jean-christophe@ec-lyon.fr

M. Gómez-Gesteira
Grupo de Física de la Atmósfera y del Océano
Universidade de Vigo
Ourense, Spain
mggesteira@uvigo.es

J. M. Domínguez
Grupo de Física de la Atmósfera y del Océano
Universidade de Vigo
Ourense, Spain
jmdalonso@gmail.com

***Abstract*— Graphics Processing Units (GPUs) appear as an accessible alternative to accelerate SPH models using a powerful parallel programming model. The proposed work introduces the framework to implement SPH codes using the best approaches and performance optimization techniques. As a first step the implementation is focused on solving the particle interactions on GPU computing since SPH models spends most of its computational time on that.**

I. INTRODUCTION

Even if they can be easily programmed, general purpose processors (CPU, Central Processing Unit) are not the very best architecture on which one can make fast scientific computations. Dedicated devices can be designed for specific applications, with the only purpose of executing given operations. They can easily outperform CPUs. But they require particular competences and are not versatile enough to allow easy development of the code. Among those dedicated devices, Graphics Processing Units (GPUs) are designed to treat a high data flow and to render pixels (at a few tens of frames per second). From a computational point of view they are highly efficient. Moreover, driven by the market of video games, their computing power has increased much faster than the one of CPUs (see Figure 1).

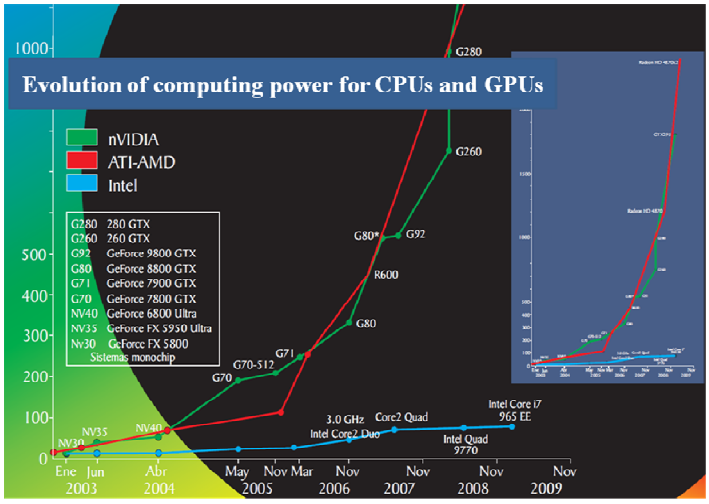


Figure 1. Evolution of computing power for CPUs and GPUs from [1]

Modeling Water Waves in the Surf Zone with GPU-SPHysics

Alexis Herault, Anna Vicari, Ciro del Negro
INGV - Sezione di Catania
Piazza Roma, 2
95123 Catania, Italy
herault@ct.ingv.it, vicari@ct.ingv.it, delnegro@ct.ingv.it

Robert A. Dalrymple
Department of Civil Engineering
John Hopkins University
3400 NO. Charles St., Baltimore, MD 21204
rad@jhu.edu

***Abstract*—Water waves impinging on a beach provide a good example of the ability of SPH to model complex free surface flows. The formation of a plunging breaker was shown early with the pioneering work of Monaghan [1]. Dalrymple and Rogers [2] examined breaking waves on a beach, noting that the initially two-dimensional wave field becomes three-dimensional when the waves break. Recently Herault et al. [3] have developed a version of the SPHysics open source model (http://wiki.manchester.ac.uk/sphysics) for Nvidia graphics cards using the CUDA extended C++ programming language. Here we show 2D and 3D results using several Nvidia cards and demonstrate the speed-ups achievable with GPU programming versus CPU programming. Our speed up results are for three generations of Nvidia cards, including the new Tesla card, which has 240 streaming processors available. This paper will discuss the Nvidia CUDA code to point out the differences between using the GPU versus the CPU, the various methods used in the GPU model for such tasks as neighbor lists, the memory (shared, global, texture models), and the uses of CUDA kernels. We will then present applications of the model to different water waves at beaches. The waves are created within 2D or 3D wave basins, with sloping beaches. The wavemaker is currently a sinusoidally forced flap wavemaker, but that is easily changed. As the waves shoal and break on the slope, the motion clearly becomes three-dimensional.**

INTRODUCTION

The introduction of the massively parallel graphic processors (GPU) and particularly of the first compiler (NVidia CUDA) allowing the programming outside of a strictly graphic context, of those processors, has changed the game in terms of numerical simulation.

Indeed a graphic card of the last generation has a computational power of about 1TFlops and costs only 300 €. It is therefore quite natural to use the GPU for scientific computation and numerical simulation.

Regarding the hardware, one could consider the GPU as a massively parallel computer accessing a global *device memory*, holding hundreds of computing units employing a new architecture called Single Instruction Multiple Threads (SIMT). This architecture is akin to the Single Instruction Multiple Data (SIMD) vector organization. The NVidia GPU of the last generation (G200) has 240 processors, and a device-to-device memory bandwidth of about 140 Gb/s for a computational power of about 1 TFlop.

At this time, only NVidia distributes a C compiler dedicated to those processors, CUDA, which has a certain number of restrictions: the most important ones being the impossibility of writing recursive functions and the lack of dynamical allocation in device memory. Also, we can only hope to obtain optimum performance by conforming closely to the SIMT scheme. Indeed, for compute bound operations if the scheme is respected (except for some few exceptions) we are certain to have a code whose performances scales linearly with the number of processors. In addition, one must also acknowledge that the memory bandwidth between the graphic card and the host via the PCI bus is one-to-two orders of magnitude slower than the one for the graphics device memory.

These constraints currently make impossible the direct transformation of a numeric code, even parallelized, into a GPU version. The use of GPU as a mathematical coprocessor is also quite limited by the low bandwidth of the PCI bus.

Because of their structure and low data requirements, the SPH method is particularly well adapted to the GPU. In fact, except in a few rare cases, notably the treatment of solid boundaries, the calculation of forces, speed and positions is done rather in the same manner for all particles, in a SIMT fashion.

In this context we have developed a GPU version of the SPhysics code. To take the most advantage of the GPU performances, the ensemble of the algorithms is executed on the GPU, the transfers between host and GPU happening only to record the results of the calculation.

I. PREVIOUS WORK

The first implementations of the SPH method totally on GPU were done by Kolb and Kuntz [4], and Harada et al. [5] (both occurring before the introduction of CUDA). An earlier work by Amada et al. [6] used the GPU only for the force computation, while administrative tasks such as neighbor search was done on the CPU, requiring many two host-card memory transfers at each iteration.

Both Kolb and Kuntz and Harada used OpenGL and Cg (C for graphics) to program the GPU, requiring knowledge of computer graphics and also some tricks to work around the restrictions imposed by these languages. In particular, since OpenGL doesn't offer the possibility to write to three-

Influence of chemical reactions on gravitational fragmentation of gas-dust disc

Olga P. Stoyanovskaya, Valery N. Snytnikov
Boreskov Institute of Catalysis SB RAS
Novosibirsk State University
Novosibirsk, Russia
stop@catalysis.ru

***Abstract*—Jeans instability in two-phase protoplanetary discs is considered as a key mechanism for coarse bodies' formation (Snytnikov, 2008). The influence of gas and solid bodies subsystems on the disc dynamics was investigated. We developed PIC-SPH code Sombrero in order to carry out the serious of numerical experiments. It was shown that gas temperature had a defining influence on the disc dynamics. Local Jeans length of a massive component being small enough is a necessary condition for the disc fragmentation. For the disc with mass ratio $\frac{M_{par}}{M_{gas}} = \frac{1}{10} - \frac{1}{3}$ it was shown that unstable dust component could initiate large-scale density perturbation and disc fragmentation, even if the gas component was initially stable. Exothermic reactions can influence two-phase dynamics by means of increasing disc's stability to gravitational growth of small density perturbation.**

I. Introduction

The aim of numerical modelling we want to present here is to study physical-chemistry conditions in protoplanetary disc on the stage of coarse bodies' formation. Possible mechanism for the coarse bodies' formation is fragmentation due to gravitational instability in two-phase (gas-solid bodies) medium.

The scale of overdensities that can grow in the self-gravitating medium is characterized by Λ_{Jeans} - local Jeans wavelength.

In this paper we focused on the following questions:

- Can the estimation of local Jeans length made on initial state of the disc provide a forecast for the system dynamics?
- Is gravitational instability of one of the components or both components a necessary condition for the large-scale density perturbation and the disc fragmentation?
- Can chemical reactions affect the disc dynamics?

II. Disc model and code Sombrero

A. Basic equations

Disc thickness is much shorter than its radius. Thus the dynamics of gas and solid particles rotating around the central body can be considered in the equatorial plane only. So we used surface characteristics defined from volume ones by vertical integration:

$$\sigma_{par,gas} = \int_{-\infty}^{+\infty} \rho_{par,gas} dz; \quad A^* = \int_{-\infty}^{+\infty} A dz.$$

Chemically reactive gas component is described by gas dynamics equations completed with chemical kinetics subsystem:

$$\frac{\partial \sigma}{\partial t} + div(\sigma \overrightarrow{v}) = 0,$$

$$\frac{\partial n_i}{\partial t} + div(n_i \overrightarrow{v}) = w_i, \quad i = \overline{1,N},$$

$$\sigma \frac{\partial \overrightarrow{v}}{\partial t} + \sigma(\overrightarrow{v}, \nabla)\overrightarrow{v} = -\nabla p^* - \sigma \nabla \Phi,$$

$$\frac{\partial S^*}{\partial t} + (\overrightarrow{v}, \nabla) S^* = \frac{Q}{\sigma T^*}, \qquad Q = \sum_{i=1}^{N} q_i w_i,$$

$$p^* = T^* \sigma, \qquad \sigma = \sum_{i=1}^{N} \mu_i n_i^*.$$

Here $\overrightarrow{v}$ is gas velocity, T^*, p^* - gas surface density and pressure, $S^* = \ln \frac{T^*}{\sigma^{\gamma-1}}$, N - number of species in reactive gas phase, n_i - surface concentration of i-species, w_i - rate of species syntheses or decomposition, μ_i - molecular mass of i-species.

Dynamics of collisionless solid bodies' phase is described by Vlassov equation:

$$\frac{\partial f}{\partial t} + \overrightarrow{u} \frac{\partial f}{\partial \overrightarrow{r}} + \overrightarrow{a} \frac{\partial f}{\partial \overrightarrow{u}} = 0,$$

where

$$\overrightarrow{a} = -\nabla \Phi,$$

$\overrightarrow{a}$ - particle acceleration due to central body and self-consistent gravitational field, $\overrightarrow{u}$ - particle velocity, $f = f(t, x, y, \overrightarrow{u})$ - particle distribution function with respect to velocity, connected with particle surface density with the expression $\sigma_{par} = \int f d\overrightarrow{u}$.

Φ - gravitational potential,

$$\Phi = \Phi_1 + \Phi_2, \qquad \Phi_1 = -\frac{M_c}{r},$$

M_c - central body mass. Φ_2 - potential of self-consistent gravitational field that is defined as a solution of mixed problem for Laplace equation

$$\Delta \Phi_2 = 0, \quad \Phi \longrightarrow_{r \to \infty} 0, \quad \frac{\partial \Phi_2}{\partial z}|_{z=0} = 2\pi(\sigma_{par} + \sigma_{gas}).$$

SPH Simulation of Pulsatile Flow Inside of a Cavity

Shahrokh Shahriari, Ibrahim Hassan, Lyes Kadem
Department of Mechanical and Industrial Engineering
Concordia University
Montreal, Canada
s_shahri@encs.concordia.ca, kadem@encs.concordia.ca

***Abstract*—In this paper, the results of 2D simulations of pulsatile blood flow inside a simplified left heart cavity with rigid walls using Smoothed Particle Hydrodynamics (SPH) are presented. Such flows are characterized by moderate Reynolds number and a highly pulsatile nature. The simulated velocity patterns and profiles are discussed and compared with the results obtained using finite volume (FV) method. SPH method was able to resolve the main patterns of the flow showing its potential to be applied to complex cardiovascular flows.**

I. INTRODUCTION

Investigation of blood flow behavior in heart cavities is of great interest to biomedical engineering (development of cardiac assist devices) and cardiology (diagnosis of cardiovascular disease). Since the access to in vivo data is sometimes hardly achievable, investigators can only rely on experimental measurements and computational simulations. For this purpose, computational simulations are extremely useful tools since they allow the simulation of a large panel of normal and pathological heart conditions. All previous simulations tried to model the flow inside the heart using grid-based methods, such: immersed boundary method [1] and adaptive finite elements method [2]. However, the most significant difficulty in implementing such traditional computational methods is their mesh dependent nature. Moreover, for clinical applications, these methods generate some data that do not contribute directly to the clinical decision (such as wall shear stress, vorticity fields, …). As an alternative to overcome these limitations and their subsequent effects, in the present work, as the first approach, we attempted to simulate the flow in a very simplified model of left heart cavity using Smoothed Particle Hydrodynamics (SPH) method. Current applications of SPH to simulate unsteady flows are mainly limited to free surface flows [3] or internal low Reynolds number flows [4]. The objective of the present work is to extend SPH applications to moderate Reynolds number pulsatile flows similar to the conditions uncounted in cardiovascular flows.

II. FORMULATION AND IMPLEMENTATION

In this study, blood flow is modelled as an incompressible, Newtonian fluid flow. The assumption of having Newtonian fluid behavior is realistic for blood flow in the heart cavities (since the characteristic length is much higher than the typical size of red blood cells (8 μm)). Therefore, the unsteady governing equations, Navier-Stokes equations, of a Newtonian fluid flow in Lagrangian form are used as,

$$\frac{1}{\rho}\frac{d\rho}{dt} + \vec{\nabla}.\vec{V} = 0 \tag{1}$$

$$\frac{d\vec{V}}{dt} = -\frac{\vec{\nabla}P}{\rho} + \frac{\mu}{\rho}\nabla^2\vec{V} \tag{2}$$

where ρ is fluid density, $\vec{V}$ is velocity vector, P is pressure, μ presents fluid dynamic viscosity and t stands for time. The gravity effect is negligible in the present case. As this is the first attempt to simulate the flow in a simplified model of the left heart cavity using SPH method, we assumed this flow as a laminar flow in 2D coordinate. The assumption of laminar flow is realistic as soon as the left heart is under normal conditions.

The classical SPH, weekly compressible, is used to formulate the governing equations. To determine the fluid density at particle a, the formulation derived based on mass conservation equation is used [4, 5] to ensure the stability in density calculation based on interaction between velocity and pressure changes as,

$$\frac{d\rho_a}{dt} = \sum_b m_b(\vec{V}_a - \vec{V}_b)\vec{\nabla}_a W_{ab} \tag{3}$$

To model the pressure term in momentum equation under SPH formulation, the most commonly symmetric form for the pressure gradient is used as [5],

$$\left(\frac{\vec{\nabla}P}{\rho}\right)_a = -\sum_b m_b\left(\frac{P_b}{\rho_b^2} + \frac{P_a}{\rho_a^2}\right)\vec{\nabla}_a W_{ab} \tag{4}$$

Fluid pressure is derived as a function of density by using a quasi-incompressible equation of state as [4],

$$P = c^2\rho \tag{5}$$

where c is the sound speed. In this simulation because of the flow inherent characteristics (internal, pulsatile, and moderate

SPH Modelling of Viscous and Non-Newtonian Sloshing

Jason P. Hughes
School of Mathematics and Statistics
University of Plymouth
Plymouth, UK
jhughes@plymouth.ac.uk

Christopher Carne
School of Mathematics and Statistics
University of Plymouth
Plymouth, UK

Philip W. James
School of Mathematics and Statistics
University of Plymouth
Plymouth, UK

David I. Graham*
School of Mathematics and Statistics
University of Plymouth
Plymouth, UK
dgraham@plymouth.ac.uk

***Abstract*—Sloshing of highly viscous - and in many cases non-Newtonian - fluids is commonplace in food processing operations. An example is the motion of liquid foodstuffs in containers during filling processes (where seal contamination may be an issue). In order to understand elements of this process more deeply, an experimental study of sloshing of highly viscous (Glycerol) and non-Newtonian fluids (Benecel, Blanose and Keltrol solutions) in a simple rectangular box has been carried out. The results of these experiments are published here for the first time. In this paper, this experiment is modelled using SPH and ANSYS CFX and the results are compared with the experimental data. The investigation has three elements i) how well do the numerical predictions predict the bulk features of the experiments? ii) how well do the details of CFX and SPH compare? and iii) is the non-Newtonian nature of the fluids important for the flows considered?**

I. Introduction

In the food processing industry there is interest in 'clean-filling', a term used to describe the insertion and subsequent sealing of food into containers under conditions which are not aseptic but in which opportunities for product contamination are reduced. There are several factors which can affect the clean-filling process, for example the operation of the filling nozzle itself is particularly important if contamination of the processing environment due to splashing or dripping is to be avoided. Another factor, which provides the motivation for the current work, is the possibility of splashing or sloshing of food that has been placed in a container due to the motion of the container itself. Splashing of foodstuff onto surrounding surfaces degrades the clean environment and if there are food deposits on container surfaces prior to sealing then product contamination can occur. The types of products that can be clean-filled include milk, fruit juices, soups and sauces and some of these materials exhibit shear-dependent viscosities. Such materials often have high viscosities at low shear rates and may be characterised for the purposes of modelling as inelastic non-Newtonian liquids.

In order to understand aspects of this process, an experimental and numerical study has been made of the sloshing motion of highly viscous Newtonian and shear-thinning inelastic liquids. Simulations are carried out using both the commercial CFD software ANSYS CFX [1] and an SPH method. The results from the simulations are compared with each other and with the experimental data. Section II of this paper describes the experiments, whilst Section III reports the CFX and SPH methods and results. Conclusions are drawn in Section IV.

II. Experimental Work

A. Experimental rig

An experimental rig, shown in Figure 1, was constructed in order to obtain measurements of the displacement of the free surface of fluid in a rectangular container subjected to an acceleration and deceleration, such as might occur on a conveyor belt used to move filled food containers (Grundelius and Bernhardsson [4]).

Figure 1. Experimental rig.

The moving part of the rig comprises a very heavy steel platform attached to two steel blocks that slide along a pair of steel cylinders. Bearings between the blocks and cylinders allow the platform to slide freely. The platform is set into motion by means of air pistons attached to the rear of the

SPH model for viscoelastic fluids with thermal fluctuations

Adolfo Vázquez-Quesada
Departamento de Física Fundamental
UNED
Madrid, Spain
avazquez@bec.uned.es

Marco Ellero
Lehrstuhl für Aerodynamik
Technische Universität München
Garching, Germany
marco.ellero@aer.mw.tum.de

Pep Español
Freiburg Research Institute for Advanced Studies
University of Freiburg
Freiburg, Germany
pep@fisfun.uned.es

***Abstract*—We present a fluid particle model of viscoelastic fluid. Using the GENERIC framework we can introduce the thermal fluctuations in a natural way. In the simplest case of Hookean dumbbells the dynamic equations obtained are a discretization of the Oldroyd-B model. Generalization to more complex viscoelastic models is straightforward.**

I. Introduction

A very useful technique for the study of complex fluids properties is microrheology. Microrheology is an experimental technique which allows to obtain rheologic properties of viscoelastic fluids by optically procedures. In these experiments, the movement of a colloid particle inside a fluid is observed, and its viscoelastic properties are inferred through a generalized Stokes-Einstein relation [1], [2]. Microrheology has comparative advantages regarding traditional rheology: it allows one to use tiny quantities of fluids (order of picoliter), the measurements are local and higher oscillation frequencies can be studied. These advantages allow to study the rheological behavior of biological fluids and living cells [3], [4].

There are open questions about this technique, for example, the validity of the generalized Stokes-Einstein equation [5], or the influence of other colloidal particles in the measurements of a single colloidal particle. Simulations of viscoelastic fluids with colloidal particles immersed in them may give us insight and further support to this relation. Moreover, the generalized Stokes-Einstein relation assumes the fluid to behave in the linear viscoelastic regime, and it may be interesting to probe the non-linear regime also. Another interest thing to study would be the effect of the viscoelastic hydrodynamic interactions of the container, or of another colloidal particles on the measurement of a single colloidal particle.

To adress these questions it would be very useful to have a simulation technique capable to simulate the situations we could find in the microrheologic systems. In the present work, we formulate a coarse grained fluid particle model as a first step in that direction. The state of the polymer molecules within a fluid particle is represented by a conformational tensor [6]. The conformation tensor may be thought of as the second moment of the end to end distance distribution function. The GENERIC formalism [7], [8] has already been used with SPH [9], and allows us to construct a model thermodynamically consistent model. Moreover, using GENERIC we can introduce thermal fluctuations in a natural way.

II. The non-isothermal particle model for dilute polymer solutions

We model a polymeric solution through a collection of M *fluid particles* with positions $\mathbf{r}_i$ and velocities $\mathbf{v}_i$. The fluid particle labeled i contains N_i^p polymer molecules. We will characterize the elastic behavior of a fluid particle with a dimensionless conformation tensor defined by

$$\mathbf{c}_i = \frac{1}{N^p q_0^2} \sum_a^{N^p} \mathbf{q}_a \mathbf{q}_a \tag{1}$$

where $\mathbf{q}_a$ is the end to end distance of the a-th polymer molecules within the fluid particle i. The tensor has been normalized with q_0 in such a way that the equilibrium value of the conformation tensor is $\mathbf{c}_i = \mathbf{1}$.

For simplicity, the polymer molecules will be represented by dumbbells. The positions of the first and second beads of dumbbell a are $\mathbf{r}_a^1$ and $\mathbf{r}_a^2$, respectively. In the overdamped and dilute limits, its evolution is given by the Langevin equations

$$\begin{aligned} d\mathbf{r}_a^1 &= \mathbf{V}(\mathbf{r}_a^1)dt + \frac{1}{\gamma}\mathbf{F}(\mathbf{r}_a^1 - \mathbf{r}_a^2)dt + (2D_0)^{1/2}\, d\mathbf{W}_a^1 \\ d\mathbf{r}_a^2 &= \mathbf{V}(\mathbf{r}_a^2)dt + \frac{1}{\gamma}\mathbf{F}(\mathbf{r}_a^2 - \mathbf{r}_a^1)dt + (2D_0)^{1/2}\, d\mathbf{W}_a^2 \end{aligned} \tag{2}$$

Here, $\mathbf{V}(\mathbf{r})$ is the flow velocity field. We assume that *within* the fluid particle we have a homogeneous flow field given by $\mathbf{V}(\mathbf{r}) = \mathbf{v}_i + \kappa_i \cdot \mathbf{r}$, where $\mathbf{v}_i$ is the velocity of the fluid particle and $\kappa_i = (\nabla \mathbf{v})_i$ is its velocity gradient tensor. The force due to the spring connecting the two beads is $\mathbf{F}(\mathbf{r}_a^2 - \mathbf{r}_a^1)$. The friction coefficient is $\gamma = 6\pi\eta a$, with a the bead radius and η the solvent shear viscosity. The stochastic forces are proportional to the diffusion coefficient D_0 of the beads, given by the Stokes-Einstein relation $D_0 = k_B T/\gamma$, while $d\mathbf{W}$ is a vector independent increment of the Wiener process.

By changing to center of mass $\mathbf{R}_a = (\mathbf{r}_a^1 + \mathbf{r}_a^2)/2$ and

Modelling impact on aerospace structures using the coupled FE-SPH approach

R. Vignjevic and J.C. Campbell
Crashworthiness, Impact and Structural Mechanics Group,
School of Engineering, Cranfield University,
Bedfordshire, MK43 0AL. UK.
v.rade@cranfield.ac.uk

J.M. Reed
Rolls-Royce Plc,
Derby
DE24 8BJ. UK

***Abstract*— Aircraft structures must be designed to withstand a range of impacts. Impact threats include birds, hail ice and tyre fragments. Bird-strike and hail ice impact can be considered as fluid-structure impacts, due to the behaviour and level of deformation seen in the impactor. Simulation is increasingly important in the design of aerospace structures against these threats as experiments are complex and expensive. Due to the well known problems with mesh-tangling the FE method is not well suited for modelling the fluid and the method much be coupled with another method, such as SPH. Previous work at Cranfield has developed and demonstrated an algorithm for coupling meshless and finite element discretisations, allowing complex interaction in three dimensional simulations. This paper discusses the coupled FE/SPH approach, and presents simulation results from two studies of bird strike on jet engine fan blades.**

I. INTRODUCTION

Aircraft structures must be designed to withstand a range of impact conditions, with threats including birds, hail ice, tyre fragments as well as crash and ditching. In addition to ditching, bird-strike and hail ice impact can be considered as fluid-structure impacts, due to the behaviour and level of deformation seen in the impactor. Advanced numerical simulation plays an increasingly important role in the design of aerospace structures against these threats, providing a cost effective method to estimate impact damage and investigate the effect of design changes.

The objective of this research is to develop and demonstrate an analysis tool capable of predicting the response of aerospace structures to impact and crash loading. To achieve this objective the tool must be capable of representing the important feature of the problem. The explicit finite element (FE) method is established as the primary tool for the simulation of structural response and is well suited to representing the types of structural deformation that can occur. However due to the well known problems with mesh distortion this method is often not appropriate for modelling the impactor. Correct behaviour of the impactor is critical for accurate prediction of the structural loads. In order to provide the appropriate loading, the FE method must be coupled with another method appropriate for modelling the fluid-like behaviour of the impactor.

Previous work at Cranfield developed an algorithm for coupling the SPH and FE methods. This algorithm has been implemented in a code that combines the 3D SPH solver developed at Cranfield [1-4] and the explicit finite element code LLNL-DYNA3D [5] and been successfully applied to analysis of helicopter crash on water [6]. This paper discusses the coupled FE/SPH approach, and presents two examples of its application to bird strike on jet engine fan blades.

Bird strike represents a major threat to aircraft safety due to the serious potential consequences. In addition it is estimated that aircraft collisions with wildlife costs the civil aviation industry in the USA at least $500 million annually [7], with the majority of reported collisions being bird strikes The problem of bird strike has been around since the earliest flights, with the first recorded bird strike described by the Wright brothers in 1905. Today bird strike is a requirement in the certification of aircraft and jet engines and hence is an important design case.

II. FE-SPH COUPLING

A consequence of the SPH method is that the location of the boundary of a solid or liquid is not well defined. Each particle is the centre of a sub-domain with radius 2h and consequently the boundary is diffused over this length. To avoid the requirement to construct a surface geometry from the SPH nodes a contact potential, ϕ_x, based on the kernel function is used [2,3]. This potential is defined as

$$\phi_x(\mathbf{x}_i) = \int_{\Omega_c} K\left(\frac{W(\mathbf{x}_i - \mathbf{x}_j, h)}{W(\Delta p_{avg})}\right)^n dV \qquad (1)$$

where $\mathbf{x}_i$ and $\mathbf{x}_j$ are the coordinates of two particles in different bodies, Δp_{avg} denotes the average node spacing in the neighbourhood of particle i. K and n are user defined parameters that control the magnitude of the potential. A discrete form of this contact potential is then derived and used to define the contact force, $\mathbf{f}_i$:

$$\mathbf{f}_i = \frac{m_i}{\rho_i}\sum_j \frac{m_j}{\rho_j} Kn \frac{W(\mathbf{x}_i - \mathbf{x}_j, h)^{n-1}}{W(\Delta p_{avg})^n} \nabla W(\mathbf{x}_i - \mathbf{x}_j, h) \qquad (2)$$

In applying this contact potential to contact between FE and SPH, the contact force is applied between an SPH particle and FE node that lies within 2h of that particle [3], see fig. 1.

High speed machining modelling: SPH method capabilities

Jérôme LIMIDO
IMPETUS Afea
Grenade sur Garonne, France
Jerome@impetus-afea.com

Christine ESPINOSA, Michel SALAÜN, Catherine MABRU and Rémy CHIERAGATTI
Université de Toulouse, ISAE
Toulouse, France

***Abstract*— The purpose of this work is to evaluate the use of the Smoothed Particle Hydrodynamics (SPH) method within the framework of high speed cutting modelling. First, a 2D SPH based model is carried out using the LS-DYNA® software. SPH is a meshless method, thus large material distortions that occur in the cutting problem are easily managed and SPH contact control allows a "natural" workpiece/chip separation. The developed SPH model proves its ability to account for continuous and shear localized chip formation and also correctly estimates the cutting forces, as illustrated in some orthogonal cutting examples. Then, The SPH model is used in order to improve the general understanding of machining with worn tools. At last, a milling model allowing the calculation of the 3D cutting forces is presented. The interest of the suggested approach is to be freed from classically needed machining tests: Those are replaced by 2D numerical tests using the SPH model. The developed approach proved its ability to model the 3D cutting forces in ball end milling.**

I. INTRODUCTION

Machining is the most used process in industrial components production. The development of accurate and reliable machining process models has received considerable effort in the past years, but physical phenomena involved in industrial cutting cases are fully 3D and very complex. Thus, the orthogonal cutting framework is generally used in order to allow a 2D plain strain study. Orthogonal cutting conditions are reached when the cutting edge is rectilinear, perpendicular to the chip flow in each point, and perpendicular to the tool feed. In these conditions, the cutting parameters are summarized by cutting speed (Vc) and feed (f) (see figure 1).

In orthogonal cutting conditions and in stationary regime, the tool/workpiece interaction and the chip generation process can be represented in a simple way. Three principal shearing zones appear, see figure1: the primary zone (shearing causing the chip formation), the secondary zone (shearing due to tool/chip friction), and the tertiary zone (shearing due to tool/generated surface friction).

For years, significant effort has been devoted to the development of computational models of high-speed machining in order to overcome analytical models limits. Most of the machining numerical models are based on Lagrangian or Arbitrary Lagrangian Eulerian (ALE) Finite Element Methods (FEM) [3, 13].These approaches imply two major difficulties. First of all, the friction model must account for all the tribological complexity of machining. In most cases, the Coulomb model is used; it is very simple but limited [3]. Moreover, the friction parameter is often used in order to readjust the cutting forces obtained by FEM compared to experimental results [3, 13]. The workpiece/chip material separation model is the second aspect of the modelling difficulties. Lagrangian FE methods present the disadvantage of leading to large grid distortions. This implies the use of ALE methods and time consuming remeshing techniques.

Here, we present an approach based on the Smoothed Particle Hydrodynamics (SPH) method in the frame of the LS-DYNA® hydrodynamic software [12].

The paper is organized as follows. In section 2, our 2D SPH cutting model is introduced. In section 3, SPH cutting model applications are outlined and compared with other numerical or experimental data. The use of SPH as a numerical tool for a better understanding of the chip formation with worn tools is also presented. Finally, a developed 3D milling model based on 2D SPH model results is presented.

II. SPH CUTTING MODEL

A. Basic principles of the SPH method

SPH method is a meshless Lagrangian technique. Material properties and state variables are approximated by their values on a set of discrete points, or SPH particles. This avoids the severe problems of mesh tangling and distortion which usually occur in Lagrangian analyses involving large deformation. For more details on the method used, the reader can refer to [12].

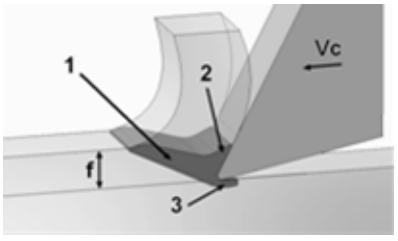


Figure 1. Shear zones definition

On the concept of a hybrid particle method to model coastal bluff collapse during extreme events

Johan Vandamme, Qingping Zou, Dominic Reeve
Centre for Coastal Dynamics and Engineering
School of Engineering, University of Plymouth
Plymouth, Devon, UK
johan.vandamme@plymouth.ac.uk

This paper investigates the concept of modelling cliff stability and collapse during an extreme erosion event using a hybrid model method combining Weakly Compressible Smoothed Particle Hydrodynamics (WCSPH) with a particle based geotechnical stability analysis (GeoSPH). This problem is of particular interest for both as a short term erosion and collapse prediction tool and also as a method to increase the accuracy and confidence in long term predictive models by increasing the understanding of individual collapse events. In particular, this paper investigates a displacement based multiple wedge stability analysis of soils using the particle method, and the overall hybrid model concept. Conventional predictions of bluff collapse are often probabilistic, and struggle to include changes to the predictions when changes occur to the wave climate, bluff makeup, or other implicated factors. Using a hybrid model will allow for site-by-site analysis and does not require data of past events to predict towards forward analysis. The groundwork of the model is laid out within the paper, with the preliminary proof-of-concept results presented and discussed, and the future development and direction examined.

I. INTRODUCTION

The recent prominence in local and national media regarding coastal erosion [1-7] has combined with the increased awareness and development of climate change predictions to add interest in both modelling the expected erosion of the shoreline and cliffs, and in achieving a better understanding of how to stabilise and protect the areas at risk, when such intervention is deemed appropriate. It is imperative for each of these goals that we better understand the mechanisms and the factors effecting coastal bluff erosion and collapse in a short timescale.

Much of the current research is focused on long term retreat of a cliff or coastal bluff, where predictions are formed by extrapolating data from old maps, surveys, and, increasing in prominence although still used rarely, satellite imagery. These methods are limited in their long term accuracy both by the underlying assumption of consistent average retreat, and also by treating the individual cliff fall as a continuous process when considering a long timescale. This approach is taken partly due to the lack of understanding and modelling in the individual cliff falls [8], but this does result in cumulative errors. In addition to this, the extrapolation of past measurements does not incorporate changes of the site parameters, from sea level rise to engineering works, which limits the usefulness of the models.

The area of short term erosion models is still in its relative infancy. Many of these models are closed loop models that assume a profile will, if exposed to the same conditions for a long enough period of time, tend towards an equilibrium profile. SBEACH [9, 10] is a two dimensional cross-shore model based on the principle that the dissipation of wave energy per unit volume of water is the significant destructive force. It incorporates wave run-up and set-up and gives a mostly accurate (although marginally under-predicted [11]) dune profile. SBEACH is similar in results and methods to EDUNE, developed by Kriebel [12, 13]. EDUNE allows for constant wave run-up but does not incorporate set-up nor variable sediment properties.

Other methods include the Coastal Construction Control Line (CCCL) [14, 15] which models uniform sediment and tends to over-predict results [11], and has not been developed significantly further in the research literature.

II. MODEL DETAILS

A. General model details

The well documented development of Smoothed Particle Hydrodynamics (SPH) from its astrophysics roots [16] to the extensive modelling of free surface fluid flows and breaking waves [17-19] will not be dwelt on in detail, as further explanation is well documented in [20] and also in [21].

The authors have used the SPH that has been parallelised by Zou [22] as this allows for faster runs on a cluster of high performance computers. This code is based on the principles of weakly compressible SPH, and was previously developed at Johns Hopkins University.

Smoothed Dissipative Particle Dynamics model for colloidal particles in suspension

Xin Bian
Lehrstuhl für Aerodynamik
Technische Universität München
85748 Garching by Munich, Germany
xin.bian@aer.mw.tum.de

Marco Ellero
[1]Lehrstuhl für Aerodynamik
Technische Universität München
85748 Garching by Munich, Germany
[2]Departamento de Física Fundamental
UNED
Apartado 60141, 28080 Madrid, Spain
marco.ellero@aer.mw.tum.de

***Abstract*—We describe a model of colloidal suspension based on Smoothed Dissipative Particle Dynamics (SDPD). The method has been developed first by Español and Revenga in 2003 [1] and can be understood as a generalization of Smoothed Particle Hydrodynamics (SPH) with the proper introduction of thermal fluctuations. The Lagrangian and meshless features typical of SPH together with the ability to model Brownian motion makes SDPD a suitable tool to describe complex microstructured fluids at the mesoscopic scales.**

The solvent is modelled by considering a set of SDPD particles defined on the fluid domain. The colloidal particle is represented by a set of "boundary particles" and modelled as rigid body moving through the fluid under the action of a total drag force exerted by the surrounding medium. Colloidal velocity and position are therefore updated at the end of each time step by using the same time-integrator used for the fluid particles. In order to enforce no-slip boundary conditions at the moving interface, the method of Morris et al. is extended [2] by assigning an artificial velocity to each of the boundary particles in order to have a prescribed interpolated velocity on the nominal solid-fluid interface.

In order to test the model, the validation case of a flow through a periodic array of disks/spheres for several concentrations is initially considered.

I. Introduction

The suspension of colloidal particles in a solvent medium represents a problem of large scientific and technical interest, with examples in biological systems (blood), home products (paint), and industrial processing (waste slurries) [3]. At length scale of microfluidics, the Navier-Stokes equations describing the dynamics of a Newtonian fluid at the macroscopic level still remain valid, therefore providing a natural framework based on the continuum description. On the other hand, when colloidal particles are in the sub-micrometer range, the surrounding fluid starts to feel the presence of its underlying molecular structure and hydrodynamics variables will be influenced by thermodynamics fluctuations according to the Landau and Lifshitz theory [4].

The physical understanding of suspension rheology and more specifically how the microstructure affects macroscopic liquid properties represents a very challenging and in many cases unresolved problem. In particular, the rheology of a concentrated dispersion of particles with possibly arbitrary shapes, suspended in a Newtonian or non-Newtonian media has not been yet sufficiently investigated.

A key ingredient in the simulation of colloids is represented by the presence of *thermal motion*. The lack of its appropriate modelling in many numerical methods makes it impossible, for example, to achieve a "random" final state with the consequence that all the results inevitably depend on the specific choice of the initial configuration [5]. The introduction of thermal motion in the simulation of colloids is also crucial for description of novel micro-rheology experiments where the rheological properties of complex viscoelastic media are deduced by the mean square displacement or by the rotational diffusion of colloidal probes [6].

A model of colloidal suspension based on Smoothed Dissipative Particle Dynamics (SDPD) is presented. SDPD has been developed first by Español and Revenga in 2003 [1] and can be understood as a generalization of Smoothed Particle Hydrodynamics (SPH) with the proper introduction of thermal fluctuations. It avoids the drawbacks of another popular mesoscopic method, Dissipative Particle Dynamics (DPD), i.e. (i) mapping and calibrating DPD model parameters to the physical parameters are not always systematic, and it is difficult to specify in advance the scale at which a DPD simulation operates. This problem is crucial, for instance, in the case of suspended colloidal particles where the physical dimensions of the external objects determine whether and, more importantly, to which extent thermal fluctuations come into play [7]; (ii) in DPD the equation of state is forced to be a quadratic function of the density [8]; (iii) the transport coefficients can not be specified a priori and must be settled by kinetic theory or preliminary runs [7] [9]. The SDPD model has a structure very similar to both SPH and DPD and carry the best of the two models [1], i.e. Lagrangian and meshless features typical of SPH together with the ability to model Brownian motion associated to DPD. These features make it a suitable tool to study complex microstructured fluids at the mesoscopic scales.

The colloidal particle is modelled as rigid body moving

Towards a resolution analysis in mesoscopic fluid flows using Smoothed Dissipative Particle Dynamics

Adolfo Vázquez-Quesada
Departamento de Física Fundamental
UNED
28080 Madrid, Spain

Marco Ellero
Lehrstuhl für Aerodynamik
Technische Universität München
85747 Garching, Germany
marco.ellero@aer.mw.tum.de

Pep Español
Departamento de Física Fundamental
UNED
28080 Madrid, Spain

***Abstract*—Smoothed Dissipative Particle Dynamics (SDPD) is a particle method recently proposed by Español et al. to simulate mesoscopic flow problems. SDPD is a version of the well-known Smoothed Particle Hydrodynamics (SPH) method, albeit with the proper inclusion of thermal fluctuations. Whereas SPH is a scale-free method, namely it produces convergent dynamics for decreasing dimension of the fluid particles, the same is not true for SDPD. Here the fluid particle sizes determine the magnitude of the thermal fluctuations associated to hydrodynamic variables and therefore different local dynamics for the SDPD particles are obtained at different coarse-graining levels. Nevertheless, even though the dynamics of the fluid particles does depend on the degree of coarse-graining, physical properties should not. Here, we show that SDPD produces the proper scaling of the fluctuations as the resolution of the simulation is varied leaving on the other side physical quantities unchanged. This is investigated in two problems: the Brownian motion of a spherical colloidal particle and a polymer molecule in suspension.**

I. Introduction

The rapid increase in the number of applications in the area of nano-micro fluid dynamics is triggering the development of new class of particle methods able to describe the flow physics occurring at spatio-temporal scales smaller than micro meters/seconds. The Brownian diffusive processes arising at such small scales requires the introduction of thermal fluctuations in the fluid variables according, for example, to the Landau&Lifshitz theory.

A particle method designed for the description of mesoscopic flows is Smoothed Dissipative Particle Dynamics (SDPD) originally developed by Español et al. in 2003 [1]. In SDPD, the fluid is represented by a collection of particles interacting with prescribed conservative, dissipative and stochastic forces. Unlike other mesoscopic particle methods, SDPD can be viewed as formally multiscale as it represents indeed a version of the well-known 'macroscopic' Smoothed Particle Hydrodynamics method (SPH) [2], [3], albeit with the proper inclusion of thermal fluctuations. The remarkable feature of the model is that the magnitude of the thermal fluctuations depends on the typical length size of the fluid particles scaled as the inverse of its square root, in accordance with usual concepts of equilibrium statistical mechanics. Therefore, for large enough fluid particles, the thermal fluctuations in the momentum and energy equation can be neglected and a suitable SPH formulation is recovered. On the other hand, if we want for example to simulate a sub-micron sized colloidal particle in a microfluidics environment we will need to resolve the solvent liquid with fluid particles of at least one order smaller than the diameter of the colloid, which will produce non-vanishing stochastic terms giving rise to its ultimate Brownian diffusive motion.

In this talk, we address the *issue of resolution* in mesoscopic flow problems [10]. In fact, one would expect that physical properties of the system should not depend on the coarse-graining level at which one operates. The issue of resolution in SPH is standard, because the model is a discretization of the Navier-Stokes equations that converges towards the continuum as the number of particles is increased. It remains to explore the nature of the change of scale in the SDPD model, that includes thermal fluctuations.

II. Smoothed Dissipative Particle Dynamics

The GENERIC framework [4] allows one to easily formulate a thermodynamically consistent SPH model, in a form which encodes automatically the First and Second Laws of Thermodynamics. In particular, it allows to introduce thermal fluctuations in a systematic way, which by construction satisfies the fluctuation-dissipation theorem. The equilibrium distribution function of the independent variables is given by Einstein's equilibrium distribution. The SDPD dynamic equations are given in Ref. [1]. Let us consider first the deterministic part which read

$$\begin{aligned}
\dot{\mathbf{r}}_i &= \mathbf{v}_i \\
m\dot{\mathbf{v}}_i &= \sum_j \left[\frac{P_i}{d_i^2}+\frac{P_j}{d_j^2}\right] F_{ij}\mathbf{r}_{ij} \\
&- \frac{5\eta}{3}\sum_j \frac{F_{ij}}{d_i d_j}\left[\mathbf{v}_{ij}+(\mathbf{v}_{ij}\cdot\mathbf{e}_{ij})\mathbf{e}_{ij}\right] \\
T_i\dot{S}_i &= \frac{5\eta}{6}\sum_j \frac{F_{ij}}{d_i d_j}\left[\mathbf{v}_{ij}^2+(\mathbf{e}_{ij}\cdot\mathbf{v}_{ij})^2\right]
\end{aligned}$$

Simulating rapid sediment scour by water flow with SPH

S. Manenti, A. Di Monaco, M. Gallati, S. Sibilla
Dipartimento di Ingegneria Idraulica e Ambientale
Università di Pavia
Pavia, Italy

G. Agate, R. Guandalini, A. Maffio
Environment and Sustainable Development Dept.
CESI RICERCA S.p.A.
Milan, Italy

***Abstract*— The present paper deals with the SPH numerical modelling of the interaction between fluid and non-cohesive bed sediments whose scour is induced by a rapid water outflow from the bottom of an experimental reservoir. The dynamic behaviour of the liquid and granular phases is simulated by following two alternative approaches: either by treating them as slightly compressible fluids whose motion results from the numerical solution of the continuity and momentum equations according to the standard SPH formulation; or by describing the evolution of the interface between the two phases (i.e. the scouring process) as a transport-diffusion phenomenon. The main features of the model are presented first and then the numerical results for representative experimental test cases are discussed and compared with available experimental data.**

I. INTRODUCTION

During the operation of an hydropower reservoir the bottom outlet is periodically opened for safety purposes: actually, the induced flushing removes part of the nearest bed sediments, thus preventing the outlet from obstruction. A reliable tool that can predict the fluid-solid coupled dynamics is required for quantifying such phenomenon and for evaluating possible malfunctioning due to clogging of the outlet and changes in the static loads of the dam.

The potential of the SPH method for such kind of investigation was shown in a previous work [4], where a 2D model simulated local sediment scour and flushing hydrodynamics induced by a rapidly varied flow in an experimental reservoir. A comparison between the numerical results and measured laboratory data proved that the SPH model can reproduce most of the relevant physical and engineering aspects of such a phenomenon.

In the present paper the features of an improved SPH model are shown and the numerical results of some representative 2D and 3D experimental test cases are discussed. Sediment scour is treated by following two alternative approaches.

In the first approach the liquid and the granular phases are treated as slightly compressible fluids, whose motion results from the numerical integration of continuity and momentum equations in Lagrangian form by adopting the classical SPH approach. The peculiar behaviour of the granular phases is taken into account first by considering the sediment fixed until its viscosity (calculated according to the Mohr-Coulomb yield criterion and the effective pressure in the porous medium) falls below a critical threshold which determines the onset of the sediment motion and then by treating it as a pseudo-fluid.

In the second approach scouring is modelled as an interface problem according to [2], who extended to the SPH method the Eulerian formulation proposed by [1]: media are simulated by assuming a proper initial volume fraction ϕ of the *solid phase* (e.g. ϕ=0 for the water particles and ϕ=1 for the sediments), the volume fraction of the *liquid phase* being equal to ϕ_w=1-ϕ. The erosion process is then simulated as a transport-diffusion phenomenon that leads to a change of the volume fraction of each particle in accordance with the concentration gradient.

The paper illustrates first a description of the numerical model; its results are then discussed and compared with available experimental data; finally the main conclusions are given.

II. NUMERICAL MODEL

In the following, a concise description of the principal features of the model is provided.

A. Fluid

The water and the sediments eroded from the bed are considered as a slightly compressible fluids whose motion is predicted by integrating in time the Navier-Stokes equations [7] through a first order explicit scheme in which the evaluation of the velocity is staggered by half time step from the evaluation of particle position, density and pressure [5].

When considering a problem whose evolution over time is longer with respect to the time-scale of the smoothing process, high pressure values arise near the free surface [8]; in order to avoid unphysical results, pressure smoothing is performed as follows:

$$p_i^{smth} = p_i + \vartheta_p \cdot \left[\frac{\sum_{j=1}^{Ni} p_j \cdot \Delta V_j \cdot W_{ij}}{\sum_{j=1}^{Ni} \Delta V_j \cdot W_{ij}} + \rho_0 g \cdot \frac{\sum_{j=1}^{Ni} \Delta V_j \cdot (z_j - z_i) \cdot W_{ij}}{\sum_{j=1}^{Ni} \Delta V_j \cdot W_{ij}} \right] \quad (1)$$

Normal flux method at the boundary for SPH

De Leffe Matthieu[1,2]
[1] HydrOcean
Nantes, France
matthieu.de-leffe@hydrocean.fr

D. Le Touzé and B. Alessandrini
[2] Fluid Mechanics Laboratory (CNRS UMR 6598)
Ecole Centrale Nantes
Nantes, France
david.letouze@ec-nantes.fr

***Abstract*— An upwind scheme for SPH in an ALE context is proposed, based on a Finite Volume with Characteristic Flux method. From this reformulation of the SPH model within the fluid, a new technique is proposed to handle different kinds of boundary conditions (inflow, outflow, non-reflecting, free slip, etc.). It is validated on one- and two-dimensional problems, against solutions provided using the more classical ghost technique to handle boundaries. The results exhibit the accuracy and the flexibility of the proposed method compared to existing ones.**

I. INTRODUCTION

The work from Vila [3] with the Arbitrary Lagrange Euler (ALE) description allows reusing finite volume techniques in the Smoothed Particle Hydrodynamics (SPH) method context. In the paper, a new scheme to provide a formula for the numerical flux exchange between two particles is provided. This scheme, FVCF (Finite Volume with Characteristic Flux scheme) is a basic upwind flux scheme from Ghidaglia and Pascal [1]. The same MUSCL technique as for the Godunov scheme permits to reduce numerical diffusion. Comparisons with a Godunov scheme on shock tubes show that the both methods exhibit similar efficiency. However, the FVCF has a lower cost and is more robust.

Then, this scheme is extended to propose a new boundary treatment. The FVCF method is particularly well suited to this purpose in the sense that it computes directly the normal flux at the boundary. Firstly, a treatment for inflow and outflow boundary conditions as well as a non-reflecting boundary condition is presented. Secondly, the boundary treatment proposed by Marongiu et al. [2] for solid boundaries is adapted to the proposed scheme, and improved. The resulting scheme allows using complex geometry with angles; which is very difficult to do with the classical ghost particle method.

Finally, a number of validation tests are performed. The inlet and outlet boundary condition are validated in one dimension. The solid boundary treatment is then validated on various cases such as hydrostatics around a complex geometry, a dam break or a wedge entry case, with comparison against solutions obtained by a classical ghost particle technique. The latter comparisons permit to highlight the accuracy and the flexibility of the proposed method with respect to existing techniques. The proposed method allows for instance to handle complex geometries with sharp angles with no specific treatment.

II. DISCRETISATION

The numerical scheme we use relies on the one proposed by Vila [3] which reads:

$$\begin{cases} \dfrac{dx_i}{dt} = v^0{}_i \\ \dfrac{dw_i}{dt} = w_i \sum_{j\in P} w_j (v^0{}_j - v^0{}_i).\nabla W_{ij} \\ \dfrac{dw_i\phi_i}{dt} + w_i \sum_{j\in P} w_j (F_E(\phi_i) - v^0{}_i\phi_i + F_E(\phi_j) - v^0{}_j\phi_{ji}).\nabla W_{ij} = S \end{cases}$$

where x_i, v_i, p_i, w_i are respectively, the position, velocity, pressure and volume of the considered particle. $v^0{}_i$ is the transport field of the ALE approach. ϕ_i is the vector of conservative variables $(\rho_i, \rho_i v_i)^t$. $F_E(\phi_i)$ is the flux of conservative variables $(\rho_i v_i, \rho_i v_i \otimes v_i + p_i I_d)^t$. ∇W_{ij} is the gradient of the kernel (here a cubic spline on a compact support of width 2h, h being the smoothing length). The system of equations is closed through the Tait state equation:

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

where ρ_0 is the reference density, c_0 a numerical speed of sound, and γ a constant coefficient.

This scheme is centred in space. When using an explicit time scheme as we do, such a space scheme is unconditionally unstable. A classical remedy is to introduce the artificial viscosity of Monaghan. Another solution is to use Riemann solvers, for example the Godunov numerical scheme as described by Vila [3]. In this paper, we focus on a third solution based on the Finite Volume with Characteristic Flux scheme (FVCF), introduced by Ghidaglia and Pascal [1].

III. THE FVCF SCHEME

The FVCF scheme is an alternative to Riemann solvers. This method relies on an upwind scheme of the linearized

A 2D+t SPH model with enhanced solid boundary treatment

A. Colagrossi
CESOS: Centre of Excellence for Ship and Ocean Structures, NTNU, Trondheim, Norway,
The Italian Ship Model Basin (INSEAN), US3, Rome, Italy.
Corrisponding author: a.colagrossi@insean.it

M. Antuono, S. Marrone
The Italian Ship Model Basin (INSEAN), US3, Rome, Italy.

***Abstract*—A $2D+$t approach is applied to study the wave pattern generated by high speed slender ships with a sharp stem. This allows approximating the ship motion through a set of equations which are mathematically equivalent to those governing the unsteady two-dimensional free-surface flow generated by a deformable body in the vertical plane transverse to the ship. To describe the body deformation, a proper description of the solid boundaries is needed and, therefore, an enhanced treatment of the moving solid boundaries is proposed within the two-dimensional SPH scheme. In the specific, the moving solid boundary is modeled through boundary particles. Differently from the ghost particles, the boundary particles are fixed with respect to the solid boundary and are associated to interpolation nodes internal to the fluid through which they take the flow properties. Finally, a set of $2D+$t numerical simulations is shown which proves that the numerical scheme based on the use of the boundary particles is robust and accurate.**

I. Introduction

Ship-generated waves have always fascinated scientists, and played a key role in surface-ship hydrodynamics contributing to hull resistance [1], generating noise [2] and radiating very long narrow wakes remotely visible [3]. Some of these phenomena originate abeam the ship in the form of extensive breaking of diverging bow and stern waves, eventually developing in wakes.

In the present work we discuss the complex fluid dynamics involved in the bow-wave radiation, including wave breaking, by means of numerical tools. The analysis is limited to high speed slender ships, with a sharp stem, for which basic insights can be achieved by an approximate quasi three-dimensional model based on the idea that longitudinal gradients of relevant flow quantities are small compared with vertical and transverse gradients. On this ground, the steady inviscid three-dimensional problem can be simplified. In fact, the resulting equations are mathematically equivalent to those governing the unsteady two-dimensional free-surface flow generated by a deformable body in the vertical plane transverse to the ship. This deformable body coincides with the ship cross section in that plane which deforms as the ship moves forward (see sketch in figure 1). Consistently, this approximation is here called 2D+t model, but it is often also referred to as 2.5D theory. Within the 2D+t framework, the nonlinearities induced by hull and free-surface deformations are fully retained. A historical recollection of slender-body theory for ship hydrodynamics is given by [4]–[6]. This approach results to be quite successful for studying steady and unsteady flows around high-speed vessels (see *e.g.* [7]). In the context of severe wave-ship interactions, the use of 2D+t has been exploited in [8], [9]. Moreover, an experimental compaign is reported in [10] and [11] where a 2D+t approach has been used to study breaking bow waves through the use of a deformable wave maker.

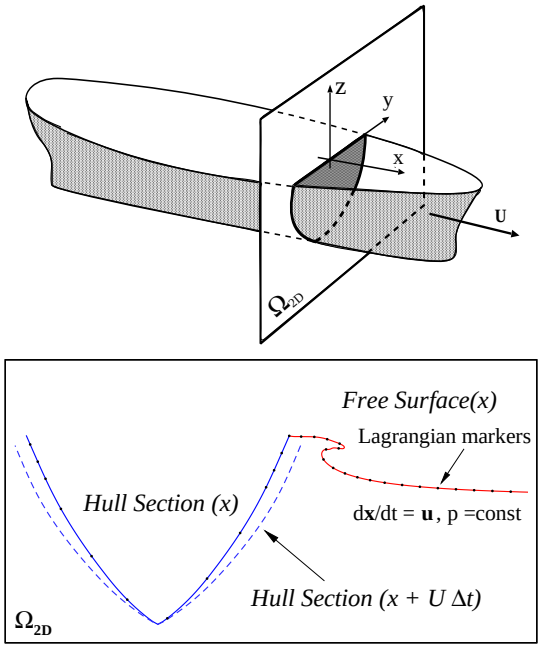


Fig. 1. Qualitative sketch of the 2D+t approximation for the steady three-dimensional flow around a ship with constant forward speed U. Left: 3D ship problem. Right: equivalent unsteady 2D problem (2D+t).

More relevant to the present investigation, the 2D+t approach has been adopted in [5] to gain the physical understanding of the genesis of divergent bow waves. In that case, the occurrence of the bow splash as essential in originating the system of divergent waves has been highlighted by numerically solving the 2D+t problem. A Boundary Element Method (BEM) was used. Though the emphasis of

A semi-analytic approach for SPH modelling of solid boundaries

A. Di Monaco, S. Manenti, M. Gallati, S. Sibilla
Dipartimento di Ingegneria Idraulica e Ambientale
Università di Pavia
Pavia, Italy

G. Agate, R. Guandalini, A. Maffio
Environment and Sustainable Development Dept.
CESI RICERCA S.p.A.
Milan, Italy

***Abstract*— A general semi-analytic approach for modelling solid boundaries in the SPH method is presented: they are considered as a material continuum with suitable distribution of velocity and pressure. Boundary contribution to each term of the SPH mass and momentum balance equations can be expressed in terms of a suitable integral extended to the volume intersection of the influence sphere of the particle with the boundary surface. Analytical details with reference to a slightly compressible viscous Newtonian fluid in three-dimensional geometry are given.**

I. INTRODUCTION

In the early application of the SPH method to astrophysical problems, such as [1] and [2], no solid boundaries were present and, consequently, no methods for handling boundary conditions was needed. More recently, when the method was extended to confined fluid-dynamics [3], in particular free surface hydrodynamics [4], several approaches for treating boundary conditions were introduced. They can be essentially gathered in the following four groups:

a) ghost (or image) particles: it appears the most rigorous method as it allows correct extension of the pressure and velocity conditions beyond the boundary. It is based on the generation, at each time step, of additional particles, as reflected images of the fluid particles located near the boundaries (seen as mirrors); the density (pressure) and velocity of each of these image particles are assigned so as to accomplish conditions of reflection or linear extension [5, 6, 7, 8, 9]. While this method can be easily implemented in two-dimensional (2D) geometry, in presence of straight boundary sides forming relative right angles; its application to three-dimensional (3D) geometry with plane boundary faces and right angles appears indeed rather cumbersome; its extension to the general case of 3D geometry, curved boundary faces and non-right angles appears particularly difficult.

b) boundary particles: it consists in replacing solid boundaries with particles which can be fixed or moving [10, 11, 12, 13]. The boundary particles, which have to fill a layer sufficiently thick layer, interact with the fluid particles through forces which depend on their relative distance; velocity and pressure of the boundary particles have to be properly assigned. The method seems to work well but it requires an extra computational effort, in terms of storage and time, for handling the additional boundary particles, which in 3D geometry can become particularly severe.

c) boundary particles and forces: which is based on the distribution of "guard" particles along the boundary sides in 2D, or faces in 3D, that exert suitable forces on the fluid particles depending on the relative distances [14]. The expressions of these forces are empirically assigned depending on the nature of the problem.

d) semi-analytic boundary integrals: based on direct evaluation of boundary contributions to SPH fluid-dynamic equations, without introducing additional particles. There are few examples of such techniques in the literature, either limited to particular problems [9] or just to the approximation of boundary forces [15], without taking into account the boundary effect on mass balance equation.

In this paper a general innovative semi-analytical technique is presented that overcomes the above limitations and drawbacks, with particular reference to those arising in 3D problems with very irregular boundaries. Its basic idea consists in the assumption that the portions of solid boundary, which give contribution to the mass and momentum balance equations of the generic fluid particle, can be considered as a material continuum (i.e. composed of an infinite number of particles) with suitable distribution of velocity and pressure.

In this way the boundary contribution to each term of the SPH fluid-dynamic equations (including the mass balance Eq.), related to the generic particle close to the boundary, can be expressed in terms of a suitable integral extended to the volume intersection of the influence sphere of the particle with the boundary surface.

In the following analytical details are provided with reference to slightly compressible viscous Newtonian fluids in three-dimensional geometry, bearing in mind that concepts and formulas can be easily reduced to two dimensions; for sake of simplicity boundaries are thought composed of plane faces (triangles and quadrangles), but the results are still valid with reference to curved boundary faces.

Applications which prove the validity of the proposed method are shown in a parallel paper [16], including in some cases a comparison with experimental data.

Towards the simulation of wave-body interactions with SPH

Jean-Marc Cherfils, Louis Blonce, Gregory Pinon, Elie Rivoalen
Laboratoire Ondes et Milieux Complexes
University of le Havre
Le Havre, France
jean-marc.cherfils@univ-lehavre.fr

***Abstract*—The present study deals with the development of a Smoothed Particle Hydrodynamics (SPH) code for the investigation of coastal engineering problems. The Navier-Stokes equations are solved using a parallel weakly compressible SPH solver. The parallelization uses a simple dynamic load balancing strategy, which greatly improves the efficiency of the code. To handle flows around submerged obstacles near shores, a new model for boundaries was required, which would allow to design complex structures. The immersed boundary method (IB-SPH) has been developed to model these submerged bodies, while the ghost particle technique is used to enforce conditions on simple exterior boundaries. The IB-SPH has been applied to model the incompressible flow around a moving square in a box (SPHERIC validation case 6), showing a good agreement.**

I. Introduction

A SPH parallel code has been developed for coastal engineering applications. More precisely, we are interested in studying wave-body interactions such as oscillatory flows around submerged bodies with complex shapes. In this kind of application, a parallel version of the algorithm is required, due to the size of the computational domain, to the massive number of particles and to the time needed to capture a particular physical phenomenon. A way to accurately enforce no-slip boundary conditions on submerged bodies with complex geometries is also required.

In this paper, we present our work on these two challenges: a simple but efficient parallelization strategy and a version of the Immersed Boundary Method to model rigid bodies interacting with SPH particles.

First the numerical scheme is described. The SPH equations are given and our the IB-SPH model for boundary conditions is described. Then, our parallelization strategy is detailled. And finally, our model is applied to the SPHERIC validation case 6.

II. Numerical scheme

A. SPH Equations

The Navier-Stokes equations are discretized in the following way:

$$\frac{D\rho_i}{Dt} = -\sum_j m_j \left(\boldsymbol{u}_{ij} - \boldsymbol{n}_{ij} \left(\frac{c_{ij}}{\rho_j} (\rho_j - \rho_i) \right) \right) \cdot \boldsymbol{\nabla} W_{ij} \quad (1)$$

$$\frac{D\boldsymbol{u}_i}{Dt} = \boldsymbol{rhs}_i + \boldsymbol{f}_i \quad (2)$$

$$\boldsymbol{rhs}_i = \boldsymbol{F}_i^p + \boldsymbol{F}_i^v + \boldsymbol{g} \quad (3)$$

$$\boldsymbol{F}_i^p = -\sum_j \frac{P_j + P_i}{\rho_i \rho_j} \boldsymbol{\nabla} W_{ij} m_j \quad (4)$$

$$\boldsymbol{F}_i^v = \mu \sum_j \frac{\boldsymbol{u}_{ij}}{\rho_i \rho_j} \zeta_{ij}^{\Delta} m_j \quad (5)$$

$$P_i = B \left(\left(\frac{\rho_i}{\rho_0} \right)^{\gamma} - 1 \right) \quad (6)$$

$$\frac{D\boldsymbol{x}_i}{Dt} = \boldsymbol{u}_i \quad (7)$$

with $\boldsymbol{u}_{ij} = \boldsymbol{u}_j - \boldsymbol{u}_i$, $\boldsymbol{n}_{ij} = \frac{\boldsymbol{x}_j - \boldsymbol{x}_i}{|\boldsymbol{x}_j - \boldsymbol{x}_i|}$ and $c_{ij} = \max(c_i, c_j)$ where

$$c_i = \sqrt{\gamma \frac{B}{\rho_0} \left(\frac{\rho_i}{\rho_0} \right)^{(\gamma - 1)}} \quad (8)$$

$\boldsymbol{\nabla} W_{ij}$ is the kernel gradient at $\boldsymbol{x}_j$ centered in $\boldsymbol{x}_i$. The kernel used for all the results in this paper is the classical gaussian function. In (1), a Rusanov flux has been introduced as proposed by Ferrari *et al.* [5]. This monotone upwind flux enhance the stability of the scheme by reducing density fluctuations, which are always observed while using a weakly compressible model. In that way, the flux plays the same role as the periodic MLS density re-initialization proposed by Colagrossi *et al.* [1]. The choice of introducing this flux instead of the density re-initialization has been motivated by the lower numerical cost it provides and by the fact that, no parameter calibration (*i.e.* the number of iterations between two re-initialization) is required. Moreover, using this flux with the immersed boundary method, which will be developed later, allows to significantly reduce the smoothing of the pressure jump across the boundary. In other words, it stabilizes the pressure field without being too dissipative.

The viscous term (5) is discretized using the Particle Strength Exchange (PSE), following the work by Eldredge *et al.* [4], the ζ_{ij}^{Δ} kernel being the gaussian function for the Laplacian approximation.

Analysis of WCSPH laminar viscosity models

L.M. González, J.M. Sánchez, F. Macià, A. Souto-Iglesias
Naval Architecture Department (ETSIN),
Technical University of Madrid (UPM)
28040 Madrid, Spain
antonio.souto@upm.es

***Abstract*—Our aim with this paper is to contribute to clarify the role played by the most commonly used WCSPH viscous terms in simulating viscous laminar flows. To achieve that, Takeda *et al.*'s [1], Morris *et al'*s [2] and Monaghan-Cleary-Gingold's [3], [4] viscous terms will be analyzed, discussing their origin, their structure and their conservation properties. Their performance will be monitored against canonical flows for which the viscosity related phenomena are well understood, and in which boundary effects have a relatively low relevance. One of the study cases has been a Lamb-Oseen vortex evolution for which both an exact solution and a finite-element numerical solution have been derived. The results of these tests confirm, those previously published. The SPH viscosity models behaviour in modelling the viscosity related effects for these canonical flows is adequate.**

I. Introduction

The important role that viscosity plays in many engineering and physical phenomena, boundary layers and forces evaluations, separation, transition flows, shear flows, etc.., underlines the need for a better understanding of modeling techniques of laminar viscous flows.

Modelling low Reynolds number viscous flows does not present excessive difficulties for industrial focused CFD methods like FVM or FEM, but these methods encounter difficulties when dealing with problems such as highly distorted free surface flows, where SPH is in general the first option. SPH method is also successful in modeling the bulk flow of these very turbulent problems. Vila [5] showed the consistency of the SPH approximation for solving the Euler equations but no similar result regarding the Navier-Stokes equations has been obtained so far. Motivated by a series of studies on SPH simulations of free surface viscous flows [6], [7], it became apparent to the authors the need for a better understanding of the SPH modelling of viscosity related phenomena. We think that the weakly compressible formulations (WCSPH) are the most adequate SPH models to tackle free surface flows which combine fragmentation and splashing, and they are as well a better option for these problems than mesh based methods, including remeshed SPH. Therefore, the WCSPH approach to laminar flows will be used in this paper.

The role played by the most commonly used SPH real viscous terms in simulating 2D viscous laminar flows will be discussed. We have selected three viscous terms Takeda *et al.*'s [1], Morris *et al.*'s [2] and Monaghan-Cleary-Gingold's [3], [4] representing important moments in the history of applications of SPH to viscous flows. They will be analyzed in terms of their conservation properties, and their performance will be monitored against canonical flows for which the viscosity related phenomena are well understood.

The idea underlying the selection of the different case-studies is that they should be representative of viscosity phenomena, with analytical solution and no boundary conditions that could have a significant effect in the flow. Basa et al. [8] carried out a study focussing on the robustness of the SPH formulations for viscous flows using a series of test cases for which no-slip boundary conditions are paramount important. We have tried to concentrate our efforts in a more primitive problem, the diffusive properties of the viscous fluids themselves.

We have selected as the most innovative and difficult of our case-studies, a viscous vortex. Of the different fluid mechanics models simulating a vortex, the Lamb-Oseen viscous vortex model has been selected. Parallel to the SPH simulation, a finite-element (FEM) simulation has been performed [9] for this case. The computations have been carried out changing the resolution and the domain size with SPH performing very much like the FEM and analytical solutions.

In the other case studies the corresponding 2D flows will be simulated using the three viscosity models and the issues of convergence, stability and accuracy will be addressed. The dependence of the accuracy of the approximation upon the exactness of the derivatives estimation is unquestionable [10]–[12]. The history of SPH viscosity runs parallel to the approximation of second derivatives that appear in the dissipative terms of the Navier Stokes equations. A study of SPH capabilities to approximate functions and derivatives will be undertaken aimed at clarifying the results obtained in the test cases.

II. Navier-Stokes discrete formulation

The complete SPH formulation considered will be the following:

$$\frac{d\rho_a}{dt} = \sum_{b\in\mathcal{N}_a} m_b\,\mathbf{v}_{ab}\,\nabla_a W_{ab} \tag{1}$$

$$\frac{d\,\mathbf{v}_a}{dt} = -\sum_{b\in\mathcal{N}_a} m_b\left(\frac{P_a}{\rho_a^2}+\frac{P_b}{\rho_b^2}\right)\nabla_a W_{ab} + \mathbf{\Pi}_a \tag{2}$$

$$\frac{d\mathbf{r}_a}{dt} = \mathbf{v}_a \tag{3}$$

$$P = \frac{\rho_0\, c_s^2}{\gamma}\left(\left(\frac{\rho}{\rho_0}\right)^{\gamma}-1\right) \tag{4}$$

An SPH Turbulence Model

J. J. Monaghan
School of Mathematical Sciences
Monash University
Australia
joe.monaghan@sci.monash.edu.au

***Abstract*—This paper is concerned with a model of turbulence which is a consistent extension of XSPH. It is similar to the Lagrangian averaged Navier Stokes alpha (LANS-alpha) turbulence model, but it satisfies different scaling laws while conserving energy, angular and linear momentum and circulation, though the latter is only conserved approximately. We use the model to simulate decaying turbulence in a two dimensional rigid box with no-slip boundary conditions. We first confirm that SPH without smoothing recovers the correct transfer of energy to long length scales and the correct decay of energy and entrosphy. When smoothing is introduced the energy in the short wave length modes is reduced. The correlation functions are also reduced for short length scales in such a way that the simulations with coarse resolution mimic those with finer resolution.**

I. Introduction

The Lagrangian averaged Navier Stokes (LANS) alpha equations are based on a the idea that there is an averaged velocity which carries the more disordered velocity with it (for references see Lunasin et al. 2007) The typical model defines a smoothed velocity $\widehat{\mathbf{v}}$ in terms of the the disordered velocity $\mathbf{v}$ by

$$\widehat{\mathbf{v}}(\mathbf{r}) = \int W(|\mathbf{r}' - \mathbf{r}|)\mathbf{v}(\mathbf{r}')\mathbf{dr}', \tag{1}$$

where the integration is over the region occupied by the fluid and W is a kernel which satisfies

$$\int W(|\mathbf{r}' - \mathbf{r}|)\mathbf{dr}' = 1. \tag{2}$$

It is common practice to replace the integral formulation by

$$\mathbf{v}(\mathbf{r}) = (1 - \alpha\nabla^2)\widehat{\mathbf{v}}(\mathbf{r}). \tag{3}$$

where ∇^2 is the Laplacian operator. If $\widehat{\mathbf{v}}$ and $\mathbf{v}$ are expanded in a Fourier series with coefficients of the Fourier term $e^{\imath\mathbf{k}\cdot\mathbf{r}}$, $\widehat{C}_k$ and C_k respectively, then equation (1) gives

$$\widehat{C}_k = C_k\widetilde{W}(k) \tag{4}$$

where $\widetilde{W}(k)$ is the Fourier transform of W. If W is a smooth function, as in the case of the SPH kernels, then $\widehat{C}_k \to 0$ more rapidly than C_k as $k \to \infty$ As an example (3) leads to

$$\widehat{C}_k = \frac{C_k}{1 + \alpha^2k^2}. \tag{5}$$

A disadvantage with (3) and with the smoothing discussed by Monaghan (2002) is that they are implicit and can only be solved by iteration. In three dimensions the large number of iterations required makes the implicit smoothing impracticable. We therefore consider a different smoothing which can be converted to the SPH formulation easily and involves negligible extra work. The smoothing we consider is given by

$$\widehat{\mathbf{v}}(\mathbf{r}) = \mathbf{v}(\mathbf{r}) + \epsilon\int(\mathbf{v}(\mathbf{r}') - \mathbf{v}(\mathbf{r}))W(|\mathbf{r}' - \mathbf{r}|)\mathbf{dr}', \tag{6}$$

where $0 < \epsilon < 1$. In this case the Fourier coefficients satisfy

$$\widehat{C}_k = C_k(1 + \epsilon(\widetilde{W}(k) - 1)), \tag{7}$$

and $\widehat{C}_k \to (1 - \epsilon)C_k$ as $k \to \infty$. Provided $(1 - \epsilon) \ll 1$ and $0 < \epsilon < 1$ then $\widehat{C}_k \ll C_k$ as $k \to \infty$. In the calculations we describe in this paper, the typical value of ϵ is 2/3, so that the high order Fourier coefficients are reduced by a factor 1/3 every step. The SPH equivalent of (6) is the XSPH equation (Monaghan)

$$\widehat{v}_a = \mathbf{v}_a + \epsilon\sum_b \frac{m_b}{\bar{\rho}_{ab}}(\mathbf{v}_b - \mathbf{v}_a)W_{ab}, \tag{8}$$

where we have used standard SPH notation. However, W does not need to be the same kernel as in the rest of the SPH equations, and it could have a different h from that used for the rest of the fluid. For example, the h could be larger to produce extra smoothing. The Euler equations of motion can be obtained by using the Lagrangian

$$L = \sum_b m_b\left(\frac{1}{2}\mathbf{v}_b\cdot\widehat{\mathbf{v}}_b - u(\rho_b)\right), \tag{9}$$

and the invariant energy (in the absence of viscous dissipation and external or boundary forces) is

$$E = \sum_b m_b\left(\frac{1}{2}\mathbf{v}_b\cdot\widehat{\mathbf{v}}_b + u(\rho_b)\right). \tag{10}$$

Because $d\mathbf{r}/dt = \widehat{\mathbf{v}}$, the Lagrangian must be written in terms of the $\widehat{\mathbf{v}}$ and $\mathbf{r}$ before the Lagrange equations are worked out. This requires inverting (8) to express $\mathbf{v}$ in terms of $\widehat{\mathbf{v}}$. The algebra is complicated but the work can be simplified in the following way. We write

$$\frac{W_{ab}}{\bar{\rho}_{ab}} = \frac{K(|\mathbf{r}_a - \mathbf{r}_b|, h_{ab})}{h_{ab}^d\bar{\rho}_{ab}} \tag{11}$$

where d is the number of dimensions, and K, and hence W, depend on a length scale h_{ab} which is symmetric in a and b, for example $h_{ab} = \frac{1}{2}(h_a + h_b)$. We assume that $\bar{\rho}_{ab}$ is

Incompressible Moving Boundary Flows with the Finite Volume Particle Method

Ruairi M. Nestor, Nathan J. Quinlan
Department of Mechanical and Biomedical Engineering
National University of Ireland, Galway
nathan.qunlan@nuigalway.ie

***Abstract*—The finite volume particle method resembles SPH but is formulated in terms of interparticle fluxes which are exactly analogous to intercell fluxes in the mesh-based finite volume method. Consequently, the method inherits many of the desirable properties of the classical finite volume method, including implicit conservation and a natural introduction of boundary conditions via appropriate flux terms. In this paper, we describe the extension to incompressible viscous flow with moving boundaries. An arbitrary Lagrangian-Eulerian approach is used to facilitate the treatment of moving bodies. Non-uniform particle distribution is used to concentrate computational effort in regions of high gradients. The method is validated for viscous incompressible flow in a lid-driven cavity at Reynolds numbers of 100 and 1000. To validate the simulation of moving boundaries, flow around a translating cylinder at Reynolds numbers of 40 and 100 is modelled. Results display good agreement with reference data from the literature, and with results for an equivalent flow around a stationary cylinder.**

I. Introduction

In the finite volume particle method (FVPM), particle interactions are defined in terms of fluxes, which are weighted depending on the overlap of the kernel supports. The FVPM is closely analagous to the classical finite volume method (FVM) and inherits many of its desirable properties, including exact conservation and a natural introduction of boundary conditions via appropriate flux terms. Well-established developments for traditional finite volume methods, such as upwind flux formulations, may be used in the FVPM without modifications. In this article, we describe a development of the finite volume particle method with pressure projection for incompressible viscous flow with moving boundaries (section III). Potential applications of this technique include flow in blood vessels and medical devices. The method is validated against well-established experimental and computational data by simulating flow over both stationary and moving circular cylinders at Reynolds numbers of 40 and 100 (section IV).

II. Governing equations

This article is concerned with flow of a Newtonian incompressible fluid. The continuity and momentum equations are written as

$$\nabla \cdot \mathbf{u} = 0 \tag{1}$$

and

$$\frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot (\mathbf{F} - \mathbf{G}) = 0, \tag{2}$$

where $\mathbf{u} = (u \quad v)^T$ is the 2-D velocity vector, $\mathbf{U} = \rho\mathbf{u}$ is the vector of conserved variables, ρ is density and t is time. $\mathbf{F} = (\rho\mathbf{u} \otimes \mathbf{u} + p\mathbf{I})$ represents the inviscid flux, where $\mathbf{I}$ is the $D \times D$ identity matrix, and D denotes the number of dimensions. The two-dimensional incompressible viscous flux is given by

$$\mathbf{G} = \begin{pmatrix} 2\mu\dfrac{\partial u}{\partial x} & \mu\left(\dfrac{\partial u}{\partial y} + \dfrac{\partial v}{\partial x}\right) \\ \mu\left(\dfrac{\partial u}{\partial y} + \dfrac{\partial v}{\partial x}\right) & 2\mu\dfrac{\partial v}{\partial y} \end{pmatrix}, \tag{3}$$

where μ is the dynamic viscosity. No equation of state is needed, and the governing equation for energy is decoupled from the momentum and continuity equations.

III. The finite volume particle method

A. Introduction

The FVPM for inviscid compressible flow is outlined in this section, following the original derivation by Hietel *et al.* [1]. The fluid is represented by a set of N particles. These particles are defined by compactly supported, overlapping test functions ψ_i of the form

$$\psi_i(\mathbf{x}, t) = \frac{W_i}{\sum_{j=1}^{N} W_j}, \tag{4}$$

where $W_i = W(\mathbf{x} - \mathbf{x}_i(t), h)$ is a compactly supported kernel function for particle i, centred at $\mathbf{x}_i$. The compact support radius is $2h$, where h is called the smoothing length, in keeping with the SPH convention. In regions of high particle density, the denominator in Eq. (4) is high, resulting in lower values of the test function. Thus the denominator normalises the kernel function to ensure that the test functions form a partition of unity, i.e.

$$\sum_{i=1}^{N} \psi_i(\mathbf{x}, t) = 1. \tag{5}$$

Each particle is associated with a volume

$$V_i = \int_\Omega \psi_i \, d\mathbf{x}, \tag{6}$$

and a discrete value of any field variable ϕ

$$\phi_i = \frac{1}{V_i} \int_\Omega \phi\psi_i \, d\mathbf{x}, \tag{7}$$

Incompressible Finite Pointset Method For Free Surface Flow

R. Vacondio & P. Mignosa
Department of Civil Engineering, Parma University
43100, Parma (Italy), Viale G.P. Usberti 181/A
e-mails: renato.vacondio@nemo.unipr.it, paolo.mignosa@unipr.it

***Abstract*—In this paper a Finite PointSet Method for the numerical simulation of viscous incompressible flow with a free surface is presented. Finite Pointset Method is a pure Lagrangian method based on Moving Least Square interpolations and uses a Projection Method to reproduce the incompressibility. In this paper a modified form of Chorin's original method is presented: the main advantage of this new formulation is the capability to reproduce the hydrostatic condition for free surface flow.**
A new technique for free surface detection based on the the geometrical characteristics of particle positions is proposed and tested: this solves problems of free surface detection of methods used in Incompressible SPH. In order to stabilize the numerical model a variable smoothing length and an algorithm to insert and remove particles are applied and tested against Taylor-Green flow. Finally the FPM for free-surface flow is validated for evolution of an intially circular water bubble and intial stages of dam break flows and a good agreement is achieved with analytic or reference solutions.

I. Introduction

Lagrangian meshless methods are becoming more popular in computational fluid dynamics because of their capability to provide a solution of Navier-Stokes equations even for complex flows with fragmentation and moving interfaces, such as free surface or multiphase separation.
The Finite Pointset Method (FPM) is a Lagrangian meshless method for numerical integration of pure incompressible Navier-Stokes equations originally introduced by Tiwari and Kuhnert [17]. This method is similar to incompressible SPH [10] because each particle carries a vector of field. Information and physical quantities are approximated using particles in a circular neighbourhood.
There are also some fundamental differences between the two methods: FPM is based on a moving least squares approach, where particles are just interpolation points without any associated mass. Because of these key features, boundary conditions can be enforced analytically using boundary particles, and particles can be added and removed in order to preserve the stability of the solution. To date the FPM has been confined to single or two phase flow, in this paper a free surface version of FPM is introduced by using a pure geometrical alghorithm in order to detect the free surface boundary condition. The capability to reproduce the hydrostatic condition in time is one of the fundamental features of free-surface numerical models, however classical FPM is not able to reproduce that condition; to overcome this problem a modification an Incremental Pressure Projection Method is used to solve the Navier Stokes equations.

II. Incremental Pressure Projection Method

We consider the incompressible Navier-Stokes equations, written using the lagrangian derivative

$$\nabla \cdot \vec{v} = 0 \tag{1}$$

$$\frac{D\vec{v}}{Dt} = -\frac{1}{\rho}\nabla p + \vec{f_e} + \vec{\Theta} \tag{2}$$

where $\vec{v}$ is the velocity, p is the pressure, ρ is the density of the fluid, $\vec{f_e}$ is the vector of external forces and $\vec{\Theta}$ is the viscous term equal to $\nu\nabla^2\vec{v}$ for a Newtonian fluid, where ν is the kinematic viscosity of the fluid.
Spatially discrete versions of the coupled Navier Stokes equations are cumbersome to solve directly. Observing that the right-hand side of equation (2) is a Hodge decomposition, Chorin [5] proposed the Projection Method (PM) in order to solve this system of partial differential equations. The key idea of the PM is to introduce a fractional step procedure: an intermediate velocity $\vec{v}^*$ is computed neglecting the gradient pressure in equation 2, then, projecting the final velocity onto a divergence free space, the pressure P is calculated solving a Laplace equation.

If the classical Projection Method is applied in a bounded domain Ω, then an issue arises: boundary conditions applied calculating $\vec{v}^*$ should be consistent with the final velocity although the final pressure p^{n+1} is not known, this problem is analyzed for Eulerian models by many authors (see for example [1] or [9]). The Incremental Pressure Projection Method (IPPM) was introduced by Brown et al. [4]; the version presented herein is an adaption for lagrangian projection methods.
The following discrete implicit formulation of (2) is initially considered

$$\frac{\vec{v}^{n+1} - \vec{v}^n}{\Delta t} + \frac{1}{\rho}\nabla p^{n+1} = 0.5\left(\vec{\Theta}^n + \vec{\Theta}^{n+1}\right) + \vec{f_e}; \tag{3}$$

with boundary condition $\vec{v} = \vec{v}_b$ on $\partial\Omega$.
In IPPM eqs. (1) and (3) are solved by means of Hodge decomposition as in Chorin's original method. In the first step

A particle based model to simulate plant cells dynamics

P. Van Liedekerke, E. Tijskens, H. Ramon
Department BIOSYST
Katholieke Universiteit Leuven
Leuven, Belgium
Paul.VanLiedekerke@biw.kuleuven.be

P. Ghysels, G. Samaey, D. Roose
Department Computer Science
Katholieke Universiteit Leuven
Leuven, Belgium
Pieter.Ghysels@cs.kuleuven.be

***Abstract*—In this paper we present a particle based model to simulate the dynamics of plant (parenchyma) cells and aggregates. The model basically considers two important features in the plant cell mechanics: (1) the fluid that is responsible for the hydrodynamics, which is modeled by Smooth Particle Hydrodynamics (SPH), and (2) the cell wall dynamics which is coarse grained by particles connected by a central interaction potential function, modeled by the Discrete Element Method (DEM). The cell wall hydraulic conductivity (permeability) is built in as well through the SPH equations. In the preliminary results presented in this paper, the model is subjected to benchmarks such as the retrieving of the Young-Laplace relation and the cell fluid pressure convergence during compression of a single cell. In addition, rheological experiments performed with cell aggregates are presented.**

I. Introduction

In their lives, plants are continuously exposed to stress situations. Exposure to drought, extreme temperatures, viral infections, sunlight, salinity and lack of oxygen are the most known herein. Without any doubt, plant stress results in a crop yield far below the genetic potential. When a plant is subjected to excessive mechanical stress (e.g. gravity, wind, impact) tissue damage (bruising) can occur, which is a major problem in the handling of fruits and some vegetables. As commercial production of fruits requires a lot of mechanical handling (transport, sorting, packaging), this can result in an economical loss of up to 10% [1]. Mechanical stress further also affects plant's physiological processes, defensive mechanisms and possibly even gene expression.

This paper describes further development of the microscopic physical model as was presented by Loodts et al. (2006) [2] that is able to predict the dynamic and static behavior of plant cells and aggregates. Therefore, we present a particle based model which is able to simulate plant cells with arbitrary shapes, possibly subjected to large deformations, and whereby both the solid phase (cell wall) and the more fluid like phase (cytoplasm) are considered. The approach for this is an SPH model where the boundaries are flexible and elastic. The overall goal of this research is to develop a multi-scale approach of plant tissue in a physical model approach. Eventually, the model should be able to predict, given an externally applied stresses (dynamic as well as static), the stress on the cellular level and this for a variety of cell physical properties and cell geometries. In the future, we intend to link the cell microcanonical properties (such as cell fluid viscosity and cell wall viscoelasticity-and plasticity) to the macroscopic properties (such as tissue (visco)elasticity and plasticity) by a multi-scale method as is presented in [3]. In this paper however, we shall in a primarily phase focus on the quasi-static mechanical behavior.

II. Cell model

Plant cells, unlike animal cells, are surrounded by a cell wall. Inside the cell, the incompressible cytoplasm builds up a hydrostatic pressure (turgor pressure) which is, besides essential for a lot of physiological processes, also responsible for the cells strength and rigidity (low hydrated plants lose their resistance against bending). The cell wall can be regarded as a thin semipermeable shell which allows for water transport. Water transport is also established between two adjoining cell walls by micro channels called *plasmodesmata*. Plant cells can be very different in nature. In this work however, we will focus on parenchyma cells. These types are typically thin walled cells that retain their cell content. Inside these cells large *vacuoles* are present, which serve as containers for the storage of water. Parenchyma are also responsible for carbon storage, gas transport and photosynthesis. In this work, our ansatz is to treat the cell wall as a linear viscoelastic solid, and the cell fluid as a homogeneous viscous fluid (in the mechanical sense). In addition we assume the cell wall is thin enough to have negligible bending resistance. Combining these assumptions, we propose a Discrete Element Model (DEM) whereby the cell wall is coarse grained by particles. Because of the focus on parenchyma, which make up the bulk in most plant tissue, the cells contain a lot of water (vacuoles). Therefore, we will assume the cytoplasm can be approximated by the Navier-Stokes equations. As shown in Fig. 1, the cell shape will be regarded as a cylindrical vessel whereby the model captures the 2D deformation (displacements as well as rates) of the cell in a plane (XY). To account for the deformation in the perpendicular direction (Z), we will restrict ourselves to simple elasticity theory and a volume constraint. As a consequence, velocity components are discarded in this

SPH Boundary Forces

J. J. Monaghan
School of Math. Sciences
Monash University
Australia
joe.monaghan@sci.monash.edu.au

J. B. Kajtar
School of Math. Sciences
Monash University
Australia
jules.kajtar@sci.monash.edu.au

***Abstract*—This paper is concerned with approximating arbitrarily shaped boundaries in SPH simulations. We model the boundaries by means of boundary particles which exert forces on a fluid. We show that when these forces are chosen correctly, and the boundary particle spacing is a factor of 2 (or more) less than the fluid particle spacing, the total boundary force on a fluid SPH particle is perpendicular to boundaries with negligible error. The variation in the force as a fluid particle moves, while keeping a fixed distance from the boundary, is also negligible. The method works equally well for convex or concave boundaries. Simulations of fluid dynamics with convex and concave surfaces confirm these results.**

I. Introduction

In most SPH simulations the boundaries of rigid bodies have been modelled by using (a) ghost particles, (b) fluid particles, (c) normalizing conditions or (d) boundary particle forces. All of these have advantages and disadvantages. Ghost particles (see for example Colagrossi and Landrini 2003) are similar to the ghost cells used in finite difference calculations. The ghost particles are produced by reflecting the real particles across the boundary (the way this is done depends on the boundary). A real particle produces an image or ghost particle which has the same density, pressure and temperature as their real particle, but with the perpendicular component of the velocity having the opposite sign, and the tangential component having the same sign (free slip) or opposite (no slip). If there is a external force on the fluid the pressure in the ghost cell may need to be adjusted. The disadvantage of these ghost particles is that when the boundary changes direction sharply, or there are two or more fluids, it is not clear how they should be placed. For example, in the latter case the interface between the two fluids must be taken into account.

If the boundaries are replaced by fluid particles, as in (b), at least two problems arise. The first is that when fluid moving away from the boundary, as in flow over a weir, the apparent density and pressure decreases, and the particles feel an attraction to the liquid particles on the boundary contrary to the effect of a real boundary. The second is that, when there is more than one fluid, the boundary particles must either be chosen to be one of the fluids throughout the calculation, or they are changed depending on which fluid interacts with them. Not only is this complicated but it is untested.

The option (c) is due to Feldman and Bonet (2004) who obtain the SPH density by summing over the particles and correcting the summation by a normalizing function near a surface. This normalizing function corrects for the absence of particles beyond the boundary. If this normalizing condition is included in the Lagrangian of the system extra forces appear in the equations of motion. These forces act normal to the surfaces, and increase from zero as the surface is approached from the direction of the fluid. In the case of fixed surfaces this method works well, but in more complicated situations it can be cumbersome. For example in a flow containing several bodies which can interact, where any two of the bodies are in rolling and sliding contact the normalizing function is different for every configuration of the bodies, and its evaluation is costly, especially in three dimensions.

The use of normal forces as in option (d) can be traced back to the work of Sirovich (1968) who formulated the boundary condition as a boundary force in the acceleration equation of the fluid. A closely related method is the immersed boundary method due to Peskin (1977). Boundary forces were used by Monaghan (1994) in the first application of SPH to the flow of incompressible bodies. The particular form of the forces was similar to the Lennard-Jones forces of molecular dynamics. These forces are, however, unsatisfactory because a fluid particle moving past the surface, and keeping a constant distance from it, can feel a non-uniform normal force and a non-zero tangential force. An alternative is to use normal forces (Monaghan et al. 2003) and the results from this method are satisfactory though not as good as ghost particles for straight line boundaries in two dimensional flow. Furthermore, the use of normals is not satisfactory for boundaries which are concave relative to the fluid because the normals can intersect.

In this paper we consider boundary forces which are radial and sufficiently smooth to ensure that, when the total force on a fluid particle is obtained by summing over the boundary particle forces, the final result is independent of the discrete nature of the boundary to a high degree of accuracy. We first establish this result for straight line boundaries in two dimensional configurations by using the Poisson summation formula to show that the force on the fluid particle is dependent only on the perpendicular distance of the fluid particle from the surface. We then confirm this result by direct summation of the boundary particle forces for typical boundaries for flow in two and three dimensions. Applications to fluid flow problems show that our results are in very good agreement with other authors using finite difference methods or exact solutions.

About compressible treatment and solid boundary conditions aspects of smoothed particle hydrodynamic

Blacodon Yohan
Department of Technical Energy Application
Institut Français du Pétrole (I.F.P)
Rueil-Malmaison, France
yohan.blacodon@ifp.fr

Bohbot Julien
Department of Technical Energy Application
Institut Français du Pétrole (I.F.P)
Rueil-Malmaison, France
julien.bohbot@ifp.fr

***Abstract*—Compressible flow simulation with moving interface is of great interest in aerodynamic study of automotive engine. In this paper a compressible approach of SPH method with special treatment of solid boundary conditions is presented. It will be shown that upwind techniques is of great interest to reduce numerical error of the method and get more accurate approximation. Following that idea a MUSCL approach is implemented. A treatment of solid boundary conditions using techniques like addition of mirror terms and resolution of partial Riemann problem will be detailed. All developments applied to the SPARTACUS-3D code are implemented according a compressible framework. The goal of the thesis work is to simulate the intake stroke in a combustion chamber of automotive engine.**

I. INTRODUCTION

Flows simulation's problem in engine come from the moving interface problem and thus from the need to remesh the computational domain often. SPH method provide us from this step in computational fluid dynamic but other problems occur. First of all we need to determine how to initialize the flow in a compressible framework. Then accuracy of calculation is one of the first priority. Finally impermeable solid boundary are needed. Following those ideas we describe the way used with the compressible-SPARTACUS code [10], first developed at EDF by R. Issa and D. Violeau, then modified at IFP, in order to simulate compressible flows with moving interfaces during the intake stroke in the engine's combustion chamber.

This paper presents the SPH formalism used in order to compute real complex fluid with solid boundary conditions. In the section 2 the physical model used for compressible simulation is described. In section 3 the particular treatment of compressible fluid is detailed with the use of a Riemann solver and its implementation in SPH formalism following Inutsuka's work [7]. Its influence on result's accuracy could be seen through the Riemann problem which is a good analytical test involving strong compressible effects. Section 4 is about our hybrid solid boundary condition developed following several works [5], [6] and [11]. Its description and implementation in the SPH formalism is shown. Finally, in section 5, all our developments are applied on a two dimensional test of intake stroke in a simple geometry of engine's combustion chamber.

II. PHYSICAL MODEL

Throughout this paper, the standard SPH formulation given below in equations (1-3) are used to discretise the Euler and Navier-Stokes equations. Following the work of J. Price [17] density is calculated by sum according to (1) in order to get better approximation in presence of contact discontinuity.

$$\rho_a = \sum_b m_b W_{ab} \tag{1}$$

$$\frac{dv_a}{dt} = -\sum_b m_b \left(\frac{p_a}{\rho_a^2} + \frac{p_b}{\rho_b^2} + \Pi_{ab} \right) \nabla_a W_{ab} \tag{2}$$

$$\frac{de_a}{dt} = \sum_b \frac{m_b}{2} \left(\frac{p_a}{\rho_a^2} + \frac{p_b}{\rho_b^2} + \frac{\Pi_{ab}}{2} \right) v_{ab} \cdot \nabla_a W_{ab} \tag{3}$$

Here ρ is the density, p the pressure, e the internal energy and γ the constant of gas. The subscript a and b denote particles in interaction. This system is closed by the ideal gas equation of state (4). A sound velocity is also associated to each particle (5).

$$p_a = (\gamma - 1)\rho_a e_a \tag{4}$$

$$c_a = \sqrt{(\gamma - 1)e_a} \tag{5}$$

W_{ab} denotes the kernel function $W(r_{ab}, h_{ab})$ evaluated with $r_{ab} = |x_a - x_b|$, the distance from particle a to particle b. h_{ab} is the mean of particle a and particle b's smoothing length defined by (6). ∇W denotes the gradient of the kernel function. Throughout this paper the cubic spline kernel of Monaghan and Lantazzio [14] is used.

$$h_a = \eta \left(\frac{m_a}{\rho_a} \right)^{1/D} \tag{6}$$

with $\eta = 1, 2$ and D the dimension of the problem.

Inlet-Outlet Boundary Conditions and Truly Incompressible SPH

G. Ramos-Becerra
IMAS, UNAM,
Apdo Postal 20-726
Mexico D.F. 01000, Mexico

C. Moulinec
STFC Daresbury Laboratory
Warrington, Cheshire
WA4 4AD, UK

D.R. Emerson
STFC Daresbury Laboratory
Warrington, Cheshire
WA4 4AD, UK

X.J. Gu
STFC Daresbury Laboratory
Warrington, Cheshire
WA4 4AD, UK

***Abstract*—As demonstrated by Lastiwka *et al.* [1], it is now possible to use reliable inlet-outlet boundary conditions for Weakly-Compressible SPH. This paper will show that it is also feasible to apply inlet-outlet boundary conditions for confined flows and truly incompressible SPH. Inlet and wall boundaries are computed using Dirichlet conditions on the velocity components and zero-Neumann boundary conditions on the pressure, with a buffer similar to that described by Lastiwka *et al.* [1]. Particles close to the outlet boundary are free to leave the domain, without any special treatment. Setting the inlet and wall boundary conditions, with the help of dummy particles, will be presented, with a special focus on the pressure condition. After validating the method for the Poiseuille flow, the backward facing step test case is simulated using the conditions described by Armaly et al. [2] at a Reynolds number of 100. Results show a good agreement with the experiment.**

I. Introduction

Smoothed Particle Hydrodynamics (SPH) has been traditionally used for free-surface flows. More recently, the method has been applied to flows commonly associated with conventional Computational Fluid Dynamics (CFD), i.e. confined flows or flows around obstacles. The type of boundary conditions usually employed in SPH are able to model the release of particles, their motion being created by gravity, to model periodic behaviour, with external forces driving the flow, or to model a wave generator, by using oscillating walls. Recently, Lastiwka *et al.* [1] have developed a method for Weakly-Compressible SPH to implement inlet-outlet boundary conditions which was applied successfully to simulate flow around a cylinder. To our knowledge, this inlet-outlet strategy has not been applied to Truly Incompressible SPH yet. The purpose of this work is to explain how to set-up these inlet-outlet boundary conditions with a special focus on the Dirichlet velocity and zero-Neumann pressure conditions. The work is first validated for Poiseuille flow and then for flow past a backward-facing step [2].

This article is arranged as follows: Section II deals with the numerics of the code, Section III explains how the the boundary conditions are implemented, Section IV presents results obtained for Poiseuille flow, and Section V simulates the flow over a backward facing step and compares to experimental obersvations. Finally, Section VI draws some conclusions.

II. Numerics

A. Projection Method

The system of equations reads:

$$\begin{aligned} \nabla \cdot \mathbf{v} &= 0, \\ \frac{D\mathbf{v}}{Dt} &= -\frac{1}{\rho}\nabla p + \nu \Delta \mathbf{v} + \mathbf{F}, \\ \frac{D\mathbf{r}}{Dt} &= \mathbf{v}, \end{aligned} \tag{1}$$

where $\mathbf{v}$ is the velocity vector, t the time, ρ the density, p the pressure, $\mathbf{r}$ the position, $\mathbf{F}$ an external force, ν the kinematic viscosity, $\frac{D\cdot}{Dt}$ the Lagrangian derivative, and ∇, $\nabla\cdot$ and Δ are the gradient, divergence and Laplacian operators, respectively. The projection method [3] [4] is used to solve the velocity-pressure coupling. The time discretization form of the momentum equation in Eq. 1 is split into two parts; the first being the prediction step based on viscous and external forces:

$$\frac{\mathbf{v}^* - \mathbf{v}^n}{\Delta t} = \nu \Delta \mathbf{v}^n + \mathbf{F}^n, \tag{2}$$

and the second step represents the correction based on the pressure force:

$$\frac{\mathbf{v}^{n+1} - \mathbf{v}^*}{\Delta t} = -\frac{1}{\rho}\nabla p^{n+1}, \tag{3}$$

where Δt is the time step and n stands for the n^{th} time iteration.
The auxiliary velocity field $\mathbf{v}^*$ is usually not divergence free and incompressibility is imposed upon $\mathbf{v}^{n+1}$. Hence, the auxiliary velocity is projected onto the divergence-free space by applying the divergence operator to Eq. (3), as:

$$\nabla \cdot \frac{\mathbf{v}^{n+1} - \mathbf{v}^*}{\Delta t} = -\nabla \cdot \Big(\frac{1}{\rho}\nabla p^{n+1}\Big). \tag{4}$$

With the condition of incompressiblity imposed on $\mathbf{v}^{n+1}$, i.e. $\nabla \cdot \mathbf{v}^{n+1} = 0$, the pressure equation reads:

$$\Delta p^{n+1} = \frac{\rho}{\Delta t}\nabla \cdot \mathbf{v}^*. \tag{5}$$

Once the pressure has been obtained from Eq. 5, the velocity is updated by using Eq. 3.

A Study of Sloshing Absorber Geometry for Structural Control with SPH

Adam Marsh[1], Mahesh Prakash[2], S. Eren Semercigil[1] and Özden F. Turan[1]

[1]School of Engineering and Science
Victoria University, Melbourne, AUSTRALIA

[2]CSIRO Mathematical and Information Sciences
Clayton, AUSTRALIA

A liquid sloshing absorber consists of a container, partially filled with liquid. The absorber is attached to the structure to be controlled, and relies on the structure's motion to excite the liquid. Consequently, a sloshing wave is produced at the liquid free surface within the absorber, possessing energy dissipative qualities. The behaviour of liquid sloshing absorbers has been well documented, although their use in structural control applications has attracted considerably less attention.

The primary objective of this work is to numerically demonstrate the relationship between the sloshing absorber's shape and its control performance. Smoothed Particle Hydrodynamics (SPH) is used to model fluid-structure interaction in two dimensions.

I. INTRODUCTION

Sloshing is the oscillation of a liquid within a partially full container. In study of sloshing, efforts are usually made in the direction of suppression, due to the damaging effects it can impose [1-3]. On the other hand, sloshing has an inherent ability to dissipate large amounts of kinetic energy via shearing of the fluid. For this reason, it is possible to employ liquid sloshing as an effective energy sink in structural control applications, providing protection for structures exposed to excessive levels of vibration [4-7].

Liquid sloshing in rectangular tanks have long been an area of study [1,2,8,9]. Significant effort has been focused on increasing the energy dissipation performance of this type of sloshing absorber. Variations on the conventional design have included the introduction of a net within the main body of fluid [10], wedge shaped objects on the container bottom [11], baffles on the container walls [12] and wall flexibility [13]. Works involving varying liquid depth have been undertaken, focusing on increasing the amount of energy dissipation produced [14-17]. These studies have found that shallow depths are more effective at dissipating energy than deep liquid levels. Yet, the effect of different container shapes has not been reported. The primary motivation of the present work is to explore potential enhancements with shapes different than a rectangle.

SPH is used in this study due to its grid free nature, and inherent ability to capture free surface behaviour accurately [18, 19]. Good correspondence between SPH predictions and experiments within rectangular liquid sloshing absorber geometry has been demonstrated for both kinematic and dynamic boundaries [20].

II. NUMERICAL MODEL

Smoothed Particle Hydrodynamics (SPH) is used in this study to model fluid-structure interaction. The code used here has been developed by CSIRO's Mathematical and Information Sciences Division. A broad range of complex industrial fluid flow problems have been modelled successfully with this code [21]. SPH is used due to its ability to capture complex free surface behaviour accurately. The rigid container of a sloshing absorber is represented by a partially constrained moving boundary.

A schematic representation of the structure is given in Figure 1(a) where tethers attached to the container, represent structural stiffness (k) and viscous damping (c). The structure's stiffness, and mass (m) are 4260 N/m and 60.5 kg respectively, to give a natural frequency of 1.33 Hz. The viscous damping coefficient is chosen to give a 1% critical damping ratio, to represent a lightly damped resonant structure. The sloshing fluid is water with a density of 1000 kg m^{-3} and dynamic viscosity of 0.001 Pa s.

A particle size of 0.8 mm by 0.8 mm has been found to be fine enough to model the water within the container in a two-dimensional space. A resolution study has been completed [20], but is not shown here for brevity. Time stepping is explicit and is limited by the Courant condition modified for the presence of viscosity [18]. The time step of integration is $1\times10^{-5}s$. The total real time simulated is 30 seconds.

The fluid within the container is allowed to settle under gravity for a period of one second so that it reaches an initial state of rest. The structure is then subjected to an initial velocity of 0.5 m/s over one time step. The structure is allowed to respond freely, its motion exciting the water within. Liquid depth is kept relatively shallow in order to promote a travelling waveform. In all cases, mass of the sloshing water is 6.05 kg, 10% of the structure's mass.

III. NUMERICAL OBSERVATIONS

The dashed line in Figure 1(b) corresponds to the response of the uncontrolled structure. Maximum displacement reached by the structure is around 60 mm which is maintained for a long while due to light damping characteristics. The response of the structure is significantly improved with a tuned rectangular sloshing absorber, indicated by the solid line in Figure 1(b). For the rectangular container, the container width and liquid depth have been chosen to produce a fundamental sloshing frequency equal

Using SPH in a co-simulation approach to simulate sloshing in tank vehicles

Alexandra Lehnart, Florian Fleissner, Peter Eberhard
Institute of Engineering and Computational Mechanics
University of Stuttgart
Pfaffenwaldring 9, 70569 Stuttgart, Germany
[lehnart, fleissner, eberhard]@itm.uni-stuttgart.de

***Abstract*—The motion of liquid cargo in transport vehicles can have significant influence on the driving characteristics. The influence can be both beneficial and negative regarding the driving stability and braking performance. The behaviour of the cargo depends strongly on the design of the tank, e.g., the form of a tank or the number of compartments. Neglecting the influence of the dynamically moving cargo in driving simulations leads to significant errors in the simulation results. We propose a new method for the dynamic simulation of tank trucks carrying fluids that couples the Lagrangian particle method Smoothed Particle Hydrodynamics and Multibody Systems using co-simulation. The capability of the new approach is demonstrated by providing simulation results of two benchmark maneuvers with two different tank geometries.**

I. Introduction

The motion of fluids in tank trucks may have a significant influence on the driving dynamics. Tank geometries and vehicle suspension systems have to be designed carefully in order to provide stability during braking and lane change maneuvers as well as to improve driving comfort. Dynamic simulations are an attractive tool to predict the influence of different design parameters during the design process. The size of separate tank compartments is only one example for an important design parameter which affects the wavelength of the sloshing motion and thus the eigenfrequencies of the system.

Approaches for the simulation of tank vehicles have been proposed in the past decades, mostly based on the coupling of Multibody Systems (MBS) and pendulum models or Eulerian fluid simulation methods such as the Finite Volume Method, see e.g. [1]. However, considering sloshing liquids, such Eulerian approaches have difficulties with treating the involved free surface flows.

Therefore, a new co-simulation approach is implemented and tested that couples Pasimodo [2], a Lagrangian simulation framework for the 3D simulation of granular materials and fluid models, with Simpack [3], a commercial Multibody System simulation software. The approach is flexible and robust and theoretically enables coupled simulations between Pasimodo and any other Multibody System simulation software that provides a TCP/IP-co-simulation interface. The coupled simulation approach is tested on the basis of driving maneuvers of tank trucks and the influence of some characteristic design parameters on the stability of driving dynamics are studied.

II. Vehicle Model

For the simulation of the vehicle we chose the classical Multibody System approach [4], [5]. The commercial MBS modelling and simulation program Simpack is used to create a model of the truck with 17 degrees of freedom, see Fig. 1. The tire forces are computed by means of the Tire Similarity Model [6]. To account for a driver two additional degrees of freedom are added which are influenced by a feedback controlled driver model. All relative rotations are defined as Cardan angles. The rotations of the front wheel around the vertical axis are supplied by a driver model [7].

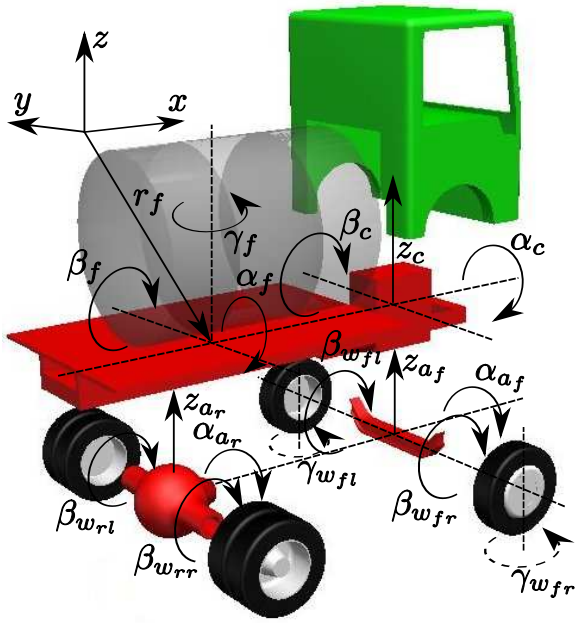


Fig. 1. Schematic representation of the MBS truck model.

In Table I some important simulation parameters of the vehicle model are listed. Also details on the tank geometries used in our simulations are given.

Different aspects of the vehicle are of interest in the MBS model and the particle model. In the MBS the particle forces acting on the tank are considered as applied forces. As the particle forces are calculated by the particle simulator, no

Fuel Loads in Large Civil Airplanes

Francesco Gambioli
Loads and Aeroelastics
Airbus
Filton (Bristol), UK
francesco.gambioli@airbus.com

Loads due to the fuel movement in wing tanks of large civil airplanes are investigated both experimentally and numerically.

I. INTRODUCTION

The certification authorities (the European Aviation Safety Agency or EASA in the EU) request the manufacturers to assess the aircraft dynamic response to pilot inputs (maneuvers), atmospheric turbulence (gust), and contact with ground (landing impact, taxi on runway). A range of mass distributions, altitudes and speeds has to be investigated to show compliance with the Certification Specifications for Large Airplanes (CS-25).

The above conditions are modeled using distributed inertia, fully flexible structure, unsteady aerodynamics and ground forces, coupling with the electronic flight control system. The results of this simulation exercise are time histories of quantities such as forces, moments and accelerations on the different aircraft components.

The peaks in the acceleration time histories output by the simulations are used to determine the acceleration in-service envelope of the wing fuel tanks. The instantaneous extreme values (maximum and minimum) of the time histories, together with the tank geometry and the fuel mass distribution in the tank, are the input for the subsequent calculation of fuel pressure loads on the wing tank walls.

Fuel pressure is traditionally calculated by quasi-static methods but an improved methodology for fuel pressure calculation has been developed based on Smooth Particle Hydrodynamics (SPH) methods. In order to assess the performance of these methods a series of tests has been defined in terms of acceleration time histories against experimental evidence. The tests have been chosen in order to represent a range of different aircraft types, in the current operational fleet. Additional test cases have been included to analyze the fuel loads for harmonic excitations and impulsive tank accelerations. The harmonic excitation tests have been tuned to the fundamental mode frequencies of the wing structure and the natural sloshing frequencies of the fluid in the tank. Both lateral (parallel to the free surface) and vertical (normal to the free surface) movements are analyzed.

The results of the test cases are used to evaluate the SPH pressure predictions against the experimental measurements, and consequently to validate the use of the SPH method for defining a fuel pressure calculation methodology.

Experimental observations of water movement generated in a rectangular tank of dimensions 1000×590×155mm^3 (length × width × height) are reported. The experiments were carried out at the University of Bristol in November 2008. A vertical slotted baffle at the middle of the tank width was used to reproduce the typical configuration of aircraft ribs in wing tanks. Video recordings and pressure measurements are presented and compared with the SPH code BAESPH, developed and run by BAE-Systems [6].

Finally a discussion of the findings is given to define the capabilities and limitations of the BAESPH code for fuel pressure loads in the wing tanks of large airplanes.

II. AIRCRAFT LOADS CALCULATION PROCESS

This section describes the process and methodology currently used for fuel pressure calculation on large civil airplanes, from the generation of acceleration data in the overall aircraft model, to assumptions used downstream for the actual pressure evaluation.

A. Loads model

In the current design process, the loads model build activity is the primary task of a group of specialists within Loads & Aeroelastics department (L&A). They incorporate information coming from other departments (e.g. Aerodynamics, Mass Properties, Structures, Systems) into a mathematical representation of the aircraft under study and its surrounding environment (the loads model). This is then used for the calculation of loads at various points on the airframe (loads stations).

The model is used to simulate the aircraft response to the environmental (gust, ground) and human (maneuvers) inputs. The simulation outputs are a series of loads cases, defined as snapshots in time of the state of the airplane under external loading balanced by the inertia forces.

The requirement to cover the complete flight envelope (combinations of aircraft speed and altitude) and all the possible mass distributions (different combinations of payload and fuel distribution) leads to a high number of computations (on the order of millions); therefore part of the L&A activity focuses on “reducing” the complexity of the loads model, while maintaining the level of accuracy needed

Laboratory experiments and sph modelling of hydraulic jumps

Diana De Padova, Michele Mossa
DIASS, Technical University of Bari
Bari, Italy
d.depadova @poliba.it
m.mossa@poliba.it

Stefano Sibilla
Dipartimento di Ingegneria Idraulica e Ambientale,
University of Pavia
Pavia, Italy
stefano.sibilla @unipv.it

Abstract

The formation of a hydraulic jump in a channel was investigated and reproduced using the SPH numerical model. In particular, the 2-D model, obtained by a pseudo-compressible XSPH scheme, is applied to the modelling of an undular jump generated in a very large channel of the Coastal Engineering Laboratory of the Water Engineering and Chemistry Department of the Technical University of Bari (Italy). The study made particular reference to the velocity and free surface profile measured in the longitudinal central section of the channel, with the aim of analysing the hydraulic jump development. The agreement between the numerical results and laboratory measurements was satisfactory.

I. INTRODUCTION

The hydraulic jump is an important type of energy dissipation structure in hydraulic engineering. Such structures are subjected to considerable pressure fluctuations due to the dynamics of turbulence inside the hydraulic jump. The literature on the macroscopic features of the hydraulic jump is very extensive, but many characteristics of the internal flow phenomena remain unanswered.

The present paper presents the modelling of the development of an undular jump using the SPH code.

SPH is a purely Lagrangian method developed during seventies (Lucy, 1977; Gingold and Monaghan, 1977) in astrophysics to study the collision of galaxies and the impacts of bolides on planets. The numerical method has been shown to be robust and applicable to a wide variety of other fields. The implemented numerical code was first tested using physical experiments on supercritical flow motion by Ben Meftah et al. (2007; 2008). In particular, it is applied to the modelling of an undular jump generated in a very large channel of the Coastal Engineering Laboratory of the Water Engineering and Chemistry Department of the Technical University of Bari (Italy).

The undular jump is formed by low supercritical inflow Froude numbers, and is characterized by undulations of the water surface without a surface roller (Chow, 1959). The characteristics of undular jumps are significant for the design and management of hydraulic structures. Ohtsu et al. (2001) observed that the formation of undular jumps depends not only on the inflow Froude number, but on the boundary layer development at the toe of the jump under conditions in which the effects of the aspect ratio and the Reynolds number on the flow condition are negligible. Ohtsu et al. (2001) proposed also a relation between the wave height of the first wave crest and the inflow Froude number. The Authors concluded that the flow conditions of undular jumps can be classified based on the intersection of lateral shock waves and the inflow Froude number, observing also the importance of the Reynolds number and the increasing of the angle of lateral shock waves to sidewall with the increase of the Froude number, in contrast with the classical shock wave theory. The flow velocity and the free surface elevation measurements were taken by using a two-dimensional Acoustic Doppler Velocimeter (ADV) and an ultrasonic profiler, respectively. SPH simulations were obtained by a pseudo-compressible XSPH scheme with pressure smoothing. Eddy viscosity is evaluated through a mixing-length model depending on the distance from the channel bottom and from the free surface.

II. EXPERIMENTAL SET-UP

The experimental investigation was carried outdone at the Coastal Engineering Laboratory of Valenzano (L.I.C.) of the Water Engineering and Chemistry Department.

The system (see Fig. 1) was compounded by a rectangular steel channel having the base and the lateral walls in transparent glass material of thickness 15 mm, connected and sealed internally with silicone rubber watertight and also able to prevent thermal dilatation. In plant the base had the surface of 15 m by 4 m and it was 0.96 m distant from the floor, whereas the height of the walls, and so the maximum allowed depth of the channel, was 0.4 m. To have a flow into the channel, a proper closed hydraulic circuit worked.

Fig. 2 shows a picture of the channel in which the upstream tank is clearly visible with the lateral pipe directed toward the downstream tank in order to regulate the total discharge flowing along the main channel.

For the measurement of the velocity the bidimensional Nortek ADV system was used, together with CollectV

Flow modelling in the injector of a Pelton turbine

Phoevos K. Koukouvinis, John S. Anagnostopoulos, Dimitris E. Papantonis
Department of Fluids, School of Mechanical Engineering
National Tecnhical University of Athens
Athens, Greece
fivoskouk@gmail.com, j.anagno@fluid.mech.ntua.gr, papan@fluid.mech.ntua.gr

***Abstract*— This work is an attempt to use SPH for modeling the fluid flow in the injector of action water turbine, like the Pelton type. The standard SPH is used along with a special technique where only one symmetric part of the fluid domain is solved, aiming to reduce the time needed for the calculations. This approach is tested at the case of jet impingement on a flat plate, where it proves that it can give a good prediction of the flow and the pressure profile, while being substantially faster than the standard SPH encouraging its use for design optimization.**

I. INTRODUCTION

Pelton turbines are the preferred type of hydro-power, when the available water source has a high hydraulic head at variable flow rates. Also they offer good efficiency and are very scalable; their size ranges from multi-ton Pelton wheels in hydroelectric plants to Pelton wheels which are only a few inches across, used to extract power from mountain streams in remote regions. The design of action turbines, though, is rather difficult in comparison to the other turbine types, since their operation involves free-surface unsteady flows. Such flows are hard to be described using an Eulerian approach with traditional grid-based techniques. On the other hand the method of Smoothed Particles Hydrodynamics (SPH), due to its Lagrangian, mesh-free nature, is able to deal with the difficulties appearing in such flows.

One drawback of SPH over grid-based techniques is the need of large number of particles in order to obtain simulations of equivalent resolutions. The large number of particles entails large number of interactions, thus increasing the computational cost. In this work a method of solving only a symmetric part of the whole problem domain will be presented. This approach produces results in considerably less time, depending on the size of the symmetric part used, but the accuracy of the results is affected below a certain size of the symmetric part. It can give, though, an average prediction of the flow. This approach can be used in any axis-symmetric problem, or problem with an axis/plane of symmetry. The advantages and disadvantages of this approach will be clarified in a test case (vertical jet impingement on a flat plate) and it will be used afterwards to model the flow in a Pelton injector. Results will be compared with the standard SPH method.

II. GOVERNING EQUATIONS

The equations used are those obtained after the standard SPH discretization procedure as it is presented by Violeau and Issa [1].

We use the continuity equation for the density calculation:

$$\frac{d\rho_a}{dt} = \sum_b m_b \mathbf{u}_{ab} \bullet \nabla_a W_{ab} \tag{1}$$

and the momentum equation :

$$\frac{d\mathbf{u}_a}{dt} = -\sum_b m_b \left[\left(\frac{p_a}{\rho_a^{\,2}} + \frac{p_b}{\rho_b^{\,2}} \right) \nabla_a W_{ab} - \Pi_{ab} \right] + g \tag{2}$$

where Π_{ab} is the viscosity term suggested by Morris et al.[2]:

$$\Pi_{ab} = \frac{(\mu_a + \mu_b)\cdot \mathbf{u}_{ab}}{\rho_a \rho_b} \left(\frac{1}{\|r_{ab}\|} \frac{\partial W_{ab}}{\partial r_a} \right), \tag{3}$$

where $\mathbf{u}_{ab} = \mathbf{u}_a - \mathbf{u}_b$, $\mathbf{r}_{ab} = \mathbf{r}_a - \mathbf{r}_b$ and $\|r_{ab}\| = \sqrt{(x_a - x_b)^2 + (y_a - y_b)^2 + (z_a - z_b)^2}$ (bold denotes vector and $\bullet$ the dot product). A small parameter is added to r_{ab} in order to avoid zero denominators and is not presented in the equations for simplicity. For the pressure calculation the following equation of state is used:

$$p_a = \frac{\rho_0 c_0^{\,2}}{\gamma} \left[\left(\frac{\rho_a}{\rho_0} \right)^{\gamma} - 1 \right], \tag{4}$$

where ρ_0 is the reference density and c_0 is the speed of sound. In order to keep density variations less than 1% the value of c_0 is chosen $\sim 10 \cdot V_{\max}$, according to Monaghan [3].

The integration in time is implemented by a fourth-order Runge-Kutta scheme. In order to reduce the large pressure oscillations at the pressure field of particles, we use density re-initialization (Shepard Filter) [4], where:

$$\rho_a^{new} = \sum_b m_b \cdot \tilde{W}_{ab}, \tag{5}$$

$$\tilde{W}_{ab} = \frac{W_{ab}}{\sum_b W_{ab} \frac{m_b}{\rho_b}} \tag{6}$$

Also for moving the particles, the XSPH variant [5] is used, where the velocities of the nearby particles are taken into account:

$$\frac{d\mathbf{r}_a}{dt} = \mathbf{u}_a - \varepsilon \sum_b \frac{m_b}{\rho_b} \mathbf{u}_{ab} W_{ab}, \tag{7}$$

Simulation of SPHERIC Benchmark Test 2, "3D schematic dam break and evolution of the free surface", by an improved parallelized particle method and SPHysics

Hitoshi Gotoh, Abbas Khayyer, Hiroyuki Ikari
Department of Urban and Environmental Engineering
Kyoto University
Kyoto, Japan

Abstract— **The paper presents the simulation of a SPHERIC benchmark test case, namely, "3D schematic dam break and evolution of the free surface" [1], by an improved parallelized particle method and the 3D SPHysics code [2] . The improved particle method is a 3D parallelized version of an improved MPS (Moving Particle Semi-implicit) method [3], namely, the CMPS-HS (Corrected MPS-Higher order Source term) method [4]. The parallelization has been carried out by a dynamic domain decomposition approach. Furthermore, two different solvers, namely, the PICCG-RP (Parallelized ICCG with Renumbering Process) [5] and the SCG (Scaled Conjugate Gradient) [6] techniques have been considered for the iterative solution process of the simultaneous linear equations corresponding to the Poisson Pressure Equation (PPE). Detailed comparisons have been made in terms of time history of pressure and water surface elevation calculated by the 3D parallelized CMPS-HS and the 3D SPHysics code [2]. The results by some other particle-based and grid-based methods have been considered for further comparisons.**

I. INTRODUCTION

The Moving Particle Semi-implicit (MPS) method [3] is a macroscopic deterministic particle method developed initially for the simulation of incompressible free-surface fluid flows. The MPS method is similar to the SPH (Smoothed Particle Hydrodynamics) method [7,8] in that both methods provide approximations to the strong form of the Partial Differential Equations (PDEs) on the basis of integral interpolants. However, the MPS method applies simplified differential operator models solely based on a local weighted averaging process without taking the gradient of a kernel function. In addition, the solution process of MPS method differs to that of the original SPH method as the solutions to the PDEs are obtained through a semi-implicit prediction-correction process rather than the fully explicit one in original SPH method.

Through the past years, the MPS method has been applied in a wide range of engineering applications such as elastic structures [9], nuclear reactor safety [10], blood flow simulation [11] or stir mixing of viscous liquids [12]. The MPS method has been improved and extended into coastal engineering to study wave breaking [13,14,15] and two-phase sediment-water interactions [16]. Gotoh et al. [17] developed a 3D numerical wave flume based on the MPS method.

In spite of its capability and simplicity, the MPS method has a few drawbacks including the non-conservation of momentum [18] and existence of spurious pressure fluctuations [4,14,19]. By focusing on the momentum conservation properties of the MPS approximations for particle interacting forces and by deriving an anti-symmetric pressure gradient model, Khayyer and Gotoh [18] proposed a revised version of the MPS method, namely, the CMPS (Corrected MPS) method. Enhanced conservation of both linear and angular momentum by the CMPS method resulted in refined reproductions of two-dimensional plunging breaking waves and resultant splash-ups [18,20]. In another recent work, Khayyer and Gotoh [4] revisited the derivation of the Poisson Pressure Equation (PPE) in the original MPS method and derived a higher order source term based on a more accurate time differentiation of particle number density. Enhanced simulations of wave impact pressure were provided by the CMPS-HS (CMPS with a Higher order Source term) method.

This paper presents the simulation of a SPHERIC benchmark test, namely, "3D schematic dam break and evolution of the free surface" corresponding to the experiment by Kleefsman et al. [1] by a 3D parallelized version of CMPS-HS method and the 3D SPHysics code [2]. The 3D CMPS-HS method has been developed on the basis of 3D MPS method by Gotoh et al. [17]. The parallelization of 3D CMPS-HS has been carried out by a dynamic domain decomposition approach for optimized load balancing among the processors. Furthermore, two different solvers, namely, the PICCG-RP (Parallelized ICCG with Renumbering Process) [5] and SCG (Scaled Conjugate Gradient) [6] techniques have been considered for the iterative solution process of the simultaneous linear equations corresponding to the PPE.

Assessment of Incompressible and Weakly Compressible SPH for Marine Applications

Olga Shipilova
Det Norske Veritas
DNV Research & Innovation
Høvik, Norway
Olga.Shipilova@dnv.com

Arne Bøckmann
Det Norske Veritas
DNV Energy
Høvik, Norway
Arne.Bockmann@dnv.com

Geir Skeie
Det Norske Veritas
DNV Research & Innovation
Høvik, Norway
Geir.Skeie@dnv.com

Pål Bergan
Det Norske Veritas
DNV Research & Innovation
Høvik, Norway
Paal.Bergan@dnv.com

***Abstract*—The paper presents preliminary results from an assessment and verification study where both the Weakly Compressible and Incompressible SPH forms are considered as numerical tools for industrial simulations of flow problems with free interfaces. General questions regarding representation of boundary conditions, convergence, robustness and accuracy as well as the feasibility of performing long simulations are addressed. In addition, the methods' ability to treat the incompressibility constraint and the accuracy of local results, i.e. the values of the pressure field in a given point, are also explored. Two-dimensional fluid sloshing in a partially filled container is considered as benchmark problem.**

I. Introduction

Violent (non-linear) free interface behavior is inherent to many marine applications such as bow slamming, sloshing in LNG tankers, life-boat impacting the sea surface, green water on a ship deck, etc. Much efforts are made in industry in order to adapt numerical methods and techniques which can be effective for simulation of such problems. The Smoothed Particle Hydrodynamics method is the one of the first candidates to be assessed as a potentially strong numerical tool for industrial use.

An intensive discussion on advantages and disadvantages of two variations of SPH can be found in the literature. From our experience, we may say that Weakly Compressible SPH (WCSPH) is more intuitive and maybe more straight-forward for solving free interface problems. The interface is tracked automatically without any external means, such as the level set or volume of fluid functions. The forces acting between particles can be considered as a counterpart of inter-molecular forces. However, the calibration of the parameters and correction terms in WCSPH requires significant work. On the other side, Incompressible SPH (ISPH) has a more conventional theoretical background; not so many parameters need to be tuned. However, many authors point out the disadvantage of large computational expenses for ISPH.

This study presents our preliminary results on verification and comparison of two variations (Weakly Compressible and Incompressible) of SPH. The goal was to understand fundamentals and to assess the accuracy and complexity of each scheme. The fluid sloshing in a partially filled, two-dimensional container is chosen as benchmark problem. It is assumed that the fluid is incompressible and viscous. A pitching excitation of the container with an initially still free surface is simulated. For practical reasons we use the gravitational force with direction depending on time (corotational reference frame) instead of implementing the actual movement of the container boundary. The first case study focuses on linear behavior of a free interface, i.e. no breaking waves and formation of jets or cavitation are involved.

The paper is organized as follows. Section II formulates the problem of fluid sloshing in the container. The basic description of Weakly Compressible and Incompressible SPH implemented in this study is given in Section III. The numerical experiments presented in Section IV and some conclusions are drawn in Section V.

II. Problem Formulation

Incompressible viscous flow can be defined by the Navier-Stokes system in Lagrangian coordinates as

$$\frac{d\mathbf{v}}{dt} = -\frac{1}{\rho}\nabla P + \mathbf{g} + \mathbf{\Pi} \quad \text{in} \quad \Omega \times (0,T), \tag{1}$$

$$\nabla \cdot \mathbf{v} = 0 \quad \text{in} \quad \Omega \times (0,T), \tag{2}$$

where $\mathbf{v}$, P and ρ are the fluid velocity, pressure and density, Π is the viscous term. $\Omega = [0,a] \times [0,b]$ is the computational domain, and $(0,T)$ is the time interval of the simulation. The boundary of the domain consists of solid walls, Γ_w, and the free interface, Γ_f, of the fluid. On Γ_w we assume free-slip conditions and on Γ_f - zero-value of the pressure field. Equations (1)-(2) are supplemented with the initial condition for velocity. In this study, we suppose that the fluid initially is at rest state, i.e. $\mathbf{v}(\mathbf{x},0) = 0$. Note, that WCSPH additionally requires the initial condition for density.

To compare WCSPH and ISPH we consider a partially filled container, where sloshing is excited by the following mass force

$$\mathbf{g}(\mathbf{x},t) = \begin{bmatrix} g_0 \sin\Theta + (\dot{\Theta}^2 r_x - \ddot{\Theta} r_z) \\ g_0 \cos\Theta + (\dot{\Theta}^2 r_z + \ddot{\Theta} r_x) \end{bmatrix}, \tag{3}$$

where $g_0 = 9.81$, $\Theta = A\sin(B(t - t_{ex}))$ with the beginning

A Stabilised Incompressible SPH Based on the Projection Method

Rui Xu Peter Stansby Dominique Laurence
School of Mechanical, Aerospace, and Civil Engineering
University of Manchester
Manchester, UK

***Abstract*—The stability and accuracy of three methods which enforce either a divergence-free velocity field, density invariance, or their combination are tested here through the standard Taylor-Green problem. While various approaches to incompressible SPH (ISPH) have been proposed in the past decade, the present paper is restricted to the projection method for the pressure and velocity coupling. It is shown that the divergence-free ISPH method cannot maintain stability in certain situations although it is accurate before instability sets in. The density invariant ISPH method is stable but inaccurate with random-noise like disturbances. The combined ISPH, combining advantages in divergence-free ISPH and density-invariant ISPH, can maintain accuracy and stability although at a higher computational cost. A new divergence-free ISPH approach is proposed here which maintains accuracy and stability without increasing computational cost by slightly shifting particles away from streamlines while correcting their hydrodynamic characteristics. This avoids the highly anisotropic particle spacing which eventually triggers instability. The algorithm is not conservative strictly, but tests show the accuracy is not sacrificed comparing with other projection-based methods. Importantly pressure fields are free from spurious oscillations, up to the highest Reynolds numbers tested.**

I. Introduction

In ISPH methods the incompressibility has been generally achieved by the projection method [5], [10], [14], [18]. The recent alternative approach of imposing the kinematic constraint of a constant volume for each fluid particle through non-thermodynamic pressure [7] is not considered in this paper. It is however potentially competitive computationally as shown through comparisons with WCSPH in [7]. In this work, only the projection-based ISPH methods are studied.

In 1998, Koshizuka et al. [12] presented the incompressible method, where a pressure Poisson equation was solved instead of a penalty method [11] with a source term proportional to density variation. In 1999, Cummins and Rudman [5] applied the projection method in SPH, which projects an intermediate velocity field onto a divergence-free field and a curl-free field respectively. Shao and Lo [18] used an incompressible method, similar to that in [12], to describe the free surface in dam-break flow. Colin et al. [4] proposed an improved Laplacian operator in a method similar to [7]. In 2007 and 2008, Lee et al. [13], [14] pointed out that a truly ISPH method improved the accuracy of the SPH method. In 2007, different from the projection-based ISPH in [7], the non-thermodynamic pressure is calculated to maintain the constant volume for each fluid particle. In 2007, Hu and Adams [10] proposed a stable algorithm to obtain both a velocity-divergence-free field and constant density, with an additional Poisson equation solution. By remeshing particles on a uniform grid, Chaniotis el at. [2] managed to obtain DNS simulation capability while maintaining its robustness and adaptivity. In 2008, Fang and Parriaux [8] also used a remeshing method to overcome the instability problem in the Lagrangian finite point method (LFPM). However, in the finite point method, strict conservation is not maintained as it is in ISPH methods.

In [10], it was pointed out that if only a divergence-free velocity field is enforced, density variation, or particle clustering could happen due to the spatial truncation error; moreover, this density error could accumulate during the simulation. But this problem is firstly addressed in [7] in a different context. Fang and Parriaux [8] stated that irregular particle distributions will exhibit increasing numerical errors in SPH, and for LFPM, an ill conditioned matrix in the linear system could appear. On the one hand, with the ISPH method proposed in [5] and [14], a divergence-free velocity field is enforced, but the particle spacing is not considered, which could introduce instability in the simulation, based on the study in [10] and [8]. On the other hand, in [18], the particle spacing can be well maintained, resulting from the enforcement of density invariance, but the divergence-free condition for velocity field is not enforced. To better illustrate the advantages and disadvantages of all these existing projection-based ISPH methods [5], [10], [14], [18], they are applied and examined here, based on the open-source code SPHysics [20], recently extended to account for the incompressibility. Moreover, a new method, where the particle positions are shifted slightly across streamlines, and the hydrodynamic variables at new positions are interpolated by Taylor series, is presented here. It is a new remeshing approach, but still with the advantage of meshless method.

In this paper, the error caused by particle clustering will be introduced in §2. The projection-based ISPH algorithms are introduced in §3. In §4, the finite-volume commercial package STAR-CD is introduced. Two test cases, Taylor-Green vortices and lid-driven cavity, are simulated here and defined in §5. For the first case, Taylor-Green vortices, previous projection-

An Improved Incompressible SPH Method For Wave Impact Simulations

Abbas Khayyer, Hitoshi Gotoh
Department of Urban and Environmental Engineering
Kyoto University
Kyoto, Japan

Songdong Shao
School of Engineering, Design and Technology
University of Bradford
West Yorkshire, United Kingdom

Abstract— **The paper presents an improved incompressible SPH method for refined prediction of wave impact pressure. The Incompressible SPH (ISPH) method by Shao and Lo [1] has been selected as the starting numerical tool. Three modifications have been made to enhance the pressure calculation in an ISPH-based simulation. The first modification is the application of a correction technique [2] same as that in the Corrected ISPH (CISPH) method [3]. The second modification corresponds to the derivation and application of a higher order source term of the Poisson pressure equation [4]. Finally, as for the third modification a new criterion is proposed for enhancement of free-surface assessment and removal of false zero-pressure points especially at the wall boundaries. The CISPH-HS-ASA (CISPH-Higher order Source term-Assessment of free-surface particles based on Symmetric Arrangement of non-free-surface particles) has been applied to simulate a few wave impact problems including two cases of dam break with impact [5,6] and a flip-through impact [7]**.

I. INTRODUCTION

The Smoothed Particle Hydrodynamics (SPH) method [8,9] has been proven to be a capable numerical tool for simulation of a wide variety of engineering problems including free-surface hydrodynamic flows. Despite its capability and wide range of applicability, the SPH method has a few drawbacks that may considerably affect its accuracy and performance. Such drawbacks are mainly brought about by the interpolation features of SPH, that is, local kernel-based interpolations on the basis of moving calculation points by relatively simple differential operator models. Non-exact conservation of momentum [2,3,10], lack of interpolation completeness [11] existence of spurious pressure fluctuations [12,13] are among the major drawbacks associated with the SPH method.

The accuracy and performance of particle methods have been enhanced by applying corrective techniques correcting the kernel function itself and/or its gradient to improve the completeness of kernel interpolants and/or to enhance the conservation of momentum, as in case of Corrected SPH (CSPH [2]) or Corrected Incompressible SPH (CISPH [3]) methods. Recently, Khayyer and Gotoh [10] improved the performance of MPS (Moving Particle Semi-implicit) method [14] by deriving an anti-symmetric pressure gradient term which exactly guarantees the conservation of both linear and angular momentum.

To resolve the problem of artificial pressure fluctuation in their Weakly Compressible SPH (WCSPH) calculation, Colagrossi and Landrini [13] re-initialized the density field at distinctive calculation time steps by applying a first-order accurate interpolation scheme via the employment of a moving-least-square kernel approximation. A more accurate interpolation scheme enhances the consistency of mass-density-occupied area and thus results in a less fluctuating and more accurate source term for pressure equation (equation of state). Therefore, a less fluctuating and more accurate pressure field would be obtained. On the contrary to the WCSPH method, the ISPH method [1] employs a Poisson Pressure Equation (PPE) in which the pressure is a direct function of the time rate of change of density rather than the density itself. Thus, to achieve a less fluctuating and more accurate pressure field by the ISPH method, a more accurate source term of PPE based on a higher order calculation of the time rate of change of density should be derived. In this paper, we derive and apply a higher order source term based on a more accurate differentiation. The CISPH method modified by the higher order source term would be referred to as CISPH-HS (CISPH-Higher order Source term).

Another important issue which affects the pressure calculation in an ISPH-based simulation is related to the criteria/criterion for assessment of the free-surface particles. In most ISPH calculations, the free-surface particles have been assessed by a simple criterion based on the fact that calculated density sharply drops at the free-surface. This criterion might be regarded as a necessary condition but not a sufficient one. In this paper, we propose a simple auxiliary criterion for an efficient assessment of free-surface particles in an ISPH-based simulation. The enhanced performances of both CISPH-HS and CISPH-HS-ASA (CISPH-HS-Assessment of free-surface particles based on Symmetric Arrangement of non-free-surface particles) will be shown by simulating two cases of dam break with impact [5,6] and a flip through impact [7].

II. INCOMPRESSIBLE SPH (ISPH) METHOD

In this section, the ISPH method is briefly explained. Detailed descriptions are provided by Shao and Lo [1] or Gotoh et al.

SPH Formulation with Lagrangian Eulerian adaptive kernel

Jean-Luc LACOME
Livermore Software Technology Corp
Livermore, California, CA
jll@lstc.com

Jérôme LIMIDO
IMPETUS Afea
Grenade sur Garonne, France
jerome@impetus-afea.com

Christine ESPINOSA
Université de Toulouse, ISAE
Toulouse, France
christine.espinosa@isae.fr

***Abstract*—This study is devoted to numerical instabilities applications for the SPH method applied to solid mechanics. A new approach that couple Lagrangian kernel SPH and Eulerian kernel SPH is proposed and evaluated. The purpose is to take advantage of both types of kernel. An adaptive kernel SPH formulation is also proposed and evaluated. This study is carried out in the framework of the LS-DYNA® finite element code.**

I. INTRODUCTION

SPH is a numerical method that can be apply in many fields. It has been used successfully in many applications: gas, fluids, solids [1]. However, in its original definition, the SPH method suffers from several drawbacks as described in details in [2, 3].

We are focus in this paper on numerical instabilities problems for solid mechanics SPH.

"Standard" SPH methods are based on an Eulerian kernel. In this case, the support remains constant and the neighbors search is carried out at each time step. This allows large deformations problems but the well-known tensile instability problem often limits the field of applications of the models [2]. As proved by Bonet and Belytschko [2; 3], tensile instability problem can be eliminated by using a Lagrangian kernel formulation.

In the Lagrangian kernel case, the neighbors list remains the same throughout the calculation and the kernel support becomes deformed. This approach eliminates the tensile instability problems but the treatment of large deformations becomes limited [2]. In order to solve this problem, Vidal et al. [3] proposed a very interesting approach based on a formulation with updated Lagrangian kernel. They showed that the reference state update and the neighbor list update allow large deformations modeling. Nevertheless, when this reference state is too frequently actualized, numerical instabilities appeared [3].

In the following, we propose a new approach based on Eulerian and Lagrangian kernel SPH coupling. First, Eulerian kernel SPH and Lagrangian kernel SPH are compared through classical tests cases Then, we will show that it is possible to couple Lagrangian kernel particles with Eulerian kernel particles. A validation based on a finite element reference solution is presented. Finally, we will present SPH with Lagrangian Eulerian adaptive kernel. In an initial state, all the particles have a Lagrangian kernel and when necessary some Lagrangian kernel particles are converted into Eulerian kernel particles.

II. EULERIAN KERNEL VS LAGRANGIAN KERNEL SPH

A. Eulerian and Lagrangian kernel SPH formulations

We quickly recall the momentum conservation equation in "standard" SPH formulation that uses Eulerian kernel in order to compare it to that which uses a less classical Lagrangian kernel.

We limit this study to the formulation known as "normalized" introduced by Randles and Libersky [5]. This formulation makes the SPH approximation first order consistent by introducing a correction applied to the SPH gradient (see equation (4) and (5)).

Let us define some notation. The initial state will also serve as the reference state. The motion is described by (1):

$$\mathbf{x} = \varphi(\mathbf{X}, t) \tag{1}$$

where x are the spatial or Eulerian coordinates and X the material or Lagrangian coordinates.

The displacement u is thus given by (2):

$$\mathbf{u}(\mathbf{X}, t) = \mathbf{x} - \mathbf{X} \tag{2}$$

1) Eulerian Kernel SPH formulation

Artificial stress effects on structural dynamics using SPH

Paul H.L. Groenenboom
ESI Group Netherlands
Rotterdamseweg 183 C, 2629 HD Delft, Netherlands
pgr@esi-group.com

Libor Lobovský
Department of Mechanics
Faculty of Applied Sciences, University of West Bohemia
Univerzitní 22, 30614 Plzeň, Czech Republic
lobo@kme.zcu.cz

***Abstract*—The tensile instability of the standard SPH formulation inhibits quasi-static structural mechanics to be simulated reliably. In 2001, Gray et al. [1] proposed adding artificial stresses as derived from a dispersion relation for elastic solids. It was demonstrated that this method effectively removes the instability. Stable SPH simulations for various elastic structures may be conducted and the computed frequency of a vibrating plate was found to agree with the analytical result. Due to inconsistencies related to the computed strength of an elastic structure loaded by a fluid as discussed in another contribution [2], the current authors investigated the strength of various elastic and elasto-plastic structures. It was found that the results strongly depend on the number of particles and the smoothing length used. In general the effective strength of the SPH model is found to be significantly less than corresponding finite element method (FEM) simulations. A strength correction method will be proposed and the SPH results with and without this correction will be compared to finite element results for various test cases. In many, but not all, cases the SPH simulations using this correction yield nearly identical results as those of the FE simulations. It will be demonstrated that the artificial stress method allows performing SPH simulations of elasto-plastic necking phenomena for three-dimensional test pieces. The results for a mixed FE-SPH model will be compared to those for a pure FE model.**

I. INTRODUCTION

For structural dynamics simulations using SPH a severe problem is the tensile instability associated with unphysical clustering of particles. In many cases artificial fracture will follow. This problem may be detrimental in elasto-dynamic simulations at relatively low speed. Several possible solutions to this problem have been proposed in the literature. A solution that is suitable for most existing SPH implementations is to add artificial stresses to the structural stresses in the SPH momentum equations [1]. The formulation and the parameters for these stresses were found from a dispersion analysis for propagation of elastic waves. This method has been shown to remove the tensile instability, at least for simulations in two dimensions but whether the artificial stress correction provides sufficiently accurate results has not properly been investigated.

In many cases of fluid-structure interaction the deformation of the structures remains limited and a coupled FE-SPH solution may be the most attractive solution. Advantages of FEM over SPH are that the structural displacements are defined at the interface and that the FEM provides a reliable and cost-effective solution. Nevertheless, it is of interest to investigate whether SPH may also be used to simulate the structural dynamics of a coupled FSI problem.

One of the few cases in which SPH is applied to a fluid and a structure interacting with each other has been published by Antoci et al. [3], who studied the FSI for the case of water escaping from a reservoir through an elastic gate. In this reference reasonable agreement with the experimental results has been found. Similar results were obtained by [4] and [5]. All these simulations employed the artificial stress correction as proposed by [1]. As will be shown in another contribution [2], simulation using a combination of SPH for the fluid and finite elements for the structural part, using a well-proven FSI algorithm, leads to different results. Further investigations of this model suggested that SPH simulations of the structural dynamics may not be reliable and this will be the main topic of this paper.

For the current investigation, the method proposed in [1] has been extended to a three dimensional formulation and an optional correction is proposed. This correction is aimed to mitigate the effect of the change in the effective material strength that occurs when the artificial stress is added. The results of computational simulations for elastic structures in bending and tension are presented.

It will be demonstrated that the artificial stress method allows performing SPH simulations of elasto-plastic necking phenomena for three-dimensional test pieces.

II. NUMERICAL METHODS EMPLOYED

The current studies were conducted with two independent software codes. The first one, LOBO-SPH, is a special SPH code developed at the University of West Bohemia that allows for simultaneous solution of fluid mechanics of a nearly-incompressible fluid and of structural dynamics of elastic materials as well as a solution for the interaction between the fluid and the structural parts through the contact algorithm. More details on definition of the fluid-structure interaction and implementation of this code can be found in [2,4].

Modelling extreme wave loading of offshore structures

J.C. Campbell and R. Vignjevic
Crashworthiness, Impact and Structural Mechanics Group,
School of Engineering, Cranfield University,
Bedfordshire, MK43 0AL. UK.
j.campbell@cranfield.ac.uk

Abstract— **Structural response to water impact is important for several areas, including the aerospace and marine industries. Offshore structures are subject to extreme wave impact and green water loading. The goal is a reliable technique for predicting the structural response to extreme water loading. This is a complex problem involving the interaction of non-linear fluid behaviour (breaking waves, fluid impact) with non-linear structural behaviour (large deformations, contact, material plasticity and damage). The explicit finite element method is the established method for simulating the crash and impact response of structures and is implemented in numerous commercial and research codes. The difficulty of modelling this type of fluid-structure interaction is in correctly representing the fluid interaction with the structure so that the loading of the structure is correct. This paper will discuss the coupled FE/SPH approach for modelling the response of ships and offshore structures to extreme wave loading. Previous work at Cranfield has demonstrated the FE-SPH approach for impact of aircraft structures on water. This work has extended this to wave loading of floating structures, which requires significantly more complex initial conditions for the water component.**

I. INTRODUCTION

Extreme waves are a natural phenomenon that can cause significant damage to ships and offshore structures, in particular through green-water loading. The term green water indicates the flow of water, not just spray, over the deck of an offshore structure. This type of fluid-structure interaction is a potentially highly complex non-linear problem that can combine several features:

- The behaviour of extreme waves, including breaking,
- The response of a floating structure to waves,
- The interaction of the water with a structure,
- The response of the structure to the fluid loading including large deformations and non-linear material behaviour (including plasticity, damage and failure).

The absence of reliable theoretical models for this behaviour means that design must be based on empirical considerations and model test data.

The objective of this research is to develop and demonstrate an analysis tool capable of predicting the response of structures to extreme wave loading. To achieve this objective the tool must be capable of representing the important feature of the problem. The explicit finite element (FE) method is solidly established for the simulation of structural response and is well suited to representing the types of structural deformation that can occur in water impact. However due to the well known problems with mesh-tangling this method is not appropriate for modelling the fluid. In order to simulate the structural response to extreme wave loading the FE method must be coupled with another method appropriate for modelling the fluid behaviour. In this work we have used the Smoothed Particle Hydrodynamics (SPH) method to model the fluid behaviour as it can be easily and reliably coupled with the non-linear FE method [1].

The application of SPH to water waves and related free-surface hydrodynamics problems was started by Monaghan [2], who performed two-dimensional simulations of a dam break problem and wave propagation onto a shallow beach. More recently SPH simulations have been further compared with published experimental results, an example being Scott Russell's wave generator [3], with the SPH method in agreement with the experimental results. In these simulations use an artificial equation of state to produce a quasi-incompressible fluid, the approach used in this work. SPH has also been used for wave mechanics with exact enforcement of incompressibility [4]. This uses an implicit pressure update that allows a larger time step but requires more computational work per time step. Recently SPH methods have been successfully applied to 2D simulations of green water overtopping [5] and wave overtopping [6] using rigid representations of the impacted structure.

Other approaches for wave impact simulations typically use Eulerian or ALE methods to solve the Navier-Stokes equations. For example, Buchner, Kleefsman and co-workers [7,8] have developed an Eulerian code and applied it to wave run-up and green water loading on Floating Production Storage and Offloading (FPSO) systems. Their results show good agreement with experiment for structural loads, however the structures are represented by rigid shapes.

Previous work at Cranfield [1,9-11] has developed and demonstrated a contact algorithm for coupling meshless and finite element discretisations, allowing complex interaction in three dimensional simulations. A primary application of this method was the simulation of helicopter crash on water and it has been demonstrated for the impact of rigid and deformable structures on water [11]. Extend this approach to wave loading of offshore structures required development of appropriate initial and boundary conditions for the fluid component. All

Numerical corrections based on diffusive terms for SPH schemes

M. Antuono
INSEAN, Italian Ship Model Basin,
Rome (Italy)
m.antuono@insean.it

A. Colagrossi
INSEAN, Italian Ship Model Basin,
Rome (Italy)
CESOS, Centre of Excellence for Ship and Ocean Structures, NTNU,
Trondheim (Norway)
a.colagrossi@insean.it

D. Molteni
Dipartimento di Fisica e Tecnologie Relative,
University of Palermo,
Palermo (Italy)
molteni@unipa.it

***Abstract*— Extending the authors' work on the use of artificial diffusive terms to improve the Smoothed Particles Hydrodynamics (SPH) schemes, a new system of equations has been defined which contains diffusive terms in both the continuity and energy equations. The novel discrete equations, at the leading order, coincide with a standard SPH scheme with artificial viscosity. A proper state equation is used to associate the internal energy variation to the pressure field and to increase the speed of sound when strong deformations/compressions of the fluid occur. The increase of the sound speed is associated to the shortening of the time integration step and, therefore, allows a greater accuracy during both breaking and impact events. Moreover, the diffusive term inside the continuity equation allows reducing the high frequency numerical acoustic noise and smoothing the pressure field. The model has been tested using different free surface flows clearly showing to be robust, efficient and accurate. An in-depth analysis of the CPU time cost and comparisons with the standard SPH scheme is provided.**

I. Introduction

The Smoothed Particle Hydrodynamics (SPH) method is very appropriate for simulating complex fluid dynamics and is frequently used when the problem shows strong free surface dynamics. As any numerical approach it has its drawbacks. Here we address two problems: one is the correctness of the pressure values for strong impacts and the second is the flickering of pressure profiles that may occur also in not particularly violent flows. The first item was also addressed by Molteni et al. [1]. The essence of the improvement for the first item is that the sound speed is allowed to vary *locally*, that is only for the particles involved in strong collision. This means that we make more un-compressible the fluid where and when it is hitting against some obstacle. This procedure requires the use of the energy equation. So we define an ad-hoc thermal energy with the exclusive aim to change the sound speed as compression occur. This energy does not necessarily correspond to the real physical temperature of the fluid, since it has effects only on the numerical accuracy. The second item is related to the fact that, in general, the flow speeds look good, but, checking the distribution of pressure, large random pressure oscillations are present due to numerical high frequency acoustic signals. In the simulations of violent liquid-solid impacts the pressure value is very important and its measure turns out to be critical for practical applications. Such kind of impact problems are strongly time dependent and the analytical pressure solution are complex and limited to very simple geometries (see e.g. [2]). So this problem is not deeply investigated in the SPH literature. In [1] the authors suggested a filtering of the density with a MLS integral interpolation as a way to reduce the numerical noise in the pressure evaluation. That procedure gives good results, but, for long time simulations, it does not properly conserves the total volume of the particles' system since the hydrostatic component has been improperly filtered [3]. Following the idea of exploiting a smoothing process, we insert a simple numerical diffusion term [4]. Here we examine the effects of diffusion acting both on the density and on the thermal energy. In favor of this approach we remind that, in the framework of SPH, the standard diffusion algorithms, see Brookshaw [5], is conservative and therefore it preserves the total amount of the diffused quantity: mass, thermal energy. Obviously the diffusion coefficient has to vanish as the numerical accuracy increases in order to avoid un-physical effects and to recover the consistency of the discrete equations. The analyzed problems show clear evidence of the improvements. Particular attention has also be paid on the time integration scheme adopted which play an important role both in pressure evaluation as well as in the computation costs.

II. Governing Equations

Let us consider a non-viscous, weakly compressible fluid. The governing equations are:

$$\begin{cases} \frac{D\rho}{Dt} = -\rho\,\nabla\cdot\mathbf{u}\,; & \dfrac{D\mathbf{u}}{Dt} = -\dfrac{\nabla p}{\rho} + \mathbf{f} \\ p = F(\rho, e)\,; & \dfrac{De}{Dt} = -\dfrac{p}{\rho}\,\nabla\cdot\mathbf{u} \end{cases} \tag{1}$$

where $\mathbf{u}$ is the fluid velocity, ρ is the density, p the pressure, e the specific internal energy and $\mathbf{f}$ represents the mass force

Low Mach number numerical schemes for the SPH-ALE method. Application in free surface flows in Pelton turbines.

Jean-Christophe Marongiu, Francis Leboeuf, Joëlle Caro
Laboratory of Fluid Mechanics and Acoustics
Ecole Centrale de Lyon, University of Lyon
Ecully, France
jean-christophe.marongiu@ec-lyon.fr

Etienne Parkinson.
Hydraulic research department
ANDRITZ Hydro
Vevey, Switzerland

***Abstract*—SPH-ALE is a hybrid numerical method in which an *Arbitrary Lagrange Euler* (ALE) description of fluid flows is used together with the mesh-less numerical method Smoothed Particle Hydrodynamics. This method can be closely connected to the finite volume approach. Notably, it can take benefit from the use of upwind schemes (Riemann solvers) to improve stability, accuracy and implementation of boundary conditions. However the behaviour of upwind schemes in the limit of low Mach number flows is known to lead to pressure oscillations and to increased numerical diffusion. This work presents a preconditionned linearized Riemann solver that can perform properly in the limit of low Mach number flows. The numerical method is then used to simulate free surface flows in Pelton hydraulic turbines.**

I. Introduction

SPH-ALE is a hybrid numerical method in which an *Arbitrary Lagrange Euler* (ALE) description of fluid flows is used together with the mesh-less numerical method Smoothed Particle Hydrodynamics. This method can be closely connected to finite volumes approach. It is then possible to adapt some common techniques like upwind schemes to remedy some of the well known drawbacks of SPH like stability and accuracy (see [5]).

However in practice, the introduction of upwind schemes for the simulation of hydraulic flows produces an excessive amount of numerical diffusion, which can penalize the quality of simulations. This is a known fact in mesh-based techniques when these schemes are used to model low Mach number flows. Among the SPH practitionners community, it is usual to tune the speed of sound in water so that the Mach number remains below 0.1 in the computationnal domain, which is clearly in the range where upwind schemes show some limitations. In this work, it is proposed to adapt preconditioning techniques (Turkel, [9]) to the SPH-ALE framework. Following the work from Gallouet et al. [2] further developed by Viozat [13], a new simple linear preconditioned Riemann solver is written for the Tait equation of state. This scheme is first benchmarked on simple cases. It is then used on real application cases typical of free surface flows in Pelton hydraulic turbines. In order to speed-up these simulations, a symmetry condition adapted for SPH has been developed and is presented.

II. Preconditioned SPH-ALE method

The SPH-ALE method was mainly developed by Vila [12]. It is based on the standard SPH method but differs in the way flow is described. The SPH method is often presented as a whole, but it may be of high value to consider the core of the method (its mesh-less numerical scheme) separately from the other aspects (lagrangian flow description, particle interactions modelling).

A. Principles

The SPH-ALE adopts a dual description called *Aritrary Lagrange Euler*. It consists in writing the conservation laws, here for an inviscid flow, in a moving frame of reference, whose velocity v_0 can be chosen arbitrarily. We thus consider the following system of conservation laws written for a control volume Ω moving with speed v_0:

$$\left.\frac{d}{dt}\right|_{v_0}\int_\Omega \Phi\, d\Omega + \int_S \Phi(v-v_0)\cdot n\, dS = \int_S Q_S\cdot n\, dS + \int_\Omega Q_V\, d\Omega \tag{1}$$

Φ is the vector of conservative variables. S is the boundary of Ω, n the unit outward normal vector, Q_S and Q_V the surface and volume source terms, respectively. $\left.\frac{d}{dt}\right|_{v_0}$ stands for a time derivative along the trajectory of the control volume. Considering that the surface source term can be reduced to the pressure term, (1) becomes:

$$L_{\vec{v}_0}(\Phi) + div(F_E(\Phi) - v_0\,\Phi) = Q_V \tag{2}$$

where $L_{\vec{v}_0}(\Phi)$ is the transport operator associated with the transport field v_0 and F_E is the flux vector of the Euler

Simulations of complex hydro-elastic problems using the parallel SPH model SPH-Flow

G. Oger, E. Jacquin,P.-M. Guilcher
HydrOcean
Nantes, France
guillaume.oger@hydrocean.fr

L. Brosset
Liquid Motion dept.
Gaztransport&Technigaz
Saint-Rémy-les-Chevreuse, France

J.-B. Deuff, D. Le Touzé, B. Alessandrini
Laboratoire de Mécanique des Fluides
Ecole Centrale Nantes
Nantes, France

***Abstract*— Recent researches on sloshing-induced impact loads outlined the importance of hydro-elastic effects on extreme local pressures observed in LNG tankers. Such extreme loads may severely threaten the tank integrity, and namely its isolation components, implying the need for highly efficient predicting tools. Due to the local aspect of these pressure peaks on the one hand, and the difficulty inherent to the simulation of coupled fluid/structure interactions on the other hand, predicting accurately such cases remains a challenging topic. As an international specialist in LNG tanker components, GazTransport&Technigaz (GTT) company developed specific researches and development in this field. One of the objectives of GTT is to own some efficient fluid/structure interaction models for its applications. Among all the developments made, a PhD Thesis dedicated to a SPH model in the context of fluid/structure interactions has been achieved with Ecole Centrale Nantes. Indeed, SPH method offers some very interesting features in this particular topic. Its Lagrangian formulation strongly limits any diffusion in the calculated fields, with high conservation properties for jets impacting walls for instance, resulting in a sharp pressure peak capture. Another interesting property of SPH consists in its ability to easily approach complex multi-physics, and namely to obtain the fluid/structure coupling in a very natural way. Moreover, the compressible aspect of SPH stands for an additional useful advantage for modelling problems involving high dynamics, for which natural compressible effects may occur within the fluid medium. The development of this specific SPH model is now continued by the company HydrOcean, which is responsible for the necessary enhancements, together with the parallelization of this original model.**

I. INTRODUCTION

The multi-scale aspect of sloshing impacts in tanks of LNG carriers has to be taken into consideration. While global flows (ruled by Froude number) are well captured using various numerical tools such as VOF methods, local scale effect in the impact area cannot be extracted correctly from those results. Looking more carefully at these local effects, lots of different physical phenomena can be observed during the impact, among which gas escape, rapid change of the liquid momentum to avoid the obstacle, compression of the partially or completely entrapped gas fraction, possible condensation, and fluid-structure interactions [1][2].For the time being, few software models are able to simulate correctly one or several of these phenomena in very simplified impact conditions, but no code can take into account simultaneously all the phenomena that can be involved in a single real impact, even for simplified conditions. It is worst to imagine a simulation dedicated to predict all of these complex local effects within a global flow simulation. Such possibility would result in obtaining a sensibility to initial conditions as observed in the sloshing model tests. As a consequence, as for the model tests, very long simulations would be needed in order to obtain the thousands of impacts necessary for a statistical analysis, leading to longer and much more expensive studies than tests (at least in a first stage).Nevertheless, local simulations of impact in simplified - even very academic- conditions can be considered as a good mean of better understanding the weight of the various physical phenomena involved for different types of impacts, and hence can help in evaluating scaling factors that are needed for a good use of model test results.

Among the various numerical methods available, SPH seems to display some very interesting advantages for simulating local impacts. Thanks to its compressible and explicit features, some small time step should be imposed, that is of peculiar concordance with the short typical duration of impacts. Another advantageous feature of SPH resides in its natural multi-fluid and multi-physic predisposition, allowing to model biphasic problems with low density ratio (gas to liquid), possibly with the occurrence of fluid/structure interactions. Our mid-term objective is to have at our disposal a SPH compressible bi-fluid/structure solver in order to be able to simulate all phenomena involved in sloshing model tests (no change of phase for these tests). Some specific developments are presented here, and represent an important stage to reach this objective: the fluid-structure coupling is now included in the SPH-Flow platform enabling to capture the hydro-elasticity effects during sloshing impacts.

In this paper, the theoretical model of the SPH solver SPH-Flow is presented, and its applications to various academic and industrial test cases (free vibration of a beam, wave propagation through a heterogeneous material, compression of a composite structure) are discussed and compared with

Simulating a Real Dam Spillway Flow with 3-D SPH

Eun-Sug Lee, Damien Violeau, Réza Issa
Laboratoire d'Hydraulique Saint-Venant
Université Paris-Est, laboratoire commun EDF R&D – CETMEF – Ecole des Ponts
Chatou, France
damien.violeau@edf.fr

Stéphane Ploix, Raphaël Marc
Département SINETICS
EDF R&D
Clamart, France

***Abstract* — The SPH method is nowadays widely used for modelling complex, rapid and turbulent free-surface flows, including fluid-structure interactions. EDF R&D has already applied SPH to the simulation of real two-dimensional flows, such as the behaviour of an oil spill near an oil containment boom, using the Spartacus-2D software. Still recently, using this method for three-dimensional computations was very limited by the significant computing requirements of this numerical approach. However, the progress in high-performance computers as well as modern post-processing tools make it possible today within an industrial framework. Indeed, recent improvements in massive parallelization allow the management of several tens million particles. EDF R&D carried out simulations of the flow over the Goulours dam spillway, a prototype of studies yesterday restricted to physical modellers. This work shows the feasibility of this approach through a qualitative comparison of the results against physical model measurements.**

I. MOTIVATION

A. *Introduction*

Spillways are river works designed in order to control the flow rate of a river through a dam under all circumstances, including floods. Depending on the river topography and the dam geometry, many types of spillways exist, among which ski-jumps and PK (Piano Key) weirs are widely used all around the World (see Fig. 1). The design of such waterworks is essential for various reasons: 1) the spillway must deliver a maximum flow rate compatible with extreme floods; 2) the efforts experienced by the structure must not exceed a maximum threshold; 3) it is expected that a maximum rate of energy is dissipated by turbulence while the water is flowing down, in order to avoid extensive erosion at the toe of the dam.

The complicated features of those flows make the numerical simulation of dam spillway flows difficult, and most of them are still studied from laboratory models (see Fig. 2). However, recent advances in Computational Fluid Dynamics now allow the modellers to perform 3-dimensional (3-D) free-surface flows including complex Physics (*e.g.* turbulence and multiphase flows), and engineers can now handle real spillway simulations with mesh-based methods like Volume-Of-Fluids (VOF). However, Lagrangian methods appear to be more appropriate to that kind of studies; at EDF R&D, our aim is to use SPH to make a 3-D industrial modelling tool capable of addressing all kinds of geometries and complex hydro-dynamics for the purpose of spillway design.

Figure 1. The Goulours dam spillways. The old ski-jump-type spillway is on the right, whereas the new PK weir appears on the left (under construction).

B. *The Goulours dam*

The waterwork considered in the present study is a concrete dam build on the Lauze river (France). It was completed in 1946 at Goulours (Ariège, Midi-Pyrénées), and is a curved gravity dam for the use of hydroelectricity production. A view of the dam site is shown in Fig. 1 and its dimensions are briefly given in Table 1. Originally, the Goulours dam was only equipped with a ski-jump spillway; however, this spillway was later proved to be not sufficient to deal with highest floods. Hence, a PK weir was installed at the end of October 2006 as an additional floodwater evacuation system.

TABLE I. CHARACTERISTICS OF THE DAM

Type	**Value**
Height	23.0 m
Crest length	71.0 m
Crest thickness	2.0 m
Base thickness	6.0 m
Dam volume	2,500 m^3
Mean flow rate	90 $m^3 s^{-1}$

SPH for free-surface flow around a heaving oscillating cylinder using variable particle mass distributions: 2-D investigations

Pourya Omidvar, Peter K. Stansby and Benedict D. Rogers,
School of Mech, Aero & Civil Engineering, The University of Manchester, Manchester, M60 1QD, UK.
pourya.omidvar@postgrad.manchester.ac.uk,-p.k.stansby@manchester.ac.uk- benedict.rogers@manchester.ac.uk

Abstract—**In this work Smoothed Particle Hydrodynamics (SPH) is used to investigate a heaving cylinder in initially still water in a two-dimensional water channel by using code SPHysics [1] enhanced using a Riemann solver [2]. A comparison between our SPH results and the experimental data of Yu and Ursell [3] for a single heaving cylinder gives good agreement. Capturing the entire three dimensional flow field however would require a simulation with an extremely large number of particles. To circumvent this issue, this work employs variable mass distribution around the cylinder. Using variable mass distribution in a pre-selected area avoids the need for both variable kernel size and particle refinement schemes. Here, we show how this technique works in still water, and then for the heaving cylinder problem. By comparing the ratio of the wave amplitude to the cylinder amplitude the variable mass distribution gives comparable results to using a uniformly fine particle size. Using variable mass distribution in the vicinity around the cylinder also leads to a speedup of nearly 200%. This work is a part of a study examining extreme wave loading on wave energy devices using SPH.**

I. INTRODUCTION

Generating energy from surface water waves with floating/heaving bodies is attracting considerable attention as a potential way to overcome the growing concerns of future energy requirements. However, energy devices must be able to operate under extreme condition design and performance of such devices where the flow can be particularly violent making the simulation very difficult. For the case of multiple heaving wave energy devices the flow field is potentially very complex since each device is moving and the hydrodynamics can involve complex interactions of wave breaking, reflection and diffraction processes. At present, there is only a limited body of work investigating floating bodies in extreme waves, most of which uses either physical experimentation or sea trials. Standard approaches such as linear and second order wave diffraction theory also do not represent highly non-linear effects associated with extreme waves.

This work is a part of a joint project between the Universities of Plymouth, Manchester, Oxford and Manchester Metropolitan funded by EPSRC looking at the suitability of different computational fluid dynamics (CFD) schemes to model heaving wave energy devices and to understand their behaviour in real seas. Simulating free-surface flow with most Eulerian CFD methods is potentially very difficult as explicit treatment of the free surface is required. On the other hand, one of the main advantages of the smoothed particle hydrodynamics (SPH) method is to deal with such complex problems where no treatment of the free surface is needed, which combined with its Lagrangian nature in order to simulate highly nonlinear and potentially violent flows. At the University of Manchester, we are using the SPH method to simulate wave propagation and loading on the wave energy device in a water channel.

In this paper, following the work of Vila [2], Guilcher *et al.* [4], Rogers *et al.* [5] and Omidvar *et al.* [6], first, a brief introduction to numerical scheme will be presented. Then, SPH results for surface waves generated by a heaving cylinder motion will be compared with the experimental data of Yu and Ursell [3]. To compute the free-surface motion efficiently, variable mass distribution is employed around the cylinder where surface waves are generated paying careful attention to avoid tensile instability with the interpolating kernel function. Finally, the computational advantage of implementing variable mass distribution of particles will be demonstrated.

II. EXPERIMENTAL DATA

The experiment data of Yu and Ursell [3], for surface waves generated by a heaving cylinder motion, is used in order to validate the SPH results. In their paper, first, they presented the theory behind this problem where a half submerged cylinder is forced to oscillate vertically in still water and in a channel of infinite length. They derived the theoretical formulations to obtain the surface evaluation and force coefficient of a heaving cylinder. Then, they preformed the measurements in a channel of 30*m* in two-dimensional and symmetrical about the vertical plane through the axis of the cylinder. The comparison between the theoretical and experimental amplitude ratio, which is

$$R_A = \frac{\text{Wave amplitude at infinity}}{\text{amplitude of motion of cylinder}}, \quad (1)$$

Levee Breaching with GPU-SPHysics Code

Robert A. Dalrymple
Department of Civil Engineering
Johns Hopkins University
3400 No. Charles St., Baltimore, MD 21204
rad@jhu.edu

Alexis Hérault
Sezione di Catania, Istituto Nazionale di Geofisica e Vulanologia
piazza Roma
2 95123 Catania, Italy
herault@ct.ingv.it

***Abstract*—A GPU-based Smoothed Particle Hydrodynamics model is used to model the water flooding associated with various types of levee failures. The failures include instantaneous flood wall section failure, a slowly toppling wall, and a dropping wall. The intent of the paper is to illustrate the complex non-hydrostatic flows associated with the levee failures, and their resulting impact on the nearby structures.**

I. Introduction

The failure of levees around the City of New Orleans during Hurricane Katrina turned a bad hurricane into a major urban catastrophe. Although levees would have been overtopped in the eastern portion of the city due to the high storm surge during the storm (leading to local flooding), the major storm damage was due to the levee failures along the Industrial Canal, which bisects the city into eastern and western portions, and the 17th Street and the London Avenue Canals.

The floodwall failures were due to geotechnical failure of the wall foundations, resulting in a horizontal sliding and the overturning of the wall at the 17th Street Canal or simply an overturning as at the Industrial Canal [1]. In many locations, the force of the water was strong enough to move structures from foundations–such as would occur during a tsunami. In the analyses post-storm, the time at which flooding occurred or the extent of flooding by a single breach was important. This flooding of course is dependent on the nature and timing of the breach and the nature of the land over which the flooding occurred.

II. Modeling

A. Flood Modeling

The numerical modeling of floods in rivers has been underway for a long time. However, most of these models are based on solving the shallow water equations or the St. Venant equations, with the underlying assumptions that the flow is hydrostatic, or nearly so. Examples of such models are HEC-RAS [2]. However, there have been very few models that include non-hydrostatic flows and deal with dam breaks and levee breaches. Hesselink et al. [3] examined the historical flooding of a Dutch polder using a two-dimensional hydrostatic flow model. Ying et al. [4] use similar equations for dam and levee breaches along with riverine modeling. Jaffe and Sanders [5] examine the use of engineered breaches as a way to reduce riverine flooding, by diverting water in a breach-like manner into a designed storage area.

Fig. 1. Flow-related damage immediately landward of the 17th Street Canal breach site. In addition to the structural damage, trees were uprooted.

Satter et al. [6] examined via a 1:50 scale hydraulic model of the 17th Street Canal breach in New Orleans various closure schemes for breaches. For example rather than dumping sand bags in the breach site, where the velocities are the highest, they recommended a variety of other options that involved wider ”coffer dams” based on the existing structures in the area.

This paper uses the Smoothed Particle Hydrodynamics methology to examine the dramatic flow near a breach site, where the flow is non-hydrostatic and is dependent on the nature of the breach. The breaches will be due to the failure of a section of the floodwall. Several different breaching mechanisms are examined: instantaneous section failure, basically a 3-D dam break problem; a falling wall section, where the speed of falling is controlled, and a downward moving vertical wall section.

III. GPU-SPHysics

GPU-SPHysics is a Smoothed Particle Hydrodynamics model, programmed in CUDA and running on Nvidia graphics cards (GPUs). It was developed by Herault and presented at the Third SPHERIC meeting in Lausanne [7]. The formulation follows the open-source code SPHysics ([8], http://wiki.manchester.ac.uk/sphysics, which

Coastal flow simulation using SPH: Wave overtopping on an impermeable coastal structure

E. Didier[1,2] , M. G. Neves[1]
[1]Harbours and Maritime Structures Division Hydraulics and Environment Department
Laboratório Nacional de Engenharia Civil – LNEC, Lisbon, Portugal
edidier@lnec.pt; gneves@lnec.pt
[2]Marine and Environmental Technology Center – MARETEC
Instituto Superior Técnico – IST, Lisbon, Portugal

***Abstract*— Wave overtopping is a violent natural event that involves highly complex phenomenon such as large deformation of free surface, turbulence and eddy vortices, strong interaction between the wave and the structure. Models based on Smoothed Particle Hydrodynamics (SPH), that used a mesh-free technique, are an option to address wave overtopping and other phenomena involved on the interaction between waves and coastal structures. In the present paper, SPHysics model is validated and applied for wave propagation and wave overtopping of an impermeable seawall. Validation and convergence study is carried out considering several parameters such as the initial particle density and the ε_{XSPH} parameter. Free surface elevation in several gauges and overtopping discharge over the structure are analyzed and compared to experimental data and other numerical results. A very satisfactory agreement is obtained with experimental measurements. Finally, the numerical model is applied for modelling wave propagation with breaking and overtopping of an impermeable sea wall coastal defence structure, a common structure employed at the Portuguese coast. Numerical results are compared with experimental data from model scale tests carried at the National Civil Engineering Laboratory (LNEC). Good agreement is obtained for both free surface elevation and overtopping discharge over the structure.**

I. INTRODUCTION

Sea walls are structures that allow the protection of coastal areas from the wave attack. In the project of those structures, wave-structure interaction study should be made to define the viability and efficiency of the structure, namely the overtopping discharge and the forces applied on the structure. Wave-structure interaction generates very complex phenomena involving nonlinear processes, like wave propagation and transformation, run-up, wave breaking, and overtopping. Coastal structures could have different structural characteristics: could be impermeable or porous structures, composed by artificial blocs, be an arc crown wall structures, etc.

Numerical models, more or less complexes depending on the approach and on the physical assumptions, allow simulating near shore transformation and propagation of waves.

The models based on the nonlinear Boussinesq equations, such as COULWAVE [1], give good predictions comparing with field data and laboratory physical modelling. However, it does not model the breaking wave and highly nonlinear processes that occur between waves and coastal structures, such as breaking and overtopping.

Some numerical models allow simulating these very complexes phenomenon. Those models are generally based on fluid dynamic equations, i.e. the Navier-stokes equations, and developed using an Eulerian approach. Numerical simulation of free surface flows is treated using the Volume of Fluid (VOF) approach, such as the Reynolds Average Navier-Stokes (RANS) model COBRAS-UC [2]. However the recent advances on Smoothed Particle Hydrodynamics (SPH) models show that Lagrangian method is a very promising alternative approach to simulate wave breaking and overtopping due to its completely mesh-free technique.

In the present paper, SPHysics numerical model [3] is validated for wave propagation through an impermeable coastal structure for two different cases. In the first case, wave overtopping discharge over an impermeable sea wall defence structure is simulated. Numerical results of SPHysics model are compared with results from a SPH model [4], from Eulerian numerical models [5] and with experimental data obtained by Saville (from Shao et al. [4]). Validation of the numerical model for this very complex phenomenon is performed studying the influence of various parameters, such as the initial density of particles, the viscosity model (artificial [6] and SPS [7]) and the ε_{XSPH} parameter of the XSPH variant of Monaghan [8] that allows correcting the velocity of a particle. In the second case, SPHysics model is applied for modelling wave propagation with breaking and overtopping of an impermeable sea wall coastal defence structure, a common structure employed at the Portuguese coast. Numerical results of free-surface deformation at several positions along the flume and overtopping discharge are compared with experimental data from model scale tests obtained at the National Civil Engineering Laboratory (LNEC) in the framework of the Composite Modelling of the Interactions between Beaches and Structures (CoMIBBs) project – HYDRALAB III European project [9].

Improvements On SPH Neighbor List

J.M. Domínguez
Grupo de Física de la Atmosfera y del Océano
Universidad de Vigo
Orense, Spain
jmdalonso@gmail.com

A.J.C. Crespo
Grupo de Física de la Atmosfera y del Océano
Universidad de Vigo
Orense, Spain
alexbexe@uvigo.es

M. Gómez-Gesteira
Grupo de Física de la Atmosfera y del Océano
Universidad de Vigo
Orense, Spain
mggesteira@uvigo.es

***Abstract*—Different methods to improve the efficiency of the neighbor list are presented here within the framework of SPHysics code. The efficiency of the methods is evaluated in terms of memory requirements and computational velocity using a dam break problem. The efficiency of the methods based on linked list has shown to be higher than observed in methods based on vectors or matrices.**

I. INTRODUCTION

SPH (Smoothed Particle Hydrodynamics), developed in 1977 for simulating astrophysical problems ([1], [2]), is the most popular mesh-free method. Conceptually, the method uses integral interpolation theory and transforms the partial differential equations into an integral form (the methodology is reviewed in [3], [4] and [5]. The method was first applied in fields such as impact penetration in solids ([6] and [7]) and two- phase flow ([8]). In the particular case of fluid dynamics, the technique, which was first applied to the study of free-surface flows [9], has experienced a remarkable improvement during the last decade ([10], [11], [12], [13], [14], [15], [16], [17], [18], [19], [20], [21], [22], [23], [24], [25], [26]).

SPHysics is a Smoothed Particle Hydrodynamics (SPH) ([3]) numerical model developed to study free-surface flows. It is the product of a collaborative effort amongst researchers at the Johns Hopkins University (US.), the University of Vigo (Spain) and the University of Manchester (UK). The first version of SPHysics was released on August 2007 and is available to free download and for public use at http://wiki.manchester.ac.uk/sphysics. SPHysics is used to simulate free-surface flows problems such as dam breaks, landslides, sloshing in tanks and wave impacts on structures. The model, written in FORTRAN90, has been put into modular form and a variety of features are available to choose different compiling options. One of the purposes of this code is encourage other researchers to try SPH more easily and to contribute to the SPHysics project. A complete description of the software is found in the SPHysics user's guide ([27]).

Conceptually, each particle is only influenced by the particles in its close neighbourhood. Actually, only particles at a distance shorter than *nh* (*n* depends on the kernel choice and *h* is the smoothing length) should be evaluated. Thus, the determination of which particles are inside the interaction range requires the computation of all pair-wise distances, a procedure whose computational time is very demanding in terms of computational time for large domains. Thus, the more efficient the neighbour search is, the more efficient the code will be. Different approaches have been developed to reduce the unnecessary computation of distances.

The aim of this paper is looking for implementations of the neighbor list improving SPH methods in terms of computational speed and memory storage.

II. NEIGHBOR LIST ON SPH

SPH is based on integral interpolants. The key idea is to consider that a function *A(r)* can be approximated be approximated by

$$A(\vec{r}) = \int A(\vec{r}')\,W(\vec{r}-\vec{r}',h)\,d\vec{r}' \qquad (1)$$

where *h* is the smoothing length and it controls the influence domain - (see Figure 1). This parameter controls the size of the area around a particle where contribution from the rest of the particles cannot be neglected. Typically, value of *h* must be higher than initial particle separation. This approximation, in discrete notation, leads to:

$$A(\vec{r}) = \sum_b m_b \frac{A_b}{\rho_b} W_{ab} \qquad (2)$$

Smoothed dissipative particle dynamics for the polymer in diluted solution

Litvinov S., Ellero M., Hu X. Y, Adams N. A.
Lehrstuhl für Aerodynamik,
Technische Universität München
85748 Garching, Germany

***Abstract*— The effects of hydrodynamic interaction (HI) on polymer dynamics are investigated by smoothed dissipative particle dynamics (SDPD), which is a new type of dissipative particle dynamics (DPD). The calculated static and dynamic properties agree well with previous studies by coarse-grained molecular dynamics (CMD), Brownian dynamics (BD), DPD and the predictions of the Zimm model, which implies that HI is fully developed under the conditions of current simulations. As the SDPD method is capable of simulating on the length and time scales of experiments, and includes the effects of HI and excluded volume in explicit and natural ways, more accurate predictions are expected for complex interaction evolving polymer dynamics.**

I. Introduction

The simulation of polymer dynamics in a solution requires an appropriate polymer interaction model and a suitable fluid model since the hydrodynamic interactions (HI) have strong effects [1]. Experiments have confirmed that for simple flows polymer dynamics in a dilute solution can be described by the Zimm model [2], where HI are modeled by the Oseen tensor and the solvent is constrained to behave like an incompressible Stokes flow. For more general flows, in particular wall bounded unsteady flows such simplified models are unsuitable. HI can be represented directly by molecular dynamics (MD). Such studies have been performed and provided valuable information about polymer dynamics [3]. An obvious disadvantage is the computational cost, and practically relevant length and time scales of experiments are out of reach. Coarse-grained molecular dynamics (CMD) [4] allows for some reduction of the computational effort, but the problem essentially remains. As a mesoscopic method, dissipative particle dynamics (DPD) allows for simulations on much larger length and time scales [5]. SDPD is a thermodynamically consistent version of smoothed particle hydrodynamics (SPH) [6] discretizing the Navier-Stokes equations on a set of Lagrangian particles and introducing thermal fluctuations according to the fluctuation-dissipation theorem [7]–[9] for representing mesoscale effects. In this note we propose a new model based on smoothed dissipative particle dynamics (SDPD) for the numerical simulation of polymers in dilute solution, which is suitable for complex flows. The method inherits the favorable properties of SPH for complex flows and the efficient representation of mesoscopic effects of SDPD. Immersed polymers are taken into account by a straight-forward modification of the SDPD-particle interactions of such particles which contain parts of the polymer. The method is validated by comparison with theoretical results for generic cases.

The main objective of the model validation is to show the effect of the hydrodynamic interactions on the properties of the polymer. We used two theoretical models as references: Rouse model [10] — chain represented as a string of beads connected by spring, hydrodynamic interactions is ignored and mentioned Zimm model involving hydrodynamic interactions. The main dynamics quantities which can be validated in simulation are diffusion coefficient (D) and a chin relaxation time (τ). The Rouse model predicts $D \propto N^{-1}$, $\tau \propto NR_g^2$, according to Zimm model $D \propto R_g^{-1}$, $\tau \propto R_g^3$ (where N is the numbers of beads, and R_g is the gyration radius of the chain). Some more fine validation tests can be based on analysis of the Rouse modes of the polymer.

The behaviour of static properties of the polymer are not effected by the presents of hydrodynamic interactions but allows us to make conclusions about topological properties of the simulating chain.

II. Smoothed Dissipative Particle Dynamics

Smoothed Dissipative particle Dynamics is a mesoscopic particle method able to introduce thermal fluctuations as in DPD and solve correctly the Navier-Stokes equations as in SPH. Indeed, SDPD is 'exactly' a generalization of SPH for mesoscopic flow problems, i.e. with the thermodynacally consistent inclusion of thermal fluctuations. Additionally, the thermal fluctuations in SDPD posses a consistent thermodynamic scaling with the particle volumes, recovering SPH in the limit of large physical systems.

A. Macroscopic solvent modeling: hydrodynamics

The starting point are the isothermal Navier-Stokes equations written in a Lagrangian form

$$\frac{d\rho}{dt} = -\rho\nabla\cdot\mathbf{v} \quad (1)$$

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

where ρ and $\mathbf{v}$ are the mass density and the flow velocity respectively. It has been shown that a second-order discretization

Remarks on FSI simulations using SPH

Libor Lobovský
Department of Mechanics
Faculty of Applied Sciences, University of West Bohemia
Univerzitní 22, 30614 Plzeň, Czech Republic
lobo@kme.zcu.cz

Paul H.L. Groenenboom
ESI Group Netherlands
Rotterdamseweg 183 C, 2629 HD Delft, Netherlands
pgr@esi-group.com

***Abstract*— Within this study attention is paid to numerical simulation of fluid-structure interaction (FSI) by means of the smoothed particle hydrodynamics (SPH) method. A series of FSI simulations has been carried out using two independent solvers, the pure SPH code and the coupled SPH and finite element (FE) solver. Within both codes, the FSI is realised through a penalty contact force algorithm (which is relatively simple, fast and flexible). The results of the pure SPH and FE-SPH simulations have been confronted with each other and with experimental and numerical data published in literature. Even though a satisfactory agreement between results of the performed simulations and the published data can be achieved, a significant inconsistency between SPH and FE-SPH results is observed when the same model is applied. It is shown that the inconsistency between results of the pure SPH code and of the FE-SPH simulation is not caused by the SPH model for the fluid or by the penalty contact force algorithm, but by a difference in the elastic response of the FE elastic material and of the SPH elastic solid, which is related to the SPH solid model representation and its additional stabilising terms.**

I. INTRODUCTION

Since its introduction in late 1970s, [9] and [17], the Smoothed Particle Hydrodynamics (SPH) has become a popular tool for numerical simulations of various fluid flow studies, in particular those involving free surfaces or interfaces, e.g. [5], [20]. The meshless Lagrangian nature of the SPH description makes the method capable of handling the large deformations that occur for fluid flow while tracing the particles without the numerical dispersion that may occur due to transporting the interface geometry across cells such as for mesh-based approaches. In case an interface between a fluid and an elastic structure is being modelled, an application of the meshless method for both the fluid and the structure would be convenient especially for cases when large displacements and large deformations at the interface may occur.

Several studies on SPH application within the structural dynamics were published in the past, e.g. [11], [16], many of them related to high velocity impacts. When the SPH method is applied to low speed elastodynamic problems, the results might get corrupted by the tensile instability of the SPH model [25]. This is associated with numerical clustering of particles which may result in an unphysical behaviour of the modelled structure and consequently also in its artificial fracture. Thus several approaches treating the SPH tensile instability were proposed in the literature, e.g. [7], [10].

Besides several publications on numerical modelling of fluid to structure or structure to structure interaction by coupling the SPH method to mesh-based methods, e.g. [6], [26], few studies related to computational analysis of these problems purely by the SPH approach were performed, e.g. [3], [4] or [23]. In a publication by Antoci et al., [1], experimental results as well as SPH analysis of an FSI problem of a fluid escaping from a reservoir through an elastic gate were presented. Antoci et al. proposed a complex FSI algorithm utilising an approximate SPH evaluation of a surface integral which requires an explicit identification of the interface between the fluid and the structure. That might be challenging when a complex distortion or fracture of a structure surface is involved. The simulation of Antoci et al. has been repeated in [19], where a simplified FSI algorithm without a need for interface topology identification was applied and similar results were obtained.

In the following, results of the pure SPH simulations using the simplified FSI algorithm are confronted with coupled finite element (FE) and SPH simulations for the Antoci problem. An inconsistency between the pure SPH and the coupled FE-SPH solution is identified and discussed.

II. NUMERICAL METHODS EMPLOYED

The simulations of the FSI problem are carried out using two independent software codes: LOBO-SPH, a special SPH solver developed at the University of West Bohemia, and PAM-CRASH, the commercial finite element code with SPH solver.

The first solver, LOBO-SPH, is a pure SPH solver capable of simulating the fluid dynamics as well as the elastodynamics by means of standard SPH method. The fluid model utilises the standard SPH model for inviscid flow of a slightly compressible fluid, [20], with Murnaghan equation of state. The viscosity of the fluid is modelled using the approximation of Morris et al. [22]. The isotropic elastic material is modelled following the SPH approach of [16] while employing the artificial stress term by [10]. The stability of the SPH code is enforced by an application of additional numerical stabilising terms; artificial viscosity [21] and correction of particle motion [20]. Several types of boundary conditions are implemented. In the following, the rigid boundary conditions are represented by two sets of boundary particles while an additional corrective term may be applied. Details on LOBO-SPH implementation can be found in [19].

Graphics Processing Unit Accelerated Calculations of Free Surface Flows using Smoothed Particle Hydrodynamics

Christopher McCabe, Derek M. Causon and Clive G. Mingham
Centre for Mathematical Modeling and Flow Analysis
Manchester Metropolitan University
Manchester, UK
c.mccabe@mmu.ac.uk

***Abstract*—The Graphics Processing Unit offers the potential to significantly accelerate calculations for computational fluid dynamics due to its ability to rapidly process Single Instruction Multiple Data algorithms that are very common in CFD. One such algorithm is Smoothed Particle Hydrodynamics. Previous attempts at implementing SPH on GPUs have generally been more concerned with using the video output for interactive graphics which has given significant accelerations when compared to implementations on a CPU. The Nvidia Tesla C870 does not have video output but instead creates and manages many threads, and particles can easily map onto the threads. The C870 offers the potential to rapidly simulate a problem with SPH without interactive graphics and the results viewed post-simulation, but very little work of this kind has been done. This paper compares the performance of a number of implementations of the same SPH algorithm executed on a Nvidia Tesla C870 and a vector supercomputer for a 2D dam break problem in order to compare the performance of the two distinct computer architectures and to also compare the performance of different implementations of the same SPH algorithm on a C870.**

I. Introduction

The Graphics Processing Unit (GPU) has come to the interest of the scientific computing community due to its speed, size, power consumption and its ability to process SIMD algorithms. Meshless methods such as Smoothed Particle Hydrodynamics (SPH) are very suitable for this kind of parallel processing in which particles easily map to threads in a program. A manufacturer of graphics cards is Nvidia who have been producing graphics cards with video output, such as those used for VDU display, for years. Nvidia have recently begun producing multiprocessors based on graphics processing card architecture but without the video output. These multiprocessors have a unique architecture and thus must be programmed with a unique language which Nvidia have developed and called CUDA. CUDA and the Nvidia Tesla system architecture are covered in a later section of this paper.

There is very little, if any, literature on performance comparisons of different SPH algorithms on different computer architectures. Most work on SPH on GPUs appears to be concerned with interactive algorithms for gaming etc., which may well be due to the fact that the processors used so far have been actual graphics processors with video output, all involving a certain degree of rendering which makes comparison of the execution times of the underlying SPH algorithms potentially invalid if not impossible.

Harada et al [1] proposed an algorithm employing a variation of SPH and rendering which used the video functionality of the Nvidia GeForce 8800GTX processor to store the physical properties of the particles, and compared the execution times of the same algorithm executed on a Core 2 X6800 2.93GHz CPU, and quote a maximum speed up of 28.5x for a 3D dam break problem consisting of 262144 particles. Although the GPU used, the GeForce 8800GTX, is CUDA-enabled it is not clear if CUDA was used in that work. The authors of that work stress that one of the advantages of their algorithm is that all computation is done on the GPU. This is the case in the GPU-based implementations proposed in this work. Amada et al [2] also implemented SPH on a GPU, but the particle interactions were searched for on the CPU and thus did not exploit the power of the GPU. The GPU used in that work was a Nvidia GeForce FX 5950, which is not CUDA-enabled. Strzodka et al [8] look at implementations of both grid and particle based methods on GPUs but this was done before the Nvidia Tesla C870 had been manufactured. Kolb et al [12] proposed an implementation of SPH on a Nvidia 6800 GT GPU, which is not CUDA enabled, with all computation performed on the GPU and used it for interactive graphics. More recently Mark Harris [3] presented on Nvidia's PhysX gaming technology which implements a SPH algorithm in CUDA. The SPH algorithm used was based on that proposed by Müller [4]. An interesting application of a particle method, though not SPH, was made by Cuntz et al [13] in interactive simulation of the climate. This work was done on a Nvidia GeForce 5900, which is not CUDA-enabled, using the texture function of the GPU.

The purpose of this paper is to compare the execution of the same SPH algorithm, without any rendering for interactive graphics, on two different computer architectures to assess their performance. The authors are more concerned with the potential of SPH as an algorithm for scientific investigations and the potential of GPUs in accelerating such research. Visualization of the simulations was possible post-simulation via

Two way Boussinesq-SPH hybrid model to study free surface waves

M. Narayanaswamy
Department of Civil Engineering,
Johns Hopkins University
Baltimore, MD
muthu@jhu.edu

A.J.C. Crespo
Grupo de Física de la Atmosfera y del Océano
Universidad de Vigo
Orense, Spain
alexbexe@uvigo.es

M. Gómez-Gesteira
Grupo de Física de la Atmosfera y del Océano
Universidad de Vigo
Orense, Spain
mggesteira@uvigo.es

R.A. Dalrymple
Department of Civil Engineering,
Johns Hopkins University
Baltimore, MD
rad@jhu.edu

***Abstract*—A hybrid model (SPHunwave) is developed combining the main advantages of a Boussinesq model (FUNWAVE) and a SPH model (SPHysics). The details of the coupling procedure along with preliminary validation tests are presented.**

I. INTRODUCTION

The study of wave propagation constitutes an arduous task due to the multiple spatial and time scales involved. Thus, different models have been used to study wave propagation from the open ocean to the coast (see [1] for an overview of the different models used to describe wave propagation. In general, each model has several advantages and limitations and it is especially well suited for a certain area and conditions.

During the last few years, research has been focused on coupling models based on different numerical, mathematical and, even, conceptual approaches to combine the main advantages of the individual models in a single model [2,3,4]. This should result not only in more accurate and efficient models but also in an increased regime of validity.

Depth averaged models such as those based on Boussinesq equations have been widely used to describe wave propagation from the offshore to the nearshore region [5,6,7]. These models are attractive due to their good balance between computational efficiency and physical accuracy, which permits their use over large domains. However, these models do not provide accurate information about flow details when the wave arrives at the coast and highly nonlinear processes become dominant.

SPH (Smoothed Particle Hydrodynamics), developed in 1977 for simulating astrophysical problems [8,9], is one of the most popular mesh-free methods(see the bases of the methodology are described in [10,11,12]). In the case of fluid dynamics, the technique was first applied to the study of free-surface flows [13] and has been widely used during the last decade [14,15,16,17,18,19,20,21,22,23,24,25,26,27,28,29,30]. In particular, SPHysics is a SPH numerical model developed to study free-surface flows. It is the product of a collaborative effort amongst researchers at the Johns Hopkins University (US), the University of Vigo (Spain) and the University of Manchester (UK). The first version of SPHysics was released on August 2007 and is available to free download and for public use at http://wiki.manchester.ac.uk/sphysics. SPHysics is used to simulate free-surface flows problems such as dam breaks, landslides, sloshing in tanks and wave impacts on structures. The model, written in FORTRAN90, has been put into modular form and a variety of features are available to choose different compiling options. One of the purposes of this code is encourage other researchers to try SPH more easily and to contribute to the SPHysics project. A complete description of the software is found in the SPHysics user's guide [31]. Although the method allows a fine description of the flow in the shore areas, its main drawback is its high computational cost, in such a way that the model cannot be efficiently applied over large domains.

The aim of this paper is the development of a hybrid model that that combines the advantages of two existing wave propagation models. This hybrid model (SPHunwave) couples a finite difference Boussinesq model (FUNWAVE) to SPHysics. The main features of both models can be seen in their respective *user's guides* [31,32]. The methodology of coupling the two models is presented here and the simulation of a solitary wave in a tank with constant depth is discussed

Massively-Parallel SPH-Simulations of Viscous Flows

Christian Ulrich, Thomas Rung
Institute of Fluid Dynamics and Ship Theory (M-8)
Hamburg University of Technology (TUHH)
Hamburg, Germany
christian.ulrich@tuhh.de, thomas.rung@tuhh.de

***Abstract*—The paper reports on the predictive-prospects of SPH for massively-parallel computations of hydrodynamic engineering flows. Accordingly, a cosmological N-body/SPH simulation code GADGET-2, based on a parallel TreeSPH code [8] has been modified to compute viscous engineering flows. The present SPH-implementation is based on weakly compressible viscous flows using Tait's pressure-density relation. Time integration follows from a second-order accurate leap-frog scheme, where adaptive time-steps may also be employed. The employed domain decomposition is based on a space-filling curve. An oct-tree concept is utilized for the nearest-neighbour search. The parallel efficiency of the modified code has been scrutinized on both PC-Clusters and massively-parallel HPC systems. Computed cases for up to 40 Mio particles have been analyzed. Validation examples included refer to simple shear-driven axissymmetric Couette-Flows and an SPH-model for sloshing in an anti-roll tank. Results indicate a fair parallel performance up to 256 processors on a SGI-Altix XE 250 machine featuring 2 quadcore sockets each for Intel Xeon Harpertown processors running at 3.0 GHz.**

I. Introduction

The industrial application of SPH-codes is traditionally limited by the maximum number of praticles which can be employed if the simulation is supposed to be completed within a reasonable time-to-solution. Reliable fluids-engineering simulations often rely on fairly accurate discretisations, leading to large numbers of particles. The discretisation effort increases with the complexity of the computed geometry and the involved physics (e.g. multiple phases or continua), as well as the need to accurately resolve local details. The only conceivable options to increase the number of particles are the efficient use of commodity-hardware "Beowulf" clusters in conjunction with high-speed interconnects or massively parallel processing (MPP) supercomputers.

The paper is concerned with a data-parallel SPH-procedure which is designed to simulate weakly-compressible, viscous engineering flows following Monaghan's SPH-formalism [6].

The employed SPH-code is an immediate modification of the cosmological simulation code GADGET-2, based on the work of Springel [8]. The original procedure is devoted to hydrodynamical cosmological simulations. It follows the evolution of a self-gravitating collisionless N-body system, and allows gas dynamics to be optionally included. GADGET-2 is a data-parallel, single-programm/multiple-data (SPMD) procedure. Inter-processor communication is realized using the standard Message Passing Interface (MPI) library. Unlike earlier versions of GADGET and various other particle-simulation approaches (e.g. [3]), GADGET-2 employs an efficient domain-decomposition technique. The strategy is based on the position-based mapping of particles into a space-filling curve (SFC), as proposed by Warren and Salmon [10].

Although being an ideal starting point, GADGET-2 is not directly applicable to wall-bounded fluids-engineering simulations of viscous incompressible fluids. Therefore, we adapt Springel's parallelisation strategy to a different simulation area using modified governing equations and their respective SPH representation. We refer to the modified code as "GADGET-$\mathrm{H}_2\mathrm{O}$".

The paper is structured as follows: In section II, the employed governing equations and their respective finite approximations are described. Section III provides an overview over the parallelisation strategy. Validation cases are presented in section IV. Section V depicts a performance analysis for simulations with up to $40 \cdot 10^6$ particles. Final conclusions are summarised in section VI.

II. SPH-Formalism

The following section outlines the governing equations and their respective SPH-based approximations. Vectors and tensors are defined by reference to cartesian coordinates. The notation uses latin subscripts to identify particle locations and greek superscripts to mark cartesian tensor coordinates. The latin subscript i denotes to the focal particle whereas the subscript j refers to its neighbours. Mind, that Einstein's summation is employed over repeated Greek superscripts.

A. Conservation of Mass

The transient evolution of the density for a particle i follows from the continuity equation

$$\frac{D\rho_i}{Dt} = \sum_{j=1}^{N} \left[m_j (v_i^\beta - v_j^\beta) \right] \frac{\partial W_{ij}}{\partial x_i^\beta} \,, \tag{1}$$

where m denotes the particle mass and ρ marks the particle density. The velocity and the position of a particle refer to the vectors v^β and x^β, respectively. The kernel function around

Influence of the compressibility in Fluid-Structure interaction Using Weakly Compressible SPH

G. Viccione, V. Bovolin
Dept. of Civil Engineering
University of Salerno
Via Ponte Don Melillo, 84084, Fisciano, Italy
gviccion@unisa.it, v.bovolin@unisa.it

E. Pugliese Carratelli
C.U.G.Ri. – University Centre for
Research on Major Hazards
P.zza V. Emanuele, 84080, Penta di Fisciano, Italy
epc@unisa.it

***Abstract*— For computational reasons conventional SPH method often rely on a weakly compressible fluid model, based on the assumption that below a certain Mach number the solution will not depend upon the compressibility. Doubts have been risen on the acceptability of such a procedure for highly non steady flows, such as the impact on a rigid wall. A number of Authors have therefore been experimenting with Incompressible SPH.**

In this paper, the authors report on work carried out to enquire into the effects of compressibility on pressure distributions in a particular non-stationary example. The impact of a current generated by a 3D dam break over a rectangular channel is investigated. The Authors have then tested an Incompressible SPH method for the same problem, in order to compare the quality and the efficiency of the two approaches.

I. INTRODUCTION

Rapidly-varied free surface flows feature large deformations that can be easily handled by the Smoothed Particle Hydrodynamics (SPH) technique [1], [2]. Many computations, being carried out by different scientists, proved its efficiency into reproducing accurate kinematic results. Nevertheless, the original method, based on the fluid being treated as weakly compressible (WCSPH), does not generally yield acceptable pressure fields due to the noise produced by the propagation of elastic waves which decrease and eventually vanish as the Mach number decreases. Unfortunately, the lower the Mach number, the smaller the timestep needed to satisfy the Courant condition will be. Refs. [3], [4] have shown that reproducing an impact, respectively on a square cylinder and on a solid wall, produces unreliable pressure distributions. A number of implementations have then been developed around the primitive WCSPH, in order to smooth out these spurious oscillations. In Ref. [5], for instance, the Authors put a conservative diffusion term in the continuity equation, which preserves mass conservation, together with an artificial viscosity term into the momentum equation making stable the resulting integrator scheme. Alternatively, dynamic loads may be accurately reproduced treating the fluid as strictly incompressible, making use of the Incompressible Smoothed Particle Hydrodynamics (ISPH) method [6], [7].

Main differences among these approaches are shown in Table 1.

TABLE I. DIFFERENCES AMONG THE WCSPH AND ISPH TECHNIQUES

Feature	Technique	
	WCSPH	***ISPH***
Integration in time	Explicit[a]	Implicit
Pressure evaluation	Straight with a stiff equation of State	By solving the Poisson equation in matrix form
Courant condition	Sound speed or Mach	Max velocity of particles
Density	Variable, within 2%	Constant

In the following, a 3D impact on a solid vertical wall of a Newtonian fluid flow originated from a dam failure is simulated with both WCSPH and ISPH techniques. In the first case, the influence of compressibility on the pressure is investigated by varying the Mach number in the range [0.01 – 0.1]; in the latter, pressure distribution is given for the same water stage behind the solid.

II. SPH MODELS

The Navier – Stokes equations,

$$-\frac{1}{\rho}\frac{D\rho}{Dt} = \nabla \cdot \underline{v} \qquad (1.a)$$

$$\rho\frac{D\underline{v}}{Dt} = -\nabla p + \mu\nabla^2\underline{v} + \underline{f} \qquad (1.b)$$

governing fluid motion, are in the following rewritten in the discretized form considering the fluid as slightly compressible or strictly incompressible.

A. The modified WCSPH technique

The original WCSPH technique has been modified according to [5].