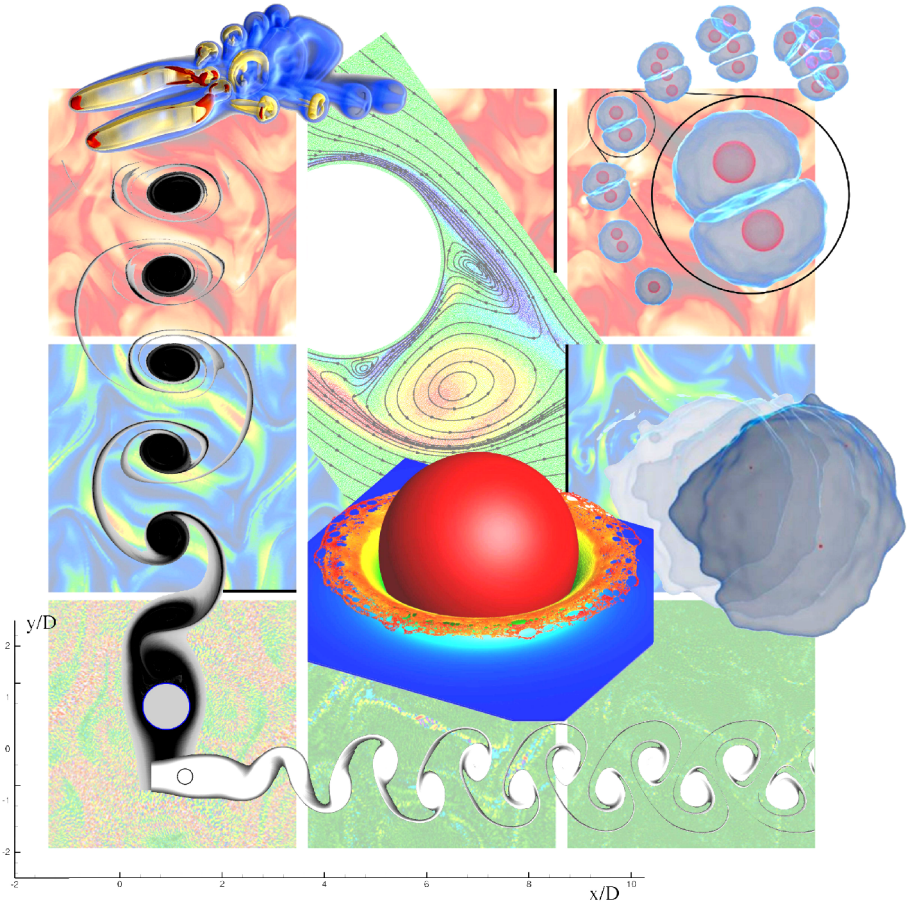
7th spheric 2012

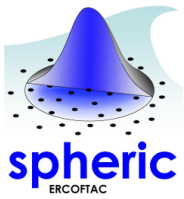
http: //spheric7.monash.edu/

Prato, Italy

29 - 31 May 2012



Foreword to the 7th International SPHERIC Workshop Proceedings



Monash University (Australia), INSEAN (Italy), and the University of Pavia (Italy) are pleased to organise the 7th SPHERIC workshop, and to welcome you to the ancient but stimulating city of Prato.

The SPHERIC workshop is the main annual event on smoothed particle hydrodynamics (SPH). The governing organisation is SPHERIC (the SPH European Research Interest Community) which is part of ERCOFTAC. The vitality SPHERIC is due to the interdisciplinary nature of the subject where engineers, physicists and mathematicians can work together.

The SPHERIC community plays an important role in the advancement and dissemination of SPH methods and ideas. The method has matured over the last decade and includes an extraordinary range of applications that will be evident by a quick scan of the table of contents of this document.

The depth of interest in SPH can be measured by the fact that nearly 80 abstracts were submitted and a final set of 60 papers accepted for presentation.

This format of the workshop in 2012 involves some changes to previous SPHERIC workshops. We have introduced discussion periods and made the materials we use more environmentally friendly. We hope you find that these changes make the workshops even better. Do let us know what you think.

A workshop such as this requires considerable support. In particular we would like to thank Monash University, ERCOFTAC, and HydrOcean for their contribution.

Joe Monaghan and Jules Kajtar
On behalf of 7th SPHERIC local organising committee.

Contents

Day 1: Tuesday 29 May 2012

Session 1: Multi-Fluids

SPH modelling of two-phase bubbly flows
E. Torti, S. Sibilla 1

SPH multiphase simulation of bubbly flows
N. Grenier, M. Kerhuel, D. Le Touzé, A. Colagrossi, G. Colicchio, M. Antuono, D. Zuzio . . 7

Surface tension and wetting phenomena with SPH
T. Breinlinger, A. Hashibon, T. Kraft 14

Contact line hydrodynamics with SPH
S. Adami, X.Y. Hu, N.A. Adams 18

Session 2: Geotechnical Applications

SPH non-Newtonian model for ice sheet and ice shelf dynamics
A. M. Tartakovsky, W. Pan, J. J. Monaghan 23

Simulation of film and droplet flow on wide aperture fractures using smoothed particle hydrodynamics
J. Kordilla, A. Tartakovsky, T. Geyer 31

Application of SPH to erosion and excavation problems on the examples of jet grouting and offshore engineering
B. Stefanova, J. Bubel, J. Grabe 38

A simple SPH model of water-soil interaction in porous
C. Ulrich, T. Rung 43

Session 3: Solids and Fracture Mechanics

SPH simulation of granular material collapses
E. Paris, L. Minatti 51

Towards simulations of abrasive flow machining
C. Nutto, C. Bierwisch, H. Lagger, M. Moseler 59

A modified Godunov SPH method for materials with strength
A. Connolly, L. Iannucci 65

Dynamic refinement for SPH simulations of post-failure flow of non-cohesive soil
Y. R. López, D. Roose, C. R. Morfa 71

Session 4: Validation

Flow prediction of reactive rotational molding using smoothed particle hydrodynamics method
S. Riviere, S. Farzaneh, A. Tcharkhtchi, S. Khelladi, F. Bakir 78

SPH simulations and experiments of sloshing in an egg-shaped shell
J. Grant, M. Prakash, S. E. Semercigil, O. F. Turan 84

On the use of a time-dependent driving force in SPH simulations
Sh. Khorasanizade, J. F. Pinto, J. M. M. Sousa 92

Incompressible smoothed particle hydrodynamics: proposition and validation of a fully-explicit algorithm
D. A. Barcarolo, D. Le Touzé, F. de Vuyst 99

Modeling of gravity wave viscous attenuation
A. Colagrossi, A. Souto-Iglesias, M. Antuono 107

Session 5: Astrophysical Applications

Hyperbolic divergence cleaning for SPH
T. S. Tricco, D. J. Price 115

Modelling magnetic fields and turbulence with SPH
D. J. Price 123

An algorithm for dusty gas with SPH
G. Laibe, D. J. Price, B. A. Ayliffe 131

Day 2: Wednesday 30 May 2012

Session 6: Boundary Conditions and Validation

2D and 3D sloshing simulation by SPH
M. Leonardi, S. Manenti, S. Sibilla 137
SPH modeling of non-rectangular channel flows with open boundaries
K.-H. Chang, T.-J. Chang 144
Study of differential operators in the context of the semi-analytical wall boundary conditions
A. Mayrhofer, B. D. Rogers, D. Violeau, M. Ferrand 149
Apply C1 consistency to SPH with free surface
H. Xu, M. H. Dao, E. S. Chan, P. Tkalich 157
SPH modelling of viscous flows around cylinders from Re=10 to Re=1000
S. Marrone, M. Antuono, A. Colagrossi, G. Colicchio, G. Graziani 163

Session 7: Turbulence Modelling

Turbulent coherent structures under breaking water waves
R. Jalali-Farahani, R. A. Dalrymple, A. Hérault, G. Bilotta 171
A SPH model for incompressible turbulence
X. Y. Hu, N. A. Adams 179
SPH simulations of 2D turbulence driven by stirrers
A. Valizadeh, J. J. Monaghan 187
Advective and diffusive turbulent mixing
J. J. Monaghan, J. B. Kajtar 195

Session 8: Alternative Formulations

On the use of numerical diffusive terms in weakly-compressible SPH schemes
M. Antuono, A. Colagrossi, S. Marrone 200
SPH-ALE for simulations of rotor-stator interactions
M. Neuhauser, J.-C. Marongiu, F. Leboeuf, M. Rentschler, E. Parkinson 208
FPM simulations of a 3D impinging jet on a flat plate comparison with CFD and experimental results
C. Vessaz, E. Jahanbakhsh, F. Avellan 214
Remeshed Particles: a robust and efficient method for multiphysics simulations
W. M. van Rees, P. Koumoutsakos 221

Session 9: Maritime Applications

Use of SPHERA code to investigate local scouring effects induced by fluvial structures downstream a barrage
G. Agate, R. Guandalini, S. Manenti, S. Sibilla, M. Gallati 229
SPH modelling of propeller induced harbour-bed erosion by a container vessel
C. Ulrich, T. Rung 236
SPH simulations of bow waves dynamics
B. Bouscasse, S. Marrone, A. Colagrossi, R. Broglia 244
Using SPHysics to simulate a Wigley hull in head waves
M. Pearce, G. Thomas, D. Hudson 252

Session 10: High Performance Computing

Efficient parallelisation of 3D SPH scheme
D. Guibert, M. De Leffe, G. Oger, J.-G. Piccinali 259
New OpenMP-MPI-CUDA implementation for parallel SPH simulations on heterogeneous CPU-GPU clusters
J. M. Domínguez, A. J. C. Crespo, M. Gomez-Gesteira, D. Valdez-Balderas, B. D. Rogers . 266
A journey from single-GPU to optimized multi-GPU SPH with CUDA
E. Rustico, A. Hérault, G. Bilotta, C. del Negro, G Gallo, R. A. Dalrymple 274
Parallelisation of a finite volume particle method code
M. Basa, L. Lobovský, N. J. Quinlan 282

Day 3: Thursday 31 May 2012

Session 11: Boundary Conditions and Validation

Use of complex inlet boundary conditions for accelerated studies of green water events
C. Pâkozdi, C. T. Stansberg, SINTEF SCORE-team 287
Absorbing inlet/outlet boundary conditions for 2D SPH turbulent free-surface flows
O. Mahmood, D. Violeau, C. Kassiotis, B. D. Rogers, M. Ferrand 296
On the boundary condition enforcement in SPH methods
L. M. González, J. L. Cercós, F. Maciá 303
A diffusion based shifting algorithm for incompressible smoothed particle hydrodynamics: Validation with cases involving slamming bodies and cylinder exit
A. Skillen, S. Lind, B. D. Rogers, P. K. Stansby 311

Session 12: Multi-Fluids

SPH multi-fluid model with interface stabilization based on a quasi-buoyancy correction
A. C. H. Kruisbrink, F. R. Pearce, T. Yue, K. A. Cliffe, H. P. Morvan 317
A multiphase incompressible-compressible smoothed particle hydrodynamics method
S. J. Lind, P. K. Stansby, B. D. Rogers 324
SPH for two-phase fluid flow including cavitation
P. Groenenboom 333
A consistent particle method for simulation of multiphase flows with high density ratios
A. Khayyer, H. Gotoh 340

Session 13: Alternative Formulations

Development of SPH variable resolution using dynamic particle coalescing and splitting
R. Vacondio, B. D. Rogers, P. K. Stansby, P. Mignosa, J. Feldman 347
Development of the finite volume particle method for internal flow with rigid body dynamics
N. J. Quinlan, L. Lobovský, M. Basa, R. M. Nestor 355
Third-generation RSPH in 3D
S. Børve 361
Development and validation of a SPH model using discrete surface elements at boundaries
A. Amicarelli, G. Agate, R. Guandalini 369
An improved consistent 3D particle method for enhanced wave impact calculations
H. Gotoh, A. Khayyer 375

Session 14: Hydraulic Applications

Experimental and numerical modeling of the impulsive dynamics of an underwater non-cohesive sediment deposit subjected to a gaseous jet
S. Manenti, S. Sibilla, M. Gallati, G. Agate, R. Guandalini 381
3-D coastal inundation simulation using a shallow-water solver
J. Zhao, D. Le Touzé, L. Gentaz, P. Ferrant 387
Improved accuracy in modelling armoured breakwaters with SPH
C. Altomare, X. F. Gironella, A. J. C. Crespo, J. M. Domínguez, B. D. Rogers 395
Simulation of dam-break flow in channel expansion with coupled 2-D/3-D SPH model
E. Džebo, D. Žagar, M. Četina, G. Petkovšek 403
SPH benchmarking: a comparison of SPH variants on selected test cases within the NextMuSE initiative
D. Le Touzé, D. A. Barcarolo, M. Kerhuel, F. Leboeuf, J. Caro, F. Le, N. Quinlan, L. Lobovsky, M. Basa, A. Colagrossi, S. Marrone, J.-C. Marongiu, M. De Leffe 409

Author Index 417

SPH modelling of two-phase bubbly flows

Emanuela Torti, Stefano Sibilla
Dipartimento di Ingegneria Civile e Architettura
Università di Pavia
Via Ferrata, 1 – 27100 Pavia, Italy
stefano.sibilla@unipv.it

***Abstract*— The convection-diffusion of gas bubbles in a liquid is modeled by a fully coupled Lagrangian approach in which the liquid flow is simulated by a traditional SPH scheme while the trajectories of the particles in the bubbly phase are directly tracked in time. Two families of particles (SPH liquid particles and gas bubbles) are defined. The momentum exchange between the bubbles and the surrounding fluid is taken into account by a suitable kernel approximation centered either in a bubble or in a SPH particle. Therefore, the motion of a liquid particle depends also on the reaction forces (such as drag and lift) exerted on it by the gas bubbles included in its domain of influence. Preliminary tests (which do not account for bubble-bubble interactions) have been performed by simulating an horizontal gas-liquid jet deflected by the interaction with the bubbly phase and the bubble-driven flow in a column. The results confirm the validity of the implemented approach for the simulation of the coupled liquid-gas system.**

I. INTRODUTION

The convection-diffusion of gas bubbles in a liquid flow impacts on many engineering branches, ranging from the traditional hydraulic applications (e.g. aerated hydraulic jumps, cavitation bubbles in pipes, etc…) to the most advanced chemical and environmental engineering systems, such as various types of reactor tanks or of wastewater biological treatment plants.

In the latter applications, usually, the concentration of certain species (e.g. oxygen) into the liquid phase is obtained by injecting a bubbly gas flow into the liquid (through jets or porous plates or other systems) in order to enhance the dissolution of the dispersed gas phase into the continuous liquid one. A proper evaluation of these mass transfer processes requires a detailed prediction of the motion of the gas bubbles and of their interaction with the surrounding liquid.

In recent years many efforts have been spent to try to improve the hydrodynamic modeling of these flows by adopting either Eulerian/Eulerian (E/E) or Eulerian/Lagrangian (E/L) approaches, in order to reproduce the complex dynamics of the interacting phases [1, 2].

The problems regarding bubbly flows are usually classified according to their complexity level. The simplest problems are the so-called "one-way coupling" problems in which the dispersed phase is simply convected by the continuous phase but has negligible effect on the liquid flow. In the "two-way coupling" problems, the bubbly flow interacts and modifies the liquid flow evolution. "Four-way coupling" problems consider also the interactions occurring within the dispersed phase, such as coalescence and rupture of bubbles.

Several numerical studies are available in literature. Studies on aeration columns involve E/E [3-5] and E/L approaches [6-8]. "One-way coupling" E/L analyses have been conducted also on bubbly jet systems used for oxygenation purposes [9].

Lagrangian/Lagrangian (L/L) approaches for the analysis of dispersed multiphase flows have been mostly applied in the case of solid dispersions in liquid flows. In particular, methods coupling a SPH representation of a continuous liquid or gas phase with a DEM description of a particulate have been used to simulate the sedimentation of solid particles [10, 11].

The present paper discusses a simplified L/L approach based on SPH for the representation of the continuous liquid phase, to simulate two-phase bubbly flows. The obtained results are compared with literature experiments which are relevant in the chemical and environmental engineering field, such as the motion of bubble columns originating from porous plates in aeration tanks [12] and bubbly jet flows [13].

II. NUMERICAL METHOD

The bubbly two-phase flow is here analyzed by a L/L approach which couples a SPH solution of the Navier-Stokes equations which govern the liquid flow with the direct solution of the Newton's law for the computation of the trajectories of a set of computational particles, each representing a certain number of gas bubbles. The interaction between the two phases is taken into account by a force term added to the momentum balance equation of the liquid phase and by a local equilibrium hypothesis when solving the bubble trajectories.

The approach is close, with some simplifications, to the SPH-DEM coupling approach adopted by [10, 11] to simulate solid suspensions in liquid flows.

The Navier-Stokes equations for the weakly compressible liquid phase can be written as.

$$\frac{D}{Dt}(\omega\rho) = -\omega\rho\vec{\nabla}\cdot\vec{v} \tag{1}$$

$$\frac{D\vec{v}}{Dt} = -\frac{\vec{\nabla}p}{\omega\rho} + \frac{1}{\omega\rho}\vec{\nabla}\cdot\tau + \vec{g} + \vec{f}_b \tag{2}$$

SPH multiphase simulation of bubbly flows

N. Grenier, M. Kerhuel, D. Le Touzé
LUNAM Université, École Centrale Nantes
LHEEA Lab. (UMR CNRS)
Nantes, France
nicolas.grenier@ec-nantes.fr

A. Colagrossi, G. Colicchio, M. Antuono
INSEAN, The Italian Ship Model Basin
Roma, Italy
CeSOS, Centre of Excellence for Ship
and Ocean Structures, NTNU
Trondheim, Norway
a.colagrossi@insean.it

D. Zuzio
ONERA
Toulouse, France
davide.zuzio@onera.fr

***Abstract*—The widely-used single-fluid SPH scheme allows for the successful modelling of various high dynamic flows although the role of additional non miscible phases cannot be neglected in many cases. These phases must actually be taken into consideration in order to obtain results more in line with the experiments for flows such as, sloshing, flooding of a confined volume, bubbly flows, or to capture a pressure peak smoothing by air entrapment in various situations.**

Two stable two-phase SPH models addressing the issue of accurately evaluating the quantities which are discontinuous across the interface were derived and validated in [1] and [?]. They are used here to study bubbly flows of increasing complexity. The viscosity and surface tension role are especially investigated. The study cases chosen are, a Rayleigh-Taylor instability, a single rising bubble reaching its terminal velocity and two merging bubbles. The results are compared to solutions given by mesh-based Level Set solvers.

I. Introduction

Multi-fluid flows of non miscible phases occur in numerous engineering fields and problems and present highly-nonlinear interfacial phenomena such as fragmentation and reconnection which are difficult to handle with traditional methods. The SPH method with its robust handling of interfaces is capable of maintaining sharp interfaces with virtually no numerical diffusion of one fluid into the other. Due to its Lagrangian character, interface creation and destruction happens naturally with no need to localize the interface.

II. Numerical Models

Near the interface, some physical properties such as pressure or normal velocity are continuous (in the absence of surface tension as far as pressure is concerned) while others such as density, tangential velocity for inviscid fluids, velocity divergence or pressure gradient are discontinuous. Yet the SPH interpolation is based on the evaluation of these variable on a radial kernel, which smoothes discontinuities. The challenge is thus to derive a discrete model which ensures the coexistence of both continuous and discontinuous fields.

In the two models described below, the classic modified Tait equation closes the system.

A. Multi-fluid ALE SPH formulation with Riemann solvers

The following ALE formulation is based on [2]:

$$\begin{cases} \left.\dfrac{d\overline{x_i}}{dt}\right|_{\overline{v_0}} = \overline{v_{0_i}} \\ \left.\dfrac{d\omega_i}{dt}\right|_{\overline{v_0}} = \sum_{j\in\Omega} \omega_i\omega_j \left(\overline{v_{0_j}} - \overline{v_{0_i}}\right)\cdot\overline{\nabla}W_{ij} \\ \left.\dfrac{d(\omega\overline{\Phi})_i}{dt}\right|_{\overline{v_0}} = -\sum_{j\in\Omega} \omega_i\omega_j \left(\overline{\overline{F_i}} + \overline{\overline{F_j}}\right)\overline{\nabla}W_{ij} + \omega_i\overline{S_i} \end{cases} \tag{1}$$

x_i, $\overline{v_{0_i}}$, ω_i, $\overline{\Phi}_i$, $\overline{\overline{F_i}}$ and $\overline{S_i}$ are, respectively, the position, arbitrary velocity, volume, conservative variable vector, flux and source term of particle i. W_{ij} represents the SPH interpolation kernel function.

Rather then using artificial viscosity to stabilize this scheme, it is possible to solve a mono-dimensional Riemann problem between each pair of particles. The multi-dimensional flux is obtained by spatially averaging these interactions. This procedure provides the minimum upwind to ensure numerical stability. Practically, the centred flux $\left(\overline{\overline{F_i}} + \overline{\overline{F_j}}\right)$ is replaced with the Riemann problem solution, taking into account the interface movement. To limit numerical dissipation, a MUSCL (Monotone Upstream-centred Scheme for Conservation Laws) procedure is also implemented.

This solution generates a flux of momentum between the particles but also a flux of mass. The latter must be blocked at the interface in order to avoid diffusion of one phase into the other, which is possible according to Leduc et al. [**?**] by carefully choosing the referential in which to solve the Riemann problem so as to cancel the mass flux.

Inside each phases, the Godunov solver (exact) is used; it is based on the resolution of a nonlinear equation which is approached with a Newton-Raphson iterative method, see [3]. At the interface, as in [**?**] we rather use an approximate Riemann solver where we impose consistently that the interface is a moving contact discontinuity. This kind of linearized solver is often referred to as acoustic solver.

B. Grenier et al. formulation

Grenier et al. [4] derived a multi-fluid formulation which is an improvement of the Hu and Adams one [5]. In partic-

Surface tension and wetting phenomena with SPH

Thomas Breinlinger, Adham Hashibon, Torsten Kraft
Fraunhofer Institute for Mechanics of Materials IWM
79108 Freiburg, Germany
thomas.breinlinger@iwm.fraunhofer.de

Abstract— **In numerous technological applications, surface tension effects are highly relevant. As the surface to volume fraction increases with smaller scales, the behavior of a fluid in contact with solid structures is primarily dominated by surface tension effects. In this study, we propose an efficient and reliable implementation in SPH, which prescribes the normal direction of the phase boundary at the triple line based on the desired equilibrium contact angle. The proposed algorithm is able to capture different wetting angles, pinning effects and wetting of structured surfaces.**

I. INTRODUCTION

Surface tension is very important for nano- and micro-scale free-surface and multiphase flows. Compared to macroscopic length scales, surface tension forces become dominant while the impact of inertial forces on the flow is strongly diminished. The wetting of water drops on the leaves of the Lotus plant is a popular example of a natural phenomenon caused by surface tension. Surface tension effects are also important in numerous industrial processes and technological applications, such as surface coating, inkjet printing, soldering and in microfluidic devices [1]. Due to this broad range of applications, the multitude of numerical simulation methods which have been developed to describe surface tension and flow is not surprising [2].

The inclusion of surface tension in SPH simulations has been addressed in several studies. The approaches used may be divided into two main classes. The first class uses pairwise interaction force-potentials to account for surface tension [6, 7]. These force-potentials are not readily available, and need to be fitted for each case in order to reproduce the desired contact angles and surface tension. Since in most practical applications, the surface tension and contact angles are given as macroscopic input parameters which vary often from case to case, this approach is not necessarily the most practical one.

The second classes of approaches are those based on the continuum surface force (CSF) model [8]. Here the pressure jump at the gas-liquid phase boundary, given by the Young-Laplace equation, is applied by introducing a volume force on a thin finite layer of matter on either side of the interface:

$$\mathbf{F}^{sv} = -\sigma\,\kappa\boldsymbol{n} \tag{1}$$

The magnitude of this force is determined by the surface tension coefficient σ and the curvature κ of the interface. Its orientation is given by the normal $\boldsymbol{n}$ of the interface. Morris [9] adapted this concept to an SPH scheme using a smoothed continuous color function to distinguish between the two phases. However, this resulted in difficulties calculating the normal and curvature of the interface, which were caused by non-full support of the kernel at the borders of the phase transition region [9]. Using a discontinuous color function the calculation of the curvature can be circumvented by introducing an interface stress tensor [10]. The remaining errors in the furthermost normal become irrelevant as the magnitude of the corresponding forces approaches zero. This method has been further developed using a density weighted summation gradient and a renormalized Laplacian of the color function, which allows for high ratios of density [11].

Besides the description of surface tension effects, modeling wetting phenomena using CSF requires the implementation of suitable boundary conditions, taking into account the interaction between fluids and the solid substrates near the triple line. The normal correction method [8] enforces the normal direction in the vicinity of the triple line to comply with the desired shape of the drop. This approach has been used within an asymmetric SPH implementation to simulate microchannel flow [12]. Hu and Adams [10] reproduced contact angles as derived from Young's equation by applying the tensor based scheme to three-phase systems. Alternatively, Liu and Liu [13] reproduced desired wetting angles by directly translating particles in the fluid-solid boundary region.

In this work we present an efficient and robust implementation of SPH that allows a reliable calculation of surface tension including the effects at the triple line based on the desired macroscopic contact angle. In particular, we introduce additional boundary conditions for the CSF implementation proposed by [11] to incorporate the effect of the interaction with solid walls. We apply our method to the simulation of equilibrium drop shapes. In [12] further extensive testing and validation will be presented.

II. GOVERNING EQUATIONS

This section describes the equations governing microscopic multiphase flow problems. The Navier-Stokes equations describe the motion of an incompressible fluid

$$\begin{aligned} \frac{\partial \rho}{\partial t} &= -\rho\boldsymbol{\nabla}\boldsymbol{v} \\ \frac{\partial \boldsymbol{v}}{\partial t} &= \frac{1}{\rho}\left(-\boldsymbol{\nabla}p + \eta\boldsymbol{\nabla}^2\boldsymbol{v} + \boldsymbol{F}^{(s)}\right) + \boldsymbol{g}\,, \end{aligned} \tag{2}$$

Contact line hydrodynamics with SPH

S. Adami, X.Y. Hu, N.A. Adams
Institute of Aerodynamics and Fluid Mechanics
Technische Universität München
Garching, Germany
stefan.adami@tum.de

***Abstract*—Surface tension effects can dominate multi-phase flows when the length scales of the problem are small. The resulting Capillary forces at a phase interface between two immiscible fluids are proportional to the local curvature of the flow and try to minimize the interfacial area. A more complex situation occurs when three phases are in contact or when two phases are in contact with a wall. The simulation of the contact line at a wall is still a challenging task since the motion of the contact line is contradictory to the no-slip assumption at walls.**

In this work we present a multi-phase SPH method considering surface tension effects that is capable of simulating contact line problems. Based on previous works [4] we revisit our finite-width interface model and introduce a new stress boundary condition at the wall.

I. Introduction

SPH [5] offers a powerful framework to model complex multi-phase phenomena due to its Lagrangian formulation. Using particles as discretization points and advecting them with the flow it is straightforward to introduce multiple types of particles of different phases and include phase interactions such as surface tension forces. By the nature of the method no interface capturing is required and the method is mass and momentum conservative (note, we use the mass conserving density summation form).

Morris [6] proposed a multi-phase SPH model based on the continuum surface force model (CSF) [1] to account for surface tension effects. He applied this model to isolated drops and analysed capillary waves. But this method does not conserve momentum and the calculation of the curvature is cumbersome.

Another approach to model the surface-tension effects on a macroscopic scale without the need of calculating the curvature is presented in Hu and Adams [4]. There, a stress tensor is calculated from the color-index gradients and the resulting surface tension forces conserve linear momentum. They showed that this method captures the dynamics of isolated drops in shear flows and presented a three-phase interaction with triple junction. Fundamentally different, Nugent and Posch [8] model the surface-tension effect with microscopic inter-phase attractive potentials. This method is appealing since simple pair-wise interactions are introduced without the need of a color-index gradient calculation. But on the other hand the remaining parameter in the model have to be calibrated as there is no analytical relation between the resulting surface tension coefficient and the model parameter. Tartakovsky and Meaking [9] proposed a similar method and studied the influence of contact angles on flows through bifurcations. Recently, Das and Das [2] used SPH to simulate equilibrium shapes and contact angles of sessile drops. They used a CSF model and adjusted the position of the wall-nearest SPH particles according to the static contact angle as found from the Young-Laplace equation to impose the equilibrium contact angle. Furthermore they showed that their results could be improved with a diffuse-interface approximation. But consequently, the thickness of the transition region along the interface is doubled and it is not clear how this method performs in dynamic situations since static equilibrium angles are imposed at the contact line.

In this work we propose an extension of the original method of Hu and Adams [4] to simulate contact angle problems. Using the original formulation to simulate the equilibrium contact angle and shape of a drop on a flat surface we achieved already physically reasonable results. That means the wetting or non-wetting behaviour of the fluid on the wall was represented correctly according to the surface-tension coefficients. But analysing the equilibrium state we found comparably strong spurious currents close to the triple point (in two dimensions the triple line or contact line reduce to a triple point or contact point, respectively), see Fig. 1.

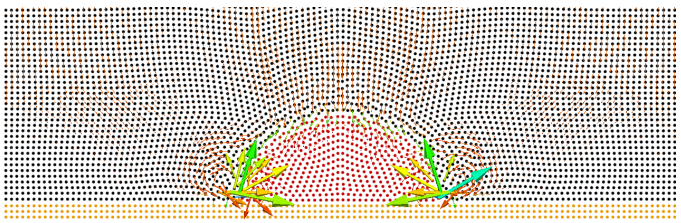


Fig. 1. Velocity vectors showing spurious currents at triple point.

Particles close the triple line are continuously accelerated since the stress singularity is not discretized correctly and consequently the kinetic energy does not decrease. To solve this problem we introduced a new stress boundary condition at the wall that requires only a simple extrapolation of the nearest adjacent phase. Then, the stress at an interface particle is calculated based on its real neighbour particles and its interpolated image particles. With this new approach we are

SPH non-Newtonian Model for Ice Sheet and Ice Shelf Dynamics

A. M. Tartakovsky and W. Pan
Computational Mathematics Group
Pacific Northwest National Laboratory
Richland, USA
Alexandre.Tartakovsky@pnnl.gov

J.J. Monaghan
School of Mathematical Sciences
Monash University
Vic 3800, Australia
joe.monaghan@monash.edu

***Abstract*—We propose a new three-dimensional smoothed particle hydrodynamics (SPH) non-Newtonian model to study coupled ice sheet and ice shelf dynamics. Most existing ice sheet numerical models use a grid-based Eulerian approach, and are usually restricted to shallow ice sheet and ice shelf approximations of the momentum conservation equation. SPH, a fully Lagrangian particle method, solves the full momentum conservation equation. SPH method also allows modeling of free-surface flows, large material deformation, and material fragmentation without employing complex front-tracking schemes, and does not require re-meshing. As a result, SPH codes are highly scalable. Numerical accuracy of the proposed SPH model is first verified by simulating a plane shear flow with a free surface and the propagation of a blob of ice along a horizontal surface. Next, the SPH model is used to investigate the grounding line dynamics of ice sheet/shelf. The steady position of the grounding line, obtained from our SPH simulations, is in good agreement with laboratory observations for a wide range of bedrock slopes, ice-to-fluid density ratios, and flux. We examine the effect of non-Newtonian behavior of ice on the grounding line dynamics. The non-Newtonian constitutive model is based on Glen's law for a creeping flow of a polycrystalline ice. Finally, we investigate the effect of a bedrock geometry on a steady-state position of the grounding line.**

I. Introduction

Mathematical modeling of ice sheet dynamics is usually complicated by the non-linearity of governing equations and boundary conditions. A grid-based Eulerian discretization of governing equations has been mostly used in previous studies of ice sheet dynamics [1]–[4]. However, an Eulerian grid-based discretization has several disadvantages: it requires a complex front-tracking technique; and it has limited capability to handle large material deformations. As a consequence, it is common to use various approximations of the momentum conservation equation [5]. These approximations replace the momentum conservation equation with a steady-state Stokes equation and assume the shallowness of ice sheet/shelf and thereby allow linearization and other significant simplifications of the momentum conservation equation [3]. The most commonly used approximations are the first-order shallow ice approximation [1] and the shallow shelf approximation [2], and they may lead to significant errors under certain conditions. Examples include ice sheets with large aspect ratio and/or large bedrock slope; tidewater glaciers; ice shelves; ice streams; surge dynamics; the dynamics of flow across the grounding line; and the dynamics in the vicinity of ice sheet divides [5]. Recently, several higher-order approximations have been proposed [3], [4], [6]–[9] to partially improve the predictive ability of grid-based models.

In typical ice sheets, the accumulation of ice occurs at the top of mountains and then the ice flows down the bedrock under the force of gravity. Ice is less dense than water, and when ice sheets terminate in the ocean, the ice tends to detach from the bedrock, and float on the water surface forming an ice shelve. The location of the detachment is called the grounding line. Since different approximations of the momentum conservation equation are used to model ice sheet and ice shelf, there is a significant challenge in coupling them at the grounding line. That fact that the position of the grounding line is unknown, makes the coupling a highly non-linear problem. Various assumptions were used in the past to facilitate the coupling. For example, [10] assumed the continuity of the depth-averaged longitudinal stresses across the grounding line, [11] suggested that the grounding line is tangential to the bedrock and [12] had to use a free slip conditions at the grounding line to obtain a unique solution. At the moment there is no consensus about the appropriate treatment of, and nor reliable models for, three-dimensional time dependent dynamics of the grounding line [13].

In this work, we propose a three-dimensional smoothed particle hydrodynamics (SPH) non-Newtonian model to study the coupled ice sheet and ice shelf behavior. This model is an extension of our two-dimensional model that treated ice as a viscous Newtonian fluid [14] and solves full momentum conservation equation for both ice sheet and ice shelf. The SPH model treats ice sheet and ice shelf as a whole, and there is no need for coupling. As any Lagrangian particle method, SPH does not require interface-tracking algorithms for modeling free-surface (e.g., [15], [16]) and moving-boundary problems (e.g., [17]–[19]), and is very efficient for ice sheet and ice shelf modeling. We verify the numerical accuracy of the model by simulating the Poiseuille flow, a plane shear flow with a free surface and the propagation of a blob of ice (modeled as a non-newtonian fluid with Glen's rheology)

Simulation of film and droplet flow on wide aperture fractures using Smoothed Particle Hydrodynamics

Jannes Kordilla, Tobias Geyer
Applied Geosciences
Georg-August-Universität Göttingen
Göttingen, Germany
jkordil@gwdg.de

Alexandre Tartakovsky
Computational Science and Mathematics Division
Pacific Northwest National Laboratory
Richland, USA
alexandre.tartakovsky@pnnl.gov

***Abstract*—Simulation of flow in fractured porous media represents a challenge due to the highly non-linear dynamics of fluid-air interfaces. Here we present small-scale flow simulations on wide aperture fractures using a modified three-dimensional multiphase SPH model [1]. The model is enhanced to include the effects of random thermal noise and able to reproduce a wide range of wetting conditions and Reynolds numbers encountered in laboratory experiments using pairwise fluid-fluid and solid-fluid interaction forces. Static and transient flow dynamics are compared to empirical and semianalytical solutions: (1) Droplets in a critical state are in agreement with laboratory experiments of [2]. (2) Well-defined random thermal noise is introduced via the fluctuation-dissipation theorem and its effect on dynamics of droplets in a critical state is investigated. (3) Transient flow dynamics on dry surfaces are validated using the dimensionless relationships established by [3] and compared to (4) dynamics on prewetted surfaces where flow velocities are shown to be nearly tripled. Finally we establish flow regimes and occurence of trailing films on initially dry fracture surfaces based on dimensionless scaling parameters and Reynolds numbers.**

I. Introduction

Aquifers are the largest water reservoirs for continental freshwater. Groundwater recharge depends on the geometrical and hydraulic properties of the vadose zone (rock formations between surface and water table) where pore space is only partially filled with water. The underlying saturated zone, delineated by the water table, may reach depths of several hundred meters and therefore is coupled to the hydrodynamic state of the unsaturated zone. Quantification of recharge and water travel times through the unsaturated zone are of importance for understanding of large-scale hydraulic behavior, transport of contaminants, management of groundwater resources and nuclear waste repositories ([4], [5]).

The large heterogeneity of the hydraulic parameter field found in fractured geological media represents a challenge for numerical modeling of flow and transport, especially in the unsaturated zone, where high flow intermittency ([6], [7], [8]), preferential pathways ([9], [10], [11]), and complex interaction of porous matrix and fractures have to be considered ([12], [13]). Most macro-scale effective unsaturated flow models underestimate flow velocities and travel time distributions on local scale even though they might be adequate to simulate catchment scale dynamics.

Small scale flow dynamics in fractures are governed by the complex interplay of body and surface forces resulting in several flow regimes such as absorbed films ([14]), droplets ([3], [15], [7]), rivulets ([16] , [17]) and traveling liquid waves which all contribute to the rapid movement of water through the unsaturated zone and interaction with the porous matrix system.

Here we want to present simulations of flow in fractured media in order to gain a deeper understanding of the complex flow dynamics and show the versatility of Smoothed Particle Hydrodynamics in this context.

II. Method

In the following we give a brief description of our model and the governing equations. Detailed derivations and approximations involved in the SPH method can be found, for example, in ([18], [19]). We use an SPH discretization of the Navier-Stokes equations following [1]:

$$\begin{aligned}\frac{d\mathbf{v}_i}{dt} = &-\sum_{j=1}^{N} m_j \left(\frac{P_j}{\rho_j^2} + \frac{P_i}{\rho_i^2}\right) \boldsymbol{\nabla} W_i(|\mathbf{r}_{ij}|, h) \\ &+ \boldsymbol{\eta}_{ij} + \mathbf{g} + m_i \sum_{j=1}^{N} \mathbf{F_{ij}}\end{aligned} \tag{1}$$

where $\mathbf{v}$ is the particle velocity, t is the time, P and ρ denote pressure and density, m is mass of particle i and $\mathbf{g}$ is the gravitational acceleration. Following [20] a fourth-order weighting function W_i is employed where the support of W is set to $h = 1.0$ and all particles are assumed to have the same mass, $m_i = 1.0$ for computationally efficiency. All variables in the SPH model are given in consistent model units. The viscosity term $\boldsymbol{\eta}_{ij}$ is given by [21] as

$$\boldsymbol{\eta}_{ij} = 2\mu \frac{\mathbf{v}_{ij}}{\rho_i \rho_j r_{ij}} \frac{\partial W}{\partial r_{ij}}. \tag{2}$$

Application of SPH to erosion and excavation problems on the examples of jet grouting and offshore engineering

Bozhana Stefanova, Julian Bubel, Jürgen Grabe
Institute of Geotechnical Engineering and Construction Management
Hamburg University of Technology
Harburger Schlossstraße 20, 21079 Hamburg
bozhana.stefanova@tu-harburg.de, julian.bubel@tu-harburg.de, grabe@tu-harburg.de

Abstract— **The Smoothed Particle Hydrodynamics (SPH) has undergone a significant development during the past few decades and has been used for a wide range of research topics. On the field of geotechnical engineering, it especially benefits from its ability to deal with large deformation and different materials or rather phases way much better than conventional mesh methods. This paper focuses on the soil-fluid interaction on the examples of jet grouting and lowering of a gravity foundation of an offshore wind turbine. The simulation of jet grouting aims to reproduce the correlation between soil and fluid stress as well as the mechanism of dissolving the soil under high pressure. The lowering of gravity foundations, on the other hand, induces additional flows, which may erode the soil underneath the foundation. Consequently, inclination of the foundation may occur and the stability of the wind turbine may be affected. Both processes are assumed to be due to the accumulation of excess pore water pressure and soil-fluid interaction. The results show the coherence between pore water pressure and soil dissolution as well as the feasibility and the potential of different soil models with SPH on geotechnical problems.**

I. INTRODUCTION

The interaction between soil and water is of importance for a wide range of geotechnical problems. This paper focuses on two example cases: jet grouting and lowering of a gravity base foundation for an offshore wind turbine. Jet grouting is a common method for ground improvement and stabilization, which uses high pressure streams to break up and cement the soil. The simulation of this process involves different phases as well as large deformations and pressure and velocity gradients. The lowering of a gravity base foundation induces additional flows, which may damage the foundation pit and cause inclinations of the offshore wind turbine. The numerical analysis of both processes involves a soil-fluid-interaction. For this purpose, the smoothed particle hydrodynamics are used. This meshfree method has been established in many research domains, including fluid and solid mechanics as well as hydrodynamics and also geotechnical engineering.

II. SPH

This section outlines the numerical model used for the simulations in this paper. The code Gadget $^{H}2^{O}$ ([10], [11], [12]) is applied for both jet grouting and lowering of a gravity foundation. It is a modification of the cosmological code Gadget2 ([8], [9]). Some adjustments in the code were also made in order to obtain better results for the soil.

A. Conservation equations

Gadget $^{H}2^{O}$ uses the conservations equation for mass (1) and momentum (2) in their SPH approximation with the Kernel function W_{ij} ([4],[5]).

$$\frac{D\rho}{Dt} = \sum_{j=1}^{N} m_j (v_i^\alpha - v_j^\alpha) \frac{\delta W_{ij}}{\partial x_i^\alpha} \tag{1}$$

where i stands for a focal particle, j for a neighbour particle. The particle mass is defined by m, its density by ρ and the velocity by v. x is the position of a particle.

$$\frac{Dv_i^\alpha}{Dt} = \sum_{j=1}^{N} \left[m_j \left(\frac{\sigma_i^{\alpha\beta}}{\rho_i^2} + \frac{\sigma_j^{\alpha\beta}}{\rho_j^2} \right) \right] \frac{\delta W_{ij}}{\partial x_i^\beta} + \frac{f_i^\alpha}{\rho_i} \tag{2}$$

where σ marks the stress and f an external force.

In contrast to the studies of Ulrich ([10], [11], [12]), a different definition for soil stress is also used in this paper.

B. Treatment of fluids

A Newtonian fluid is considered in this paper. The stress tensor consists of isotropic stress and viscous stress, which can be expressed by the viscosity and the strain rate tensor:

$$\tau = \mu\varepsilon \tag{3}$$

The isotropic stress is calculated by Tait's pressure equation as in [11]. Furthermore, a large eddy simulation is employed to describe turbulence. A XSPH smoothing technique is applied in order to meet disturbances in flow, especially due to high pressure gradients.

C. Treatment of soil

The definition of soil in Gadget $^{H}2^{O}$ was employed for the first simulations. It contains a definition of soil particles as a viscous fluid, whose viscosity is described by a criterion similar to that of Mohr-Coloumb. It also allows the development of a boundary layer between the soil and the water phase. This model has provided some reliable

A simple SPH model of Water-Soil Interaction in Porous Media

Christian Ulrich and Thomas Rung
Institute for 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 evaluation of a simple SPH two-phase flow model for water/soil-interaction to capture the change of the soil strength in porous media exposed to seepage flow. The soil-phase is modeled in line with the Mohr-Coulomb yield-stress criterion together with a combined elastic-solid/fluid approach. The framework is supplemented by an apparent cohesion model that is governed by the saturation of the porous media. A simple Darcy-approach is used to mimic the seepage flow through the soil skeleton in combination with baseline material properties, e.g. capillarity, porosity, gravity and pressures. A comparison with experimental references reveals that the approach can capture the principal mechanism at minor computational surplus and modest additional modelling efforts.**

I. Introduction

The interaction of water and soil poses problems to different areas of marine and hydraulic engineering. Typical erosive processes dislocate parts of the soil and may therefore weaken the stability of load-bearing soil formations. Such phenomena are usually induced by large relative motions between water and soil which yields a water/soil-suspension layer. Contrary to this, failure of granular material is frequently caused by internal water flows and the resulting change of soil saturation. In this case, the failure mechanism is not necessarily dominated by flow induced forces but by changing soil properties. Especially the effective material cohesion can be significantly reduced for fully saturated soils. Both phenomena might interact, thus numerical studies require simulation tools which can cope with both mechanisms.

While preceding studies [12], [13] focused on the development of a SPH-suspension model and the simulation of related erosions, our present work is devoted to flows within porous media. Accordingly, the paper outlines the extension of the modelling capabilities of the hydrodynamic SPH-code GADGET-$^{\mathrm{H}}2^{\mathrm{O}}$ towards the prediction of seepage flows. The GADGET-$^{\mathrm{H}}2^{\mathrm{O}}$-procedure is a modification of Springel's [11] cosmological TreeSPH-Code GADGET-2. The applied seepage flow description is based on a simple Darcy-approach outlined by Lenaerts [5] that does not require additional fluid particles to represent the pore water. The corresponding flow through the soil skeleton is evaluated with respect to material properties such as capillarity, porosity, gravity and pressures. A variable soil cohesion which depends on the soil particles' saturation is used to account on changing material strength. In addition to the seepage model, the paper presents a combined solid/fluid approach to predict dynamic soil deformations based on work by Leppert [6]. In order to keep the procedure as simple as possible, the Mohr-Coulomb yield criterion is applied in conjunction with Hooke's law for the elastic branch of the solid model. Large deformations are captured by considering the soil as a non-Newtonian fluid. The related viscosity that is derived from the Mohr-Coulomb criterion. The solid/fluid transition is managed by the invariant of the strain rate tensor. The combined model is introduced to overcome problems related to traditional pure fluid formulations (e.g. creeping) which are widely used in SPH studies, e.g. [4], [8], [12].

The remainder of the paper is structured as follows: In section *Computational Model*, the employed governing equations and their respective finite approximations are described, followed by the section *Model evaluation* which shows several validation examples. Final conclusions are summarised in the last section.

II. Computational Model

The 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 the focal particle whereas the subscript *j* refers to its neighbours. Einstein's summation is employed over repeated Greek superscripts.

A. Kernel Function

A standard cubic spline kernel function is used in the present study

$$W_{ij} = W(r,h) = \alpha_d \begin{cases} 1 - 6\left(\frac{r}{h}\right)^2 + 6\left(\frac{r}{h}\right)^3 & 0 \le \frac{r}{h} \le \frac{1}{2} \\ 2(1-\frac{r}{h})^3 & \frac{1}{2} < \frac{r}{h} \le 1 \\ 0 & \frac{r}{h} > 1\,, \end{cases} \tag{1}$$

with the normalisation coefficient $\alpha_d = \frac{40}{7\pi h^2}$ in 2D and $\alpha_d = \frac{8}{\pi h^3}$ in 3D. The relation between the kernel length h and the particle spacing Δ_P is kept constant, i.e. $h = 2.4\,\Delta_P$. The partition of unity might not be maintained, particularly

SPH Simulation of Granular Material Collapses

Enio Paris
Civil and Environmental Engineering Department
University of Florence
Florence, Italy
eparis@dicea.unifi.it

Lorenzo Minatti
CeRAfRi
Florence, Italy
lminatti@dicea.unifi.it

***Abstract*—Results obtained from the application of SPH for simulating granular columns collapses are presented in the paper.**

A constitutive equation for a visco-plastic fluid with a pressure dependent yield stress has been employed for simulating the granular material.

The paper describes how the afore mentioned constitutive equation has been implemented into a SPH model and how numerical issues arising from such implementation have been treated.

The proposed model, taking advantage of well known SPH abilities, can be used to model flows of incoherent granular material with a free surface. The model also represents a first step towards an alternative approach to sediment transport in natural and artificial channels: an extension of the model to multiphase flows might indeed be able to reproduce such phenomena.

The model has been tested by reproducing existing experiments on granular columns collapses in which sand columns of fixed width and different heights, initially contained inside a cylindrical container, were suddenly released.

I. Introduction

A possible way of modeling granular material is the one of treating it as a fluid with a specific rheological law. After a suitable rheology has been chosen, it is in principle possible to use any kind of numerical method for simulating the flow.

A general rheology for granular matter, suitable for any kind of situation, isn't yet available in scientific literature. Nevertheless, a number of models have been proposed, each one of them being able to simulate specific classes of granular flows. In this study, attention has been focused on the rheology proposed by Pouliquen et al. [17] and Jop et al. [9] and its adaptation to SPH.

The rheology attempts to reproduce granular matter's behavior with a non-Newtonian viscosity depending on pressure in order to take into account intergranular shear. The model has been found efficient for dry dense granular flows at low velocities [12], [17]. The rheology is particularly suitable for situations in which grains remain close to each other when moving.

Main features of the considered rheological law are the following:

- It is represented by a non-Newtonian viscous constitutive laws: the resulting momentum equation therefore contains space-varying viscosity.
- The constitutive law shows a yield value for stress: this means that when deformations tend to zero, viscosity will approach infinity. A viscosity regularization technique has to be applied to treat high values of viscosity.

In the present work, the development of a SPH model using the mentioned rheology to SPH is shown together with an application to available experimental data.

II. Constitutive equations for granular materials

The mechanics of an assembly of particles can be very complicated. All the available constitutive equations have a limited range of applicability, which depends among others on velocities, average spacing of grains during motion etc..

In the case of cohesionless granular material, if the grain size is sufficiently large (i.e. $d > 250\ \mu m$) and the interstitial fluid isn't too viscous, the interactions between grains are dominated by contact interactions [12]. This means that mechanical properties of the material depend only on momentum transfer during grains collisions or frictional contacts between grains.

Other variables, such as capillary forces or interstitial fluid-grain viscous interactions may be neglected. Among the various situations comprised under these conditions, which are not unusual in nature, the flow regime is usually divided into three classes depending on the nature of the main interaction between grains [12], [17]. Within each one of these classes, different rheologies have been proposed in literature.

When velocities are very low and grains inertia is negligible, a quasi-static regime occurs. Within this regime, soil plasticity models have been successfully used even in the SPH framework [3].

On the other hand, when the flow intensity is very high and grains are apart from each other, a collisional regime occurs. Interactions are dominated by collisions between grains and it becomes crucial to provide a correct estimate of energy losses during collisions in order to devise an efficient constitutive law for the material. Within this regime the granular matter behavior is similar to the one of a gas and kinetic theories based approaches have been developed [7].

In the intermediate regime, herein called "liquid" regime, it is impossible to use kinetic theories as particles still experience enduring contact and grains inertia becomes important. The studies of Pouliquen et al. [17] and Jop et al. [9] follow from the GDR MiDi research [12] and other related papers and are focused on this regime.

Towards simulations of abrasive flow machining

Christian Nutto, Claas Bierwisch, Hanna Lagger, Michael Moseler
MikroTribologie Centrum μTC, Fraunhofer Institute for Mechanics of Materials IWM
Wöhlerstr. 11, 79108 Freiburg, Germany
christian.nutto@iwm.fraunhofer.de

Sören Höhn, Tina Bremerstein, Annegret Potthoff
Fraunhofer Institute for Ceramic Technologies and Systems IKTS
Winterbergstr. 28, 01277 Dresden, Germany

***Abstract*—We report on the development of coupled SPH and DEM simulations for the investigation of abrasive processes of technical fluids in industrial applications. These suspensions are interspersed with small solid particles responsible for the abrasive wear. In addition, in some industrial applications, the suspensions show viscoelastic behavior. For the numerical treatment of such fluids a numerical scheme of an Oldroyd-B fluid has been implemented into the SimPARTIX® SPH code. However, we can show by simulations of an oscillation rheometer that such a model is only a crude approximation to the behavior of complex polymeric fluids.**

Furthermore we present a technique that allows us to numerically reconstruct the complete three dimensional geometry of single abrasive grains by the use of the discrete element method (DEM). These abrasive DEM grains can then be used in coupled SPH simulations of abrasive processes.

Rheological simulations of a suspension consisting of a Newtonian fluid with varying volume fractions of the DEM grains demonstrate that the simulations agree well with experimentally gathered data. This allows us to confidently use the suspension for simulations of abrasive flow machining.

I. Introduction

Important manufacturing techniques of complex geometries in technical materials often rely on the precise finishing of surfaces in order to achieve its designated performance. Obtaining the necessary surface roughness and a sufficiently high material removal rate in finishing processes of hard-to-access surfaces remains a great challenge, especially under the requirements of energy efficiency in the conducting process. The complex interactions between the fluid and the abrasive grains embedded within the fluid cause great difficulties in optimizing the process parameters for satisfying results.

The finishing processes often include cutting, burring and surface polishing within technical components exhibiting complex geometries. An example process is abrasive flow machining (AFM). In spite of the huge range of application of this, the difficulties to adjust the parameters correctly for each workpiece geometry prevents its utilization in a greater variety of industrial applications. For abrasive flow machining, there are only few numerical models available and the applied physical models are strongly simplified, e.g., abrasive grains are considered to be homogeneous, therefore neglecting their individual structure. Until now, there are no numerical simulations available that take into account the individual grain characteristics and their highly dynamic interactions with the workpiece. However, in order to optimize the wear of the material, an explicit approach of including individual particles in the abrasive suspension is promising.

AFM processes are commonly used for the finishing of internal surface geometries. Depending on the size of the inlet and the internal structure, fluids of different viscosity are applied. For smaller geometries, fluids of low viscosity are used, while for larger inlets, highly viscous and viscoelastic, flowable polymeric composites are utilized. In both cases the abrasive medium is embedded within the fluid. This abrasive suspension is forced to flow along the contours of the technical component. The relative motion between the abrasive medium and the component then causes the wear of material. Experimentally conducted research has focused on the characterization of the abrasive medium and the investigation of the wearing behavior and quality. The aim of optimizing the process parameter for a given workpiece in industrial applications is usually only achieved by trial-and-error.

The long term goal of this work is to study numerically the process of abrasive flow machining at a scale where the actual interaction between the abrasive grain and the technical component occurs. Knowing the interaction at the grain size scale, we intend to develop analytical models for the removal rate at larger scales that can then be implemented in numerical codes operating at a length-scale of the workpiece. This would allow to derive process parameters and to design suspensions with specific abrasive characteristics for a given industrial application. Our goal is to develop numerical tools and models that include the precise coupling between various viscous and viscoelastic polymeric suspensions, the abrasive grains and the technical component.

In section II, we will shortly review the underlying physical models and the SPH framework applied in our numerical approach. Section III introduces the concept of oscillation rheology for the characterization of viscoelastic suspensions and their numerical approximation. The workflow for the dig-

A modified Godunov SPH method for materials with strength

Adam Connolly
Department of Aeronautics
Imperial College London
London, SW72AZ
a.connolly08@imperial.ac.uk

Lorenzo Iannucci
Department of Aeronautics
Imperial College London
London, SW72AZ
lo.iannucci@imperial.ac.uk

***Abstract*—In this paper a modification of the Godunov Smoothed Particle Hydrodynamics (GSPH) method of [11] is presented for isotropic materials with strength. This modification removes the need to rotate the stress-tensor for interacting particles and solve the Riemann problem for the transversal wave system. A consequence of this modification is that it allows for calculations of higher spatial order of accuracy via variable reconstruction as well as the use of different kernel functions within each time-step. Results are presented for one and two dimensional test cases and compared with corresponding predictions made using conventional Artificial Viscosity SPH (AV SPH).**

I. Introduction

The use of SPH for simulations of solid dynamics is most commonly focussed on high strain-rate problems such as high velocity impacts, as the mesh free Lagrangian nature of SPH is ideally suited to simulations involving large deformations [6]. High velocity impacts generate shock-waves and in a continuum level numerical scheme the spatial discretization is typically several orders of magnitude larger than the shock-wave thickness. Therefore, in order to maintain a stable solution, the shock-wave must be smeared such that there is a smooth variation in field quantities across the shock. In SPH, this is usually achieved by applying an Artificial Viscosity (AV) [8] to the momentum and energy equations. Several forms of the AV have been presented in the literature [1], [3], [8] with the most common implementation, found in most commercial SPH codes, being that of Monaghan *et al.* [8]:

$$\Pi_{ij} = \frac{-a\bar{c}_{ij}\phi_{ij} + b\phi_{ij}^2}{\bar{\rho}_{ij}} \quad \text{if} \quad \mathbf{v}_{ij} \cdot \mathbf{x}_{ij} < 0 \tag{1}$$

where a and b coefficients control the level of damping introduced by the AV. It has been shown that the choice of a and b can significantly affect the results [6]. In addition, in order to apply an optimal level of damping, a time consuming trial-and-error analysis is required [12] which may be undesirable for the user.

Recently, the Godunov reformulation of the SPH equations have been presented [5], [11], whereby the Riemann problem is solved at the midpoint of interacting particles and thus sufficient numerical dissipation is introduced to allow stable integration. The advantage of these formulations is that no user-defined damping parameters or associated sensitivity analyses are required. While being more conservative, the method of Inutsuka [5] requires the use of the summation density equation in place of the continuity equation and is thus impractical for simulations involving free-surfaces and stiff equations of state, such as that routinely found in solid-dynamics. The method of Parshikov *et al.* [11] , however, uses the Godunov reformulation of the SPH continuity equation and therefore is more applicable to simulations of solid-dynamics. In [11] , Riemann solutions are required for both the longitudinal and transversal wave systems for materials with strength.

This work presents a modification to the GSPH method of Parshikov *et al.* [11] , whereby a time-operator-splitting procedure, based on the method presented by Howell *et al.* [4] , is used to separate the integration routine into a hydrodynamic and a deviatoric stage. The modified method negates the requirement to solve the Riemann problem for the transversal wave system and thus facilitates a higher-order reconstruction of the left and right Riemann states for the longitudinal wave-system. Another consequence of the separation of the integration procedure is that different kernel functions may be used within the temporal discretization. The use of a quadratic kernel for the deviatoric step may help alleviate the compressive instability intrinsic to the Gaussian shaped kernels, while not affecting the accuracy of the interpolation significantly. In addition, a simple modification to the GSPH continuity equation [11] is shown to enhance the accuracy of the density field in simulations involving particles with different masses [10]. Some one and two dimensional tests are presented and compared with standard AV SPH.

II. Time-operator splitting procedure

In [4] the Cauchy stress tensor is decomposed into the diagonal and non-diagonal components (2) and the integration procedure operates sequentially on each component.

$$\sigma_i^{\alpha\beta} = \tau_i^{\alpha\beta} - P_i\delta^{\alpha\beta} \tag{2}$$

Dynamic refinement for SPH simulations of post-failure flow of non-cohesive soil

Yaidel Reyes López
Dept. of Computer Sciences
KULeuven
Leuven, Belgium.
yaidel.reyeslopez@cs.kuleuven.be

Dirk Roose
Dept. of Computer Sciences
KULeuven
Leuven, Belgium.
dirk.roose@cs.kuleuven.be

Carlos Recarey Morfa
CIMCNI
UCLV
Santa Clara, Cuba
recarey@uclv.edu.cu

***Abstract*— We discuss the application of a dynamic refinement procedure to reduce the computational requirements of an elastic-plastic model to simulate non-cohesive soil. In the refinement procedure, the diagonal components of the strain tensor are used as the criterion for the refinement, and an SPH particle is refined by replacing it with new daughter particles which are located according to a square pattern centered at the refined particle position. The position of the daughter particles and their smoothing distance are determined such that the error introduced due to the refinement is kept small. Further, possible numerical instabilities are identified and avoided by using adequate refinement parameters. Obtained results are compared with those of simulations using a fine discretization in the whole domain. The comparison shows a good agreement, while the savings in computational time and memory consumption are considerable.**

I. Introduction

Smoothed Particle Hydrodynamics (SPH) has been proved as a valid alternative to mesh-based methods for solving geotechnical problems. Its application in this area has been motivated by the difficulties that arise from applying mesh-based methods, like Finite Elements, to problems involving large deformations or other complicated phenomena that include multi-physics. Recent works in this area include the simulation of seepage failure and erosion [1], water/soil-suspension [2], landslides [3], post-failure flow of soil [4]–[6], and others.

The model proposed by Bui and co-workers in [4] is of particular interest for geotechnical applications since it considers the plastic behavior of the material, which is very important for accurate soil simulations. However, when using this model, the simulations are computationally demanding.

In this work we discuss the application of a dynamic refinement procedure to reduce the computational requirements of the method while still achieving a similar accuracy. The application of this strategy allows the simulation of larger physical domains using the same computational resources.

II. SPH formulation for soil

In this section we briefly describe the SPH formulation of an elastic-plastic soil model based on the Drucker-Prager yield condition. More details can be found in the original work by Bui et al. [4]. In the formulas, Greek superscripts denote Einstein's notation, and Latin subscripts i and j denote individual particles.

A. Density and momentum equations

The continuity equation is applied to evolve the soil density ρ for particle i, using the expression

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

where m, v and x are the mass, velocity and position of a particle, and W is the cubic spline kernel function.

The momentum equation used is described by

$$\frac{Dv_i^\alpha}{Dt} = \sum_{j=1}^{N} m_j \left(\frac{\sigma_i^{\alpha\beta}}{\rho_i^2} + \frac{\sigma_j^{\alpha\beta}}{\rho_j^2} - \Pi_{ij}\delta^{\alpha\beta} \right) \frac{\partial W_{ij}}{\partial x_i^\beta} + F_i^\alpha, \tag{2}$$

with $\delta^{\alpha\beta}$ the Dirac delta function, σ is the total stress tensor and F corresponds to external forces (gravity in this work). The artificial viscosity term Π is included to reduce numerical instabilities. It is given by

$$\Pi_{ij} = \begin{cases} \frac{-\alpha c \mu_{ij} + \beta \mu_{ij}^2}{\bar{\rho}_{ij}} & : \mathbf{v}_{ij} \cdot \mathbf{x}_{ij} < 0 \\ 0 & : \text{otherwise} \end{cases}, \tag{3}$$

where h is the smoothing distance, $\mu_{ij} = \frac{h \mathbf{v}_{ij} \cdot \mathbf{x}_{ij}}{\mathbf{x}_{ij}^2 + 0.01h^2}$, $\bar{\rho}_{ij} = \frac{\rho_i + \rho_j}{2}$, the notation $A_{ij} = A_i - A_j$ is used, and α and β are constants that are taken to be ~ 0.1 as suggested in [4].

B. Stress-strain relationship within the elastic-plastic model

Within the elastic-plastic model, the strain rate tensor

$$\dot{\varepsilon}^{\alpha\beta} = \frac{1}{2} \left(\frac{\partial v^\alpha}{\partial x^\beta} + \frac{\partial v^\beta}{\partial x^\alpha} \right), \tag{4}$$

is composed of two parts, the elastic $\dot{\varepsilon}_e^{\alpha\beta}$ and the plastic $\dot{\varepsilon}_p^{\alpha\beta}$ strain rate tensors, resulting in

$$\dot{\varepsilon}^{\alpha\beta} = \dot{\varepsilon}_e^{\alpha\beta} + \dot{\varepsilon}_p^{\alpha\beta}. \tag{5}$$

The elastic term is calculated by applying Hooke's law,

$$\dot{\varepsilon}_e^{\alpha\beta} = \frac{\dot{s}^{\alpha\beta}}{2G} + \frac{1 - 2\upsilon}{3E} \dot{\sigma}^{\gamma\gamma} \delta^{\alpha\beta}. \tag{6}$$

Flow prediction of Reactive Rotational Molding using Smoothed Particle Hydrodynamics Method

S. Riviere, S. Farzaneh, A. Tcharkhtchi.
PIMM laboratory
Arts et Métiers ParisTech, CNRS-UMR 8006
Paris, France
sylvain.riviere-4@etudiants.ensam.eu

S. Khelladi, F. Bakir.
DynFluid laboratory
Arts et Métiers ParisTech
Paris, France
sofiane.khelladi@ensam.eu

***Abstract*—Prediction of the flow during reactive rotational molding is of great interest to optimize process conditions and wall thickness distribution of the molded part. Simulation is one of the ways; based on smoothed particle hydrodynamics method we performed 2D and 3D simulations in order to observe the influence of the change of viscosity on the flow, due to the chemical reactions. Adhesion of the polymer on the mold surface is modeled by new boundary conditions.**

I. Introduction

Rotational molding is a process for manufacturing the hollow plastic parts from few cm^3 to several m^3 [1]. The mains advantages of this method are; no residual stresses in final parts, no weld line and no heterogeneity of material behavior compared to other polymer processes such as injection molding or blowing extrusion. The mains weaknesses are the time to heat and melt the polymer powders and the time to cool and solidify the polymer. Consequently, the cycle time to produce a plastic part is long (0,5-1h), depending on the dimensions of the part. Nowadays rotational molding of thermoplastics is widely used in industry; nevertheless there are only few applications for reactive materials due to the complexity of the polymer transformation. The first work carried out in this field outlined these problems [2], [3]. Reactive Rotational Molding (RRM) has several advantages compare to traditional rotomolding of thermoplastic powders: process cycle time is shorter; raw material is less expensive because polymerization occurs during processing and high performance polymers may be used such as thermosets (polyepoxy [4] and polyurethane [5], [6]), thermoplastic (Polyamide 6 [7]) or blends (polyepoxy/PMMA [8]). However implementation of RRM is not easy because of chemical reactions. Fig. 1 shows the variation of viscosity related to the rheology evolution during polymerization. According to reaction rate, the viscosity increases and we can distinguish several types of flows. In the early step of the chemical reactions, the viscosity is low and the material can not adhere to the mold (pool). At that point, the viscosity begins to increase and the material starts to adhere but due to gravity, it falls down; this is the cascading flow. Then we can observe the rimming flow where the material is well distributed on the mold surface but a wave phenomenon is still occurring on the fluid surface. Finally we can observe the solid rotation; this is the desired flow where the fluid is uniformly distributed on the internal mold surface. This flow must appear before the gel point of the material (Fig.1-c), over this point the material becomes stiff and there is no more flow. The chemical reactions are not completed but the part can be removed from the mold for the post-curing process. It is important to note that these different kinds of flows are dependent on the viscosity and the rotational speed. Here we can clearly see the importance of the simulation to predict the flow according to the mold geometry, the rotational speed of both axes and the mold temperature which influences the reaction rate. Before starting researches in this field, no work was found in the literature. First simulations were carried out using the Volume Of Fluid (VOF) method with a commercial software [9], [10], these first simulations showed the influence of different parameters on cycle time (rotation speed, amount and viscosity of polymer). However the results were not good enough for a realistic prediction of the material flow mainly because of the free surface representation and the absence of a good rheokinetical model for the evolution of viscosity. Mounif [10] started to develop a solver based on Smoothed Particle Hydrodynamics (SPH) method to simulate RRM. This lagrangian particular approach is well adapted to simulate free surface flows such those occurring during RRM. SPH method is already used to model material processing, notably metal forming [11] and injection molding [12]. Since Mounif's work we improved the initial solver to be able to simulate 2D and 3D flows. To simulate the variation of viscosity according to chemical reactions, we added a rheokinetical model based on experimental results and we implemented a new type of boundary condition to model the adhesion of the reactive fluid on the mold surface.

II. Method

In pre-gel phase, where the viscosity is low, the fluid flow is characterized by a free surface flow with the competition of two main forces: gravity and viscosity. The fluid is modeled as an incompressible viscous Newtonian fluid because rotational speed is low (1 to 10 rpm) and shear force is negligible. In SPH method, material is represented by n particles of masses m_i, velocity v_i and others hydrodynamics properties such as

SPH Simulations and Experiments of Sloshing in an Egg-Shaped Shell

Jayson Grant[1,2], Mahesh Prakash[2], S. Eren Semercigil[3] and Özden F. Turan[3]

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

[2]CSIRO Mathematics, Informatics and Statistics
Clayton, AUSTRALIA

[3]HG Research and Consulting
Victoria, AUSTRALIA

Tall structures, such as towers and bridges, are subject to wind and earthquake loads which can cause them to oscillate at excessive magnitudes. Liquid sloshing absorbers can be used to suppress these excessive oscillations by tuning the frequency of the sloshing to the critical frequency of the structure to be controlled. These absorbers are simple structures consisting of a partially full container of liquid with a free surface. Tuning ensures that significant amounts of harmful energy can be extracted from the structure to the sloshing liquid. However, there needs to be a rapid means of dissipating this energy to avoid its returning back to the structure (then back to the liquid periodically).

The raw hen's egg uses liquid sloshing to efficiently dissipate energy to protect its embryo. Hence, the potential to implement the egg's unique shape as a liquid sloshing absorber design for structural control purposes is the main focus of this study.

3-dimensional numerical predictions in this paper are undertaken using Smoothed Particle Hydrodynamics (SPH). These numerical predictions are validated with experimental observations. SPH will also be used to analyse complex liquid free surface shapes and identify the natural egg's effective energy dissipation characteristics.

I. INTRODUCTION

Liquid sloshing absorbers are simple structures consisting of a partially full container of liquid with a free surface. Sloshing within these absorbers can be used to suppress excessive oscillations for structural control purposes. Investigation of liquid sloshing absorbers for structural control purposes has attracted considerable attention in the literature [1-4].

Liquid sloshing absorbers require tuning for optimum energy dissipation. This is achieved by tuning the frequency of the sloshing liquid to the critical frequency of the structure to be controlled. The free surface length and liquid height are key parameters for tuning a rectangular shaped sloshing absorber. Rectangular absorbers are the most common shape studied in the literature [3-5]. Exploring absorber shape has received considerably less attention for structural control purposes. Apart from rectangular, these studies have been mostly limited to cylindrical [6] and trapezoidal shapes [7]. So and Semercigil [8] have investigated a raw hen's egg as an effective energy dissipater through experimental observations. Marsh et al. [9] explored an egg shaped cylinder to identify physical events responsible for effective energy dissipation.

The raw hen's egg uses liquid sloshing to efficiently dissipate energy to protect its embryo. Transient displacement histories of a raw and boiled egg are displayed in Figure 1(a) [8]. The raw egg's energy dissipation is substantially higher when compared to a boiled egg as displayed in Figure 1(a) [8] where the boiled egg takes approximately 10 times longer to cease oscillations compared to the raw egg when both are released from the vertical (long axis) position.

So and Semercigil gave evidence to suggest that replacing the content of the egg with water can marginally enhance energy dissipation making the egg's design a lot more practical as a liquid sloshing absorber. Also surprisingly the egg seems to be insensitive to fill volume having similar times for the egg to cease oscillations for different fill volumes. This makes it very attractive for design purposes as tuning for a rectangular shaped absorber requires specified fill volumes to achieve effective energy dissipation. Substantial variations have been recorded in the effectiveness between deep and shallow rectangular liquid sloshing absorbers [10].

Numerical predictions in this study are undertaken using Smoothed Particle Hydrodynamics (SPH) due to its ability to simulate liquids without the need for a mesh structure. SPH can accurately capture complex free surface behaviour because of its Lagrangian nature [11, 12]. Due to prohibitive computational requirements Marsh et al. [9] was limited to two-dimensional predictions of an egg shell using SPH. Here SPH is used to model a three-dimensional hen's egg and the results are compared with experiments from So and Semercigil [8]. The simulations are also used to investigate the natural egg's effective energy dissipation characteristics and provide details, such as liquid velocity flow fields, which are not possible through experimental observations. Conclusions from Marsh et al. [9] are compared with this study to identify if there are similarities between two and three-dimensional predictions.

II. EXPERIMENTS

Experimental displacement history data for numerical validation was acquired from So and Semercigil [8]. The study gave evidence that the raw egg possesses efficient energy dissipation while keeping the white and the yolk in separate membranes as displayed in Figure 1(b) [13].

On the use of a time-dependent driving force in SPH simulations

Sh.Khorasanizade, J.M.M. Sousa
Universidade Técnica de Lisboa,
Instituto Superior Técnico, IDMEC
Lisbon, Portugal
shahab.khorasanizade@ist.utl.pt, msousa@ist.utl.pt

J.F. Pinto
Universidade de Lisboa, Faculdade de Farmácia, iMed.UL
Lisbon, Portugal
jfpinto@ff.ul.pt

***Abstract*— The paper focuses on the development and application of a new strategy to deal with the need of imposing a driving force in the simulation of internal, incompressible flows employing Smoothed Particle Hydrodynamics (SPH) together with periodic boundary conditions. With the exception of simple flows such as the plane channel problem, the value of the driving force is not known *apriori*. The usual procedure involves adjusting the value of the driving force by trial-and-error so that the desired value of the mass flow rate is met. Here we propose the use of a time-dependent driving force, which value is self-adjusting to the problem along the simulation by simultaneously enforcing global mass conservation. In the present work, we analyze the impact of this new procedure on the performance of SPH simulations of flows exhibiting self-sustained oscillations.**

Two test cases have been considered in this analysis: a) flow in a periodically grooved channel; b) flow around a square obstacle in a periodic channel. Both cases are considered in supercritical state, hence exhibiting self-sustained oscillations. In the former, finite amplitude oscillations develop due to the interaction of the complex flow dynamics inside the groove with the separated shear layer above, which via its impingement on the groove edge generates a pressure-feedback mechanism. In the latter, the well-known von Kármán vortex street develops in the wake of the obstacle, displaying periodic vortex shedding. It must be noted that, due to the nature of the aforementioned mechanisms and the different approaches to calculate the pressure field, the choice of using either a Weakly Compressible SPH (WCSPH) or Incompressible (ISPH) formulation is expected to penalize the ability of properly resolve all the physical features of these flows. Calculated results are compared with Finite Volume (FV) simulations with the same and higher resolution calculations.

I. INTRODUTION

Smoothed Particle Hydrodynamics (SPH) is a Lagrangian technique where the medium under observation is modelled with discrete number of particles, whose movement describes the motion inside the medium. It was initially developed to deal with astrophysical problems [1,2] and has since been expanded to a wide range of areas [3]. To calculate the particle variables, their interaction with the neighbouring ones is accounted for through the use of a kernel function and, subsequently, by making the summation over all the interactions.

The meshless character is a great advantage of SPH over traditional grid-based methods. On the other hand, there are two major methodologies in SPH to simulate incompressible flows. The first, called Weakly Compressible SPH (WCSPH), consists in the approximation of simulating incompressible flows with a small compressibility level [4]. Compressibility is considered by assigning a finite speed of sound to the fluid, which forces the use of smaller time-steps, thus a larger number of iterations is needed. The second methodology, called Incompressible SPH (ISPH), simulates the flow by enforcing an incompressibility condition [5]. Incompressibility is achieved by a divergence-free velocity field, which brings up the need to solve a system of linear equations. Hence, for large problems, the latter methodology becomes computationally expensive [6]. Irrespective of their differences, both of these methodologies suffer from instabilities, which may compromise their accuracy. There are a number of remedies proposed [7, 8, 9] to improve the accuracy and the stability of SPH. Among these we have the procedure known as Modified SPH (MSPH) [8] and the use of a particle shifting algorithm [9], which has been implemented in the present work.

Flows in grooved channels present a wide variety of problems of interest to both theoretical and computational dynamicists [11]. The most striking feature of these flows is their unsteadiness. Such flows arise in a large number of modern engineering applications, such as microchannel emulsification [12] and microchannel heat sinks for high performance cooling of microprocessors [13]. These flows serve as a simple, yet rich, example of separated flow, in which the complex interactions of separated vortices, free shear layers, and wall-bounded shear flows can be examined in detail. In [14] a simulation of the flow in a periodically grooved channel was carried out using a spectral method to cover a significant range of Reynolds numbers (*Re*) in the laminar regime. The critical value of this parameter, beyond which self-sustained oscillations developed in flow, and also the corresponding frequency have been determined in the aforementioned study. Later, Finite Volume (FV)

Incompressible Smoothed Particle Hydrodynamics: proposition and validation of a fully-explicit algorithm.

Daniel A. Barcarolo and David le Touzé
LUNAM Université, Ecole Centrale de Nantes
LHEEA Lab. (UMR CNRS)
Nantes, France
daniel.barcarolo/david.letouze@ec-nantes.fr

Florian de Vuyst
Ecole Normale Supérieure de Cachan
CMLA Lab. (UMR CNRS)
Cachan, France
devuyst@cmla.ens-cachan.fr

Abstract—**Recently several authors have proposed the use of SPH truly-incompressible algorithms, see e.g.[1; 2; 3; 4; 5; 6]applied in different domains of the computational fluid dynamics. Within this technique, based on a projection method, a pressure Poisson equation (PPE) is solved to obtain an incompressible pressure solution. Even though good results are obtained, the method lacks of CPU performance as an equation has to be solved in an implicit manner (through algorithms like Bi-CGStab, GMRES, etc.) and because only moderately large time steps can be used since the convection remains explicit due to the Lagrangian nature of the method.**

In this article we present a fully-explicit incompressible SPH algorithm where the PPE is solved using a Jacobi-like technique. This method is validated against analytical and literature results as well as against other existing SPH methods for both viscous and free-surface flows. In the proposed model we also use a more precise free-surface detection algorithm and a simpler and as performing stabilizing technique than the ones used in existing incompressible SPH models of the literature.

I. INTRODUCTION

Smoothed particle hydrodynamics (SPH) method is a fully Langrangian, mesh-free method where the fluid domain is discretized by a set of calculation points called particles, where differential operators are evaluated through the use of a kernel function which determinates a range of interaction with the neighbor particles. It has firstly been introduced by Lucy[7]and Gingold & Monaghan[8]and has since been applied to a great variety of flow problems. In order to simulate incompressible flow, there are two main solutions available in the literature: (1) the method called Weakly-Compressible SPH (WCSPH) where the fluid is represented by the compressible Navier-Stokes equations and incompressibility is represented as a pseudo-compressibility where the sound speed is artificially set in a way to keep the density variation below a certain level ($\pm 1\% \rho_0$); or (2) the Incompressible SPH (ISPH) method where the incompressibility of the flow is assured at each time step by solving a pressure Poisson equation to an arbitrary accuracy. Variations of the first approach exist like Riemann-SPH [9] and δ-SPH [10]. The ISPH mentioned in this paper is the one based on the projection method, as originally presented by [1], although some other alternatives exist [4].

Since the introduction of the ISPH method by Cummins & Rudman [1], it has been used to model free-surface flows [2], wave breaking problems with the inclusion of a turbulence model [11], viscous flows [5], and multiphase flows [3], [12]. The method can be divided into three main variants: the first, presented in [1], involves a dependence on the velocity divergence in the right hand side (RHS) of the Poisson equation. This variant is referred to as ISPH_DF in this article (DF standing for Divergence Free). In the second variant, the RHS of the Poisson equation depends on a density prediction as outlined by Shao & Lo [2], referred to as ISPH_DI (DI standing for Density Invariant). The third variant is a hybridization of ISPH_DF and ISPH_DI, solving two Poisson equations (PPE) as described by Hu & Adams [3], and referred to as ISPH_DFDI (Divergence Free and Density Invariant).

A detailed comparison of these three variants has been performed by Xu et al. [6]. They showed that ISPH lacks of accuracy for certain flow problems (free-surface flow mainly), due to errors associated with the truncated kernels. In order to prevent this loss of accuracy, they have proposed a FVPM-like (Finite Volume Particle Method) shifting algorithm [13] applied to the divergence free variant. This technique showed to perform well both for viscous and free-surface flows. The same author has proposed a Peclet number-based free-surface stabilization technique [14] where the viscosity of the particles close to the free-surface is artificially increased. It has shown good results, mainly for wave propagation phenomena. Recently, another approach has been developed by Lind et al.[15] based on Fick's law of diffusion and good results have been achieved for a variety of flows. In [16] a repulsive component of Lennard-Jones potential in the advection equation is used to prevent particles fracturing and therefore stabilize the method.

Other authors proposed to use some alternative corrections to increase the accuracy of the method, involving a greater effort to get in turn a more precise solution of the pressure Poisson equation. Khayyer et al. [17] proposed to correct the kernel gradient by renormalizing it. Large effort has also been made to improve the MPS (Moving Particle Semi-Implicit) method by Khayyer & Gotohin [18]where they used a higher order Laplacian operator and a renormalization-like operator to improve the PPE precision. The MPS method is very similar to Incompressible SPH, having the same solving procedure (projection method) and it is also a particle-based Lagrangian technique.

Unlike for the WCSPH method where free surface conditions are intrinsic provided a globally consistent discretization is used, see Colagrossi et al. [41], when ISPH is applied to free surface flow problems one needs to impose free surface conditions to be able to solve the PPE. The kinematic

Modeling of gravity wave viscous attenuation

A. Colagrossi
CNR-INSEAN
00128 Rome, Italy
CeSOS, NTNU,
Trondheim, Norway
a.colagrossi@insean.it

A. Souto-Iglesias
Naval Architecture Dept.(ETSIN)
Technical University of Madrid (UPM)
28040 Madrid, Spain
antonio.souto@upm.es

M. Antuono
S. Marrone
CNR-INSEAN
00128 Rome, Italy.
m.antuono,s.marrone@insean.it

***Abstract*—Viscous attenuation in a standing wave is analyzed with weakly compressible SPH. The dissipation mechanisms are explored firstly by comparing the kinetic energy decay with an approximate analytic solution, secondly by considering a decomposition of the mechanical energy variation which highlights the effect of the weakly compressibility assumption and thirdly by analyzing, using the enstrophy, the dissipation related to the generation of spurious vorticity. The roles of the Reynolds number and the number of neighboring particles are analyzed.**

I. Motivation

Computing the dissipation due to wave-breaking remains a challenging problem in the computational fluid mechanics context. Only semi-empirical approaches like that of Sun *et al.* [1] can be found in the literature. Since no boundary condition is needed at the free surface [3], it is not necessary to explicitly detect the free surface geometry, and weakly compressible SPH then becomes a promising technique in approaching this difficult problem [2]. This is a major advantage when dealing with highly distorted free surface flows because no numerical diffusion is introduced through the interface tracking. Nonetheless, dissipation mechanisms related to free surface flows have to be accurate.

In Colagrossi *et al.* [4] it was demonstrated that in the continuum, Monaghan, Cleary and Gingold's [5] viscosity formula (MCG formula from now on) provides the correct viscous dissipation for free surface flows. However, it is necessary to check to what extent this is the case when such continuous formulation is discretized. This viscosity formulation was originally devised as an artificial viscosity but was later shown to be a consistent Newtonian viscous term for incompressible flows [6].

The discretized formulation is applied in the present work to model the evolution of a standing wave. On one hand this choice is justified because this was one of the cases considered in Colagrossi *et al.* [4] continuous analysis. But more importantly, a standing wave does not propagate meaning that the computational domain can be restricted. On the other hand, an external wave maker and a much wider domain are necessary to model progressive waves, as shown in [7]. Due to the huge computational effort required, this is a major drawback when convergence properties of the discretized formulations are analyzed. The work of Antuono *et al.* [7] is relevant for its study of the dissipative properties of the artificial viscosity term. Another important reason for the standing wave choice is that an approximate analytic solution is available for its attenuation.

Other authors have focused on the the dissipative properties of discrete SPH implementations. Macià *et al.* [8] studied the accuracy of viscous diffusion processes modeling in flows without free surfaces and with negligible effect of solid body boundary conditions. They found that the prevalent SPH viscous terms behave similarly in such flows. Nonetheless, they also found that when transport and diffusion are equally important, the accuracy of the SPH dissipation mechanisms was substantially reduced.

Basa *et al.* [9] tried to describe the instability occurring in SPH simulations of viscous flows (Poiseuille and lid driven cavity) relating its inception to the Reynolds number. Also, spurious dissipation has been a concern in SPH [10], [11] but to our knowledge, no analysis of spurious dissipation in viscous free surface flows can be found in the literature.

This paper is organized as follows: first, the governing equations of the physical problem we are interested in are presented. Second, the dissipation mechanisms of the physical model are introduced by identifying different contributions to the mechanical energy dissipation. Third, the practical problem considered, which is the attenuation of the kinetic energy in a standing wave, is discussed. The SPH model and the matrix of tests, considering a wide range of Reynolds numbers, are then presented. Finally, results are discussed at different levels and a conclusions summary together with future work targets are provided.

II. Governing equations

A. Field equations

SPH modeling of the dissipation mechanisms in gravity waves due to the existence of a free surface as well as the internal dissipation are the main focus of this paper. The fluid domain Ω is limited by a free surface, $\partial\Omega_F$, a bottom solid boundary, $\partial\Omega_B$, and by lateral boundaries, $\partial\Omega_P$, in which periodic boundary conditions will be imposed (figure 1).

The flow is modeled with the compressible (to be discretized using weakly compressible SPH) Navier-Stokes equations for

Hyperbolic Divergence Cleaning for SPH

Terrence S. Tricco
Monash Centre for Astrophysics
Monash University
Melbourne, Australia
terrence.tricco@monash.edu

Daniel J. Price
Monash Centre for Astrophysics
Monash University
Melbourne, Australia
daniel.price@monash.edu

***Abstract*—We present SPH formulations of Dedner et al's hyperbolic/parabolic divergence cleaning scheme for magnetic and velocity fields. Our implementation preserves the conservation properties of SPH which is important for stability. This is achieved by deriving an energy term for the ψ field, and imposing energy conservation on the cleaning subsystem of equations. This necessitates use of conjugate operators for $\nabla \cdot \mathbf{B}$ and $\nabla\psi$ in the numerical equations. For both the magnetic and velocity fields, the average divergence error in the system is reduced by an order of magnitude with our cleaning algorithm. Divergence errors in SPMHD are maintained to $< 1\%$, even for realistic 3D applications with a corresponding gain in numerical stability. Density errors for an oscillating elliptic water drop using weakly compressible SPH are reduced by a factor of two.**

I. Introduction

Magnetic fields have the property of being divergence free, that is $\nabla \cdot \mathbf{B} = 0$. Incompressible fluids have a similar divergence free property for the velocity field. Maintaining these divergence constraints is one of the central difficulties in performing accurate simulations of magnetohydrodynamics (MHD) and incompressible fluid behaviour. For MHD in particular, the presence of magnetic monopoles introduces a spurious force which, when large, is disruptive to the dynamics of the system.

Similar approaches can be utilised to satisfy the divergence constraints in both cases. For example, projection methods construct a divergence free vector field via the solution of a Poisson equation and have been applied successfully to both systems. Specialised approaches have also been developed for each case. One example is the constrained transport method [1] for MHD, which by conserving magnetic flux through a closed surface, can keep the divergence constraint to within machine precision. For SPH simulations of incompressible fluids, a stiff equation of state can be used to limit density variations to $\sim 1\%$ [2], creating a weakly compressible fluid approximating incompressibility.

The hyperbolic divergence cleaning method of Dedner et al [3] was introduced for maintaining the $\nabla \cdot \mathbf{B} = 0$ constraint in MHD. It involves the addition of a new scalar field, ψ, which is coupled to the magnetic field by

$$\left(\frac{\mathrm{d}\mathbf{B}}{\mathrm{d}t}\right)_{\psi} = -\nabla\psi. \tag{1}$$

This ψ field evolves according to

$$\frac{\mathrm{d}\psi}{\mathrm{d}t} = -c_h^2 \nabla \cdot \mathbf{B} - \frac{\psi}{\tau}, \tag{2}$$

and combined these produce a damped wave equation

$$\frac{\partial^2(\nabla \cdot \mathbf{B})}{\partial t^2} - c_h^2 \nabla^2(\nabla \cdot \mathbf{B}) + \frac{1}{\tau}\frac{\partial(\nabla \cdot \mathbf{B})}{\partial t} = 0. \tag{3}$$

Thus divergence is spread away from sources by a series of damped waves. The wave speed, c_h, is typically chosen to be the fastest wave obeying the Courant stability condition. The damping timescale, τ, acts as a diffusion on the divergence. By using waves to spread the divergence over a larger volume, the amplitude of any single large source is diminished and the diffusion is more effective. While originally proposed for use on the magnetic field for MHD simulations, this approach would be valid for any vector field. The damping timescale is set to $\tau^{-1} \equiv \sigma c_h / h$, where h is the smoothing length and σ is a dimensionless quantity specifying the damping strength.

Hyperbolic divergence cleaning has found popular use in both Eulerian [4], [5] and Lagrangian based codes [6], [7], chiefly for its simplicity, easy implementation, and low computational cost. However, for the SPH implementation of MHD (SPMHD), this method has not been widely adopted. Initial implementation attempts by Price [8] found divergence reductions were not substantial (a factor ~ 2), and the method risked actually increasing divergence in certain test cases.

The work presented here describes a new formulation of hyperbolic divergence cleaning for SPH that removes previous difficulties [9]. Implementations for both the magnetic and velocity fields are presented. Our formulation imposes the constraint of energy conservation on the subsystem of cleaning equations, guaranteeing that energy transferred to the ψ field must either be conserved or dissipated. This prevents increases in divergence.

The paper is laid out as follows: Sec. II discusses hyperbolic divergence cleaning for the magnetic field of SPMHD. A brief description of SPMHD is presented (Sec. II-A), along with the Euler Potentials (Sec. II-A1) and artificial resistivity (Sec. II-A2) since they will be used as a basis of comparison for the new divergence cleaning method. In Sec. II-B, the energy contained in the ψ field is derived and modifications are made to the cleaning equations to conserve energy, then the energy conserving SPMHD implementation is constructed (Sec. II-C). Hyperbolic divergence cleaning for the velocity field is discussed in Sec. III. Starting from an outline of weakly compressible SPH (Sec. III-A), a new energy term is created for the ψ field for contributions from the velocity

Modelling magnetic fields and turbulence with SPH

Daniel J. Price
Monash Centre for Astrophysics (MoCA)
and School of Mathematical Sciences
Monash University, Vic. 3800, Australia
daniel.price@monash.edu

***Abstract*—In recent years several key advances have been made in modelling both magnetic fields and turbulence in smoothed particle hydrodynamics. Solving the equations of magnetohydrodynamics (MHD) has proved an ongoing challenge over the last 35 years, but we have recently made a key breakthrough by developing a robust and safe method for enforcing the divergence-free condition on the magnetic field, enabling smoothed particle magnetohydrodynamics simulations with control of divergence errors and no restrictions on the field geometry. Modelling turbulence in SPH has benefited from faster algorithms allowing high resolution calculations capable of resolving the inertial range, particularly in supersonic flow, though SPH is most efficient when studying statistics of the density field, such as the density PDF. In subsonic flow use of viscosity switches is key to reaching high Reynolds numbers, which has been the source of recent controversy.**

I. Introduction

Magnetic fields and turbulence are important physical processes not only in many areas of astrophysics but also in Earth-bound applications of smoothed particle hydrodynamics (SPH). Both of these processes are thought to play a key role during the formation of stars from the gravitational collapse of interstellar clouds [17], a problem that SPH was originally designed for [10], [16] and is very well suited to modelling because of the ability to adaptively resolve the many orders of magnitude change in length and timescales involved. Turbulence itself is a ubiquitous phenomenon that defies analytic solution, so from the outset requires a numerical approach in order to model any system it occurs in. Recently we have made great strides in modelling both magnetic fields and turbulence using SPH, both of which I will attempt to outline in this paper.

II. Magnetic fields

A. Magnetohydrodynamics

Magnetic fields are usually modelled in the magnetohydrodynamics (MHD) approximation, where the equations of fluid dynamics adopt the form

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

$$\frac{\mathrm{d}\mathbf{v}}{\mathrm{d}t} = -\frac{1}{\rho}\nabla\cdot\left[\left(P+\frac{1}{2}\frac{B^2}{\mu_0}\right)I-\frac{\mathbf{BB}}{\mu_0}\right], \quad (2)$$

$$\frac{\mathrm{d}u}{\mathrm{d}t} = -\frac{P}{\rho}\nabla\cdot\mathbf{v}, \quad (3)$$

$$\frac{\mathrm{d}\mathbf{B}}{\mathrm{d}t} = (\mathbf{B}\cdot\nabla)\mathbf{v}-\mathbf{B}(\nabla\cdot\mathbf{v}), \quad (4)$$

where ρ is the density, $\mathbf{v}$ is the velocity, P is the gas pressure, u is the specific thermal energy, $\mathbf{B}$ is the magnetic field, infinite electrical conductivity has been assumed, $\mathrm{d}/\mathrm{d}t \equiv \partial/\partial t + (\mathbf{v}\cdot\nabla)$ refers to the comoving (Lagrangian) derivative, and the equation set is closed by adopting an appropriate equation of state (e.g. $P=(\gamma-1)\rho u$).

B. Smoothed Particle Magnetohydrodynamics

Although an early attempt was made by Gingold & Monaghan [10] to model magnetic stars, Phillips and Monaghan [25] represented the first systematic attempt to formulate the MHD equations in SPH, later coined 'Smoothed Particle Magnetohydrodynamics' (SPMHD) by Joe Morris [22]. In their most basic form the equations are very similar to the usual SPH equations and, like the SPMHD equations, can be derived in a self-consistent manner using a variational principle [37]. Taking full account of a spatially variable smoothing length h, the equations on a given particle a are given by

$$\rho_a = \sum_b m_b W_{ab}(h_a), \quad (5)$$

$$\frac{\mathrm{d}v_a^i}{\mathrm{d}t} = \sum_b m_b\left[\frac{S_a^{ij}}{\Omega_a\rho_a^2}\frac{\partial W_{ab}(h_a)}{\partial x_a^j}+\frac{S_b^{ij}}{\Omega_b\rho_b^2}\frac{\partial W_{ab}(h_b)}{\partial x_a^j}\right], \quad (6)$$

$$\frac{\mathrm{d}u_a}{\mathrm{d}t} = \frac{P_a}{\Omega_a\rho_a^2}\sum_b m_b(\mathbf{v}_a-\mathbf{v}_b)\cdot\nabla W_{ab}(h_a), \quad (7)$$

where the summations are over neighbouring particles, $b = 1..N_{\mathrm{neigh}}$, within the kernel radius, the MHD stress tensor is defined according to

$$S^{ij} \equiv -\left(P+\frac{1}{2}\frac{B^2}{\mu_0}\right)\delta^{ij}+\frac{B^iB^j}{\mu_0}, \quad (8)$$

and Ω is a dimensionless correction term resulting from the smoothing length gradients (see [19], [39]).

C. Removing the tensile instability in SPMHD

Phillips and Monaghan [25] discovered that the momentum-conserving formulation of the equations of motion (Eq. 6) is unstable when the magnetic pressure exceeds the gas pressure, $\frac{1}{2}B^2/\mu_0 > P$. The reason for this is both numerical and physical. The numerical explanation is that in this regime the overall stress tensor is negative, resulting in a negative total pressure, which when combined with the negative-definite sign of the kernel gradient in Eq. 6, results in an attractive

An algorithm for dusty gas with SPH

Guillaume Laibe, Daniel J. Price
Monash Centre for Astrophysics (MoCA) and School of Mathematical Sciences,
Monash University,
Clayton, Vic 3800, Australia
guillaume.laibe@monash.edu

Ben A. Ayliffe
School of Physics,
University of Exeter,
Stocker Road, Exeter EX4 4QL

***Abstract*—We present a new algorithm for simulating two-fluid gas and dust mixtures in SPH, systematically addressing a number of key issues. This includes the generalised SPH density estimate in multi-fluid systems, the consistent treatment of variable smoothing length terms, finite particle size, time step stability, implicit integration, the treatment of non-linear drag regimes, thermal coupling terms and the choice of kernel and smoothing length used in the drag operator.**

In this paper, we detail the two key points which allow the two phase gas and dust SPH algorithm to correctly solve the problem of the propagation of a sound wave in the mixture. First, using double- hump shaped kernels improves the accuracy of the drag interpolation by a factor of several hundred compared to the use of standard SPH bell-shaped kernels, at no additional computational expense. Second, the spatial resolution criterion $h \lesssim c_s t_s$ is a necessary condition that becomes critical at high drag (i.e. small stopping time t_s) in order to correctly simulate the propagation.

I. Introduction

SPH is commonly used to simulate astrophysical objects since free boundaries and large density gradients are easily handled. However, astrophysical mediums consist not only of gas, but also of dust particles, whose sizes vary from sub-micron sized grains in the interstellar medium to kilometre sized planetesimals involved in planet formation. Moreover, the ratio of dust to gas as well as the density and temperature of the gaseous environment in which dust is embedded can also vary strongly. Extending the SPH formalism to gas and dust mixtures is therefore crucial to correctly simulate the evolution of such astrophysical objects.

A first generation of SPH gas and dust algorithms have been developed by [1], [2] and used for astrophysical simulations, mostly in the context of planet formation [3]–[8]. However, after several tests, we found that the existing algorithm was not sufficiently accurate (especially for treating compressible flows) and developed a new suitable SPH formalism for gas and dust mixtures [9], [10].

II. SPH equations for dust and gas mixtures

The SPH version of the continuity equations are given by the density summations for both the gas and the dust phase, computed according to:

$$\hat{\rho}_a = \sum_b m_b W_{ab}(h_a); \qquad h_a = \eta \left(\frac{m_a}{\hat{\rho}_a}\right)^{1/\nu}, \tag{1}$$

$$\hat{\rho}_i = \sum_j m_j W_{ij}(h_i); \qquad h_i = \eta \left(\frac{m_j}{\hat{\rho}_i}\right)^{1/\nu}, \tag{2}$$

the indices a, b, c refer to quantities computed on gas particles and i, j, k refer to quantities computed on dust particles. The volume filling fraction θ, is defined on a *gas* particle, a, according to

$$\theta_a = 1 - \frac{\hat{\rho}_{\mathrm{d},a}}{\rho_{\mathrm{d}}}, \tag{3}$$

where $\hat{\rho}_{\mathrm{d},a}$ is the density of *dust* at the *gas* particle location, calculated using

$$\hat{\rho}_{\mathrm{d},a} = \sum_{j=1}^{N_{neigh,dust}} m_j W_{aj}(h_a), \tag{4}$$

where h_a is the smoothing length of the *gas* particle computed using gas neighbours (in general, the drag term has to be smoothed using the maximum smoothing length of the two fluids, rather than using an average to avoid unphysical resolution-dependent clumping of one fluid below the scale of the other). The local density of dust at the gas location can thus be zero (giving $\theta = 1$) if no dust particles are found within the kernel radius computed with the gas smoothing length. Since $\hat{\rho}$ and h are mutually dependent, they are simultaneously calculated for each type of particle. The kernel employed to perform the SPH density calculation is the standard bell-shaped kernel W:

$$W(r,h) = \frac{\sigma}{h^\nu} f(q), \tag{5}$$

where h denotes the smoothing lengths of each phases, ν the number of spatial dimensions and $q \equiv |\mathbf{r} - \mathbf{r}'|/h$ is the dimensionless variable used to calculate the densities and the buoyancy terms. The function f are usually the M_4 cubic or the M_6 quintic spline kernel.

The SPH equations of motion for the gas and the dust

2D and 3D sloshing simulation by SPH

Marzia Leonardi, Sauro Manenti, Stefano Sibilla
Dipartimento di Ingegneria Civile e Architettura
Università di Pavia
Via Ferrata, 1 – 27100 Pavia, Italy
sauro.manenti@unipv.it; stefano.sibilla@unipv.it

***Abstract*— The paper presents SPH simulation results concerning water-induced periodic loads on the rigid wall of a tank subjected to a harmonic external forcing. Sensitivity analyses on the model were performed. Reference was made to the laboratory tests proposed as benchmark by Botia Vera et al.: experimental and numerical time-histories of hydrodynamic pressure at measurement points were compared, as well as flow visualizations at different times. A simple case of three-dimensional sloshing in a rectangular tank was finally investigated in order to assess the ability of the model to reproduce 3D wave patterns.**

I. INTRODUTION

Sloshing is the oscillation of a fluid inside a partially filled container: this phenomenon represents an important problem in many engineering applications related to fluid transportation, such as trucks transporting liquids, propellant sloshing in aircraft/spacecraft tanks, sloshing in LNG tankers etc. [1]. It has, however, a practical importance also in another class of problems, such as seismic engineering applications. The behavior of a tank structure subjected to seismic loadings can be strongly affected by the sloshing motion which can take place inside it. Consequently the development of an efficient numerical model, able to provide a significant estimate of the forces applied on the tank walls, could be an helpful instrument for civil engineering purposes. This study is part of a wider research program aiming at the set up of an effective tool applicable to the design of anti-seismic water tanks.

Due to their widespread presence in many engineering fields, sloshing phenomena have been extensively investigated from both experimental and numerical points of view. Focusing on numerical studies, past researches were mainly carried out using Finite Difference and Finite Volume Methods or by shallow water equations solvers (see, e.g. [2-3]).

Only in the last decade a different approach to sloshing motion was proposed and encouraged: use of meshless methods and, in particular, of the Smoothed Particle Hydrodynamics method. The idea was that this kind of methods, due to their intrinsic characteristics, could be more suitable than the traditional grid-based ones to deal with sloshing flow features, especially when the behaviour of the waves departs from linearity or when violent impacts and breaking against the tank walls occur. Examples of SPH application on sloshing flows are quite numerous: see, for instance, [4-7].

The tank geometry considered in most of the cited works was mainly the rectangular one. In [5] were, however, proposed also other tank geometries (i.e. rectangular with obstacles, other shapes). Different filling levels and roll amplitudes were tested and the paper proposed a comparison between experimental data and numerical results obtained by SPH simulations.

The present paper shows the results obtained in the application of a numerical SPH model to benchmark test cases recently proposed by [8]. The available data are part of a 2D experimental setup on the study of wave impacts, tuned damping and fluid structure interaction problems in a rectangular tank. The data used in this study are referred to the first of the proposed problems (i.e. wave impact): pressure time histories, provided for both lateral and roof impacts, were compared with numerical results.

A WCSPH technique was chosen because of its advantages in the study of hydrodynamics problems, such as a lower computational effort and the automatic detection of free surfaces. The calibration of the model requires, because of this choice, a significant work. Sensitivity analysis was performed in order to assess the influence of the model parameters (e.g. fluid physical properties, viscosity and turbulence models, etc.) on numerical results. This influence was also highlighted by comparing numerical and experimental images of the free surface.

The study here proposed was mainly focused on 2D simulations but some 3D simulations were finally carried out in order to assess the model capability to reproduce three dimensional dynamics of free surfaces. Due to the lack of 3D experimental data, a specific comparison with numerical pressure results was not possible.

II. SPH METHOD

The flow fields during sloshing are obtained by solving the Navier-Stokes equations for a weakly compressible liquid with a conventional SPH discretization:

$$\frac{D\rho_i}{Dt} = \sum_j m_j (\vec{v}_i - \vec{v}_j) \cdot \vec{\nabla} W_{ij} \tag{1}$$

$$\frac{D\vec{v}_i}{Dt} = -\sum_j m_j \left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2} \right) \vec{\nabla} W_{ij} + \vec{T}_i + \vec{f}_{m_i} \tag{2}$$

SPH modeling of non-rectangular channel flows with open boundaries

Kao-Hua Chang, Tsang-Jung Chang
Dept. of Bioenvironmental Systems Engineering
National Taiwan University
Taipei, 106, Taiwan
f94622026@ntu.edu.tw, tjchang@ntu.edu.tw

***Abstract*—In this study, Smoothed Particle Hydrodynamics (SPH) is used to solve the Saint-Venant equations (SVEs) for modeling non-rectangular channel flows with open boundaries. So far, SPH has been developed to simulate dam-break flows with free boundaries and rectangular channel flows with open boundaries. However, for solving practical hydraulic problems, it is necessary to extend SPH to non-rectangular channel flows with open boundaries. Hence, the in/out-flow algorithm and the characteristic boundary method are improved herein to study the aforementioned problems. The study cases are aimed at transitions between subcritical and supercritical flows with non-uniform bed slopes in trapezoidal channels. In comparison with the exact results, it can be found that the present SPH approach produces satisfactory results for non-rectangular channel flows with open boundaries.**

I. INTRODUTION

SPH as a Lagrangian meshless method, was originally invented to simulate astrophysical problems by Lucy (1977) [1] and Gingold and Monaghan (1977) [2]. Later, SPH was extended successfully to model free surface flows by Monaghan (1994) [3]. After that, SPH was applied to a variety of applications by solving the Navier-Stokes equations (NSEs), such as free-surface flow problems, impact-fracture problems and multiphase flow problems etc. In recent years, SPH has been attempted to simulate dam-break flows by means of solving the SVEs [4-7]. These researches obtain promising results. Then, Vacondio et al. [8] proposed the characteristic boundary method to solve rectangular channel flows with open boundaries. This work has made SPH capable of handling more practical hydraulic problems.

However, for natural channels, the shape of cross-section is not only limited to be rectangular. To deal with non-rectangular channel flows, instead of water depth (d_w) and velocity (u), two variables, *i.e*. wetted cross-section area (A) and discharge (Q) should be solved. Three study cases, containing two rectangular and one trapezoidal channel flows with open boundaries, are tested. The comparisons of simulated and exact results are presented.

II. METHODOLOGY

In this study, the SVEs are solved to discuss non-rectangular channel flows. The SVEs written in a Lagrangian form are:

$$\frac{DA}{Dt} = -A\frac{\partial\left(Q/A\right)}{\partial x} \tag{1}$$

$$\frac{DQ}{Dt} = -Q\frac{\partial\left(Q/A\right)}{\partial x} - gA\frac{\partial d_w}{\partial x} + gA\left(S_0 - S_f\right) \tag{2}$$

where A is wetted cross-section area, Q is discharge, d_w is water depth, S_0 is bed slope, S_f is friction slope (= $n^2Q^2/A^2R^{4/3}$), n is Manning roughness coefficient, R is the perimeter and g is gravitational acceleration.

A. *Evaluation of wetted cross-section area*

To obtain better accuracy of the solution of the SVEs, a variable smoothing length (h) scheme is applied. Thus, the smoothing length of particle i (h_i), is connected to the wetted cross-section area [4,6] with

$$h_i = h_{0,i}\left(\frac{A_{0,i}}{A_i}\right)^{1/D_m} \tag{3}$$

where $h_{0,i}$ and $A_{0,i}$ are the initial wetted cross-section area and the smoothing length for particle i, respectively and D_m is the number of space dimensions (D_m = 1 herein).

In general, the wetted cross-section area is computed through the purely weighted summation. However, a Newton-Raphson iteration is proposed [4] because of the use of the variable smoothing length scheme. The iterative procedure is as follows.

Study of differential operators in the context of the semi-analytical wall boundary conditions

Arno Mayrhofer, Benedict D. Rogers
School of Mechanical, Aerospace and Civil Engineering
University of Manchester
Manchester, United Kingdom
arno.m@gmx.at

Damien Violeau, Martin Ferrand
Laboratoire National d'Hydraulique et d'Environnement
Électricité de France R&D
Chatou, France

***Abstract*—In this paper the semi-analytical wall boundary conditions by Ferrand et al. [1] are investigated theoretically by looking at the property of skew-adjointness in a continuous setting. As stabilizing procedure a volume diffusion term [2] is used and a new interpretation for it is given in the Reynolds averaged context. Additonally, a correction term for external forces is presented. The final theoretical contribution concerns arbitrary order Robin boundary conditions. The theoretical constructs are then investigated in various confined and free-surface flows. The issue of convergence is illustrated in the case of Poiseuille flow, the external force correction terms in the volume diffusion term and the boundary conditions are demonstrated via still water simulations. Finally, a standing wave and a dam-break over a wedge is simulated and quantitative comparisons are given. The paper is concluded by highlighting the difficulties associated with the extension to three dimensions as well as giving an insight into the current developments.**

I. Introduction

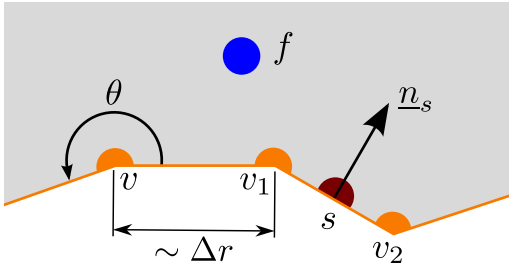


Fig. 1. The different types of elements.

Semi-analytical wall boundary conditions for arbitrary boundaries in SPH were introduced by Ferrand *et al.* in [1], [3], [4]. We first recall the main principles of this method. The key idea was to use an analytical kernel correction factor γ given by

$$\gamma_a = \int_\Omega w_{ab} \mathrm{d}\underline{r}_b. \tag{1}$$

The boundary conditions are implemented by using the sets of elements described in Table I. The fluid is discretized using particles. The boundary is split into line segments of length approximately Δr. These segments are called boundary elements and are located between two vertex particles as shown in Fig. 1.

Set	Description
$\mathcal{P}$	All particles
$\mathcal{V}$	Particles on boundary (vertex particles)
$\mathcal{F}$	Particles inside the fluid domain
$\mathcal{S}$	Boundary segments

TABLE I
Overview of elements.

In the following differential operators will be written in bold (e.g. **Div**). Vectors and matrices will be written as $\underline{B}$ and $\underline{\underline{M}}$ respectively. The general SPH approximation for a scalar f at position a is given by

$$[f]_a = \frac{1}{\gamma_a} \sum_{b \in \mathcal{P}} V_b f_b w_{ab}, \tag{2}$$

where V is the volume, $\mathcal{P}$ the set of all particles and w the SPH kernel. In order to obtain the analytical value for γ, a governing equation is used, given by

$$\frac{\mathrm{d}\gamma_a}{\mathrm{d}t} = \underline{v}_a^r \cdot \underline{\nabla} \gamma_a, \tag{3}$$

where $\underline{v}^r$ is the velocity relative to the wall. Ferrand *et al.* [3] derived an analytical expression for $\underline{\nabla}\gamma_a$ in 2-D for the Wendland kernel. Without loss of generality this kernel will be used throughout this paper. The other key idea is to rigorously derive the differential operators without neglecting boundary terms coming from the integration by parts. For the SPH approximation as given by Eq. (2) Ferrand *et al.* proposed the following SPH approximation for the divergence of a vector field $\underline{B}$

$$\mathbf{Div}_a^{\gamma,F}(\underline{B}) := -\frac{1}{\gamma_a \rho_a} \sum_{b \in \mathcal{P}} m_b \underline{B}_{ab} \cdot \underline{\nabla}_a w_{ab} + \frac{1}{\gamma_a} \sum_{s \in \mathcal{S}} \underline{B}_{as} \cdot \underline{\nabla} \gamma_{as}, \tag{4}$$

where the superscript F stands for "Ferrand" and $\underline{B}_{ab} = \underline{B}_a - \underline{B}_b$. Following the same paper, the gradient of a scalar field f

Apply C1 Consistency to SPH With Free Surface

Xu Haihua*, Dao My Ha+
*Tropical Marine Science Institute
National University of Singapore
Singapore
G0600365@nus.edu.sg
+The Singapore-MIT Alliance for Research and Technology
Singapore

Chan Eng Soon**, Pavel Tkalich++
**Department of Civil and Environmental Engineering
National University of Singapore
Singapore
++Tropical Marine Science Institute
National University of Singapore
Singapore

***Abstract*—When the particles restoration method is applied on the free surface particles, the SPH simulation may blow-up. The blow-up may due to the error accumulation when solving the linear equation with high condition number. In current study, the dynamic free surface boundary condition is applied on the free surface to remove the error accumulation and prevent the simulation blow-up. With this method, the consistency of all the particles in the simulation domain is restored to C1. An improved surface detection method is also introduced in the study. The benchmark of hydrostatic and sloshing problems show the new method can improve the energy conservation, free surface tracking and pressure estimation.**

I. INTRODUTION

The Smooth Particle Hydrodynamics (SPH) is a Lagrangian particle method and has been widely used in many areas such as astronomy, computational fluid dynamics, solid mechanics and so on. However, the traditional SPH method suffers several drawbacks. The lack of particle consistency is one of the most severe problems [1-3].

Many approaches have been published to restore the particle consistency to C1 in previous researches [4, 5]. With the particle consistency restoration, the energy conservation, the accuracy of free surface tracking and pressure estimation can be improved. However, these methods only restore the consistency of particles inside the fluid domain, while the particles near or at the free surface are not. This may due to the consistency restoration methods involve a set of linear equations and the condition number of the linear equations for particles near to at the free surface may very high. Solving these equations may introduce numerical errors and cause the SPH simulation blow-up, if the numerical error can't be removed and accumulate in the system. Although only restore the inner particles can avoid the simulation blow-up and improve the results, it still has several limitations. First, the consistency of particles inside the fluid domain has been restored to C1, but the consistency of particles near or at the free surface is C0 or less than C0. In overall the consistency of the whole system is less than C1. Second, for these low consistency particles, the accuracy of the velocity divergence and pressure gradient maybe low, which may influence the density evaluation, pressure estimation, free surface tracking and total energy conservation. Hence, the restoration of consistency for particles near or at the free surfaces is important.

In this paper, the one searching step correction method is employed to restore the particle consistency for particles inside the fluid domain, near the free surface and at the free surface. The reason of the simulation blow-up will be discussed. A simple but efficient method will be proposed to prevent the simulation blow-up. An improved surface detection method will be introduced. The new method is benchmarked by the hydrostatic simulation and the water oscillation in a 2D tank.

II. THE GOVERNING EQUATION

In the present work, the δ-SPH scheme proposed by Antuono, et al. [6] is employed. The δ-SPH scheme introduces the artificial diffusion term in continuity equation to remove the pressure oscillation. The equation can be expressed as:

$$\begin{cases} \dfrac{d\rho_a}{dt} = -\rho_a \nabla\cdot\mathbf{u}_a + \xi h c_{ab}\rho_{ab}\sum_{b=1}^{N}\psi_{ab}\cdot\nabla_a W_{ab}V_b \\ \dfrac{d\mathbf{u}_a}{dt} = -\dfrac{1}{\rho_a}\nabla p_a + \mathbf{g} + \alpha c_{ab}\rho_{ab}\sum_{b=1}^{N}\pi_{ab}\cdot\nabla_a W_{ab}V_b \\ p(\rho) = \dfrac{\rho_o c_0^{\,2}}{\gamma}\left[\left(\dfrac{\rho}{\rho_o}\right)^{\gamma} - 1\right] + p_o \end{cases} \quad (1)$$

Where $\pi_{ab} = (\mathbf{u}_a - \mathbf{u}_b)\dfrac{\mathbf{r}_{ab}}{(r_{ab}^{\,2} + \eta^2)}$, $\psi_{ab} = 2(\rho_a - \rho_b)\dfrac{\mathbf{r}_{ab}}{(r_{ab}^{\,2} + \eta^2)}$, $\mathbf{r}_{ab} = \mathbf{r}_a - \mathbf{r}_b$, ρ_a , $\mathbf{u}_a = (u_a, v_a)$ and P_a are the density, velocity and pressure of particle a, respectively. h is the smoothing length, c_0 is the initial sound speed, V_b is the volume of particle b, $\rho_{ab} = (\rho_a + \rho_b)/2$ is the average density, $c_{ab} = (c_a + c_b)/2$ is the average sound speed, $\nabla_a W_{ab}$ is the gradient of kernel function respect to particle a, ξ and α control the magnitude of the diffusion terms, $\xi = 0.0001$ and $\alpha = 0.03$ are employed in current study [1].

SPH modelling of viscous flows around cylinders from Re=10 to Re=1000

S. Marrone, M. Antuono
CNR-INSEAN
Rome, Italy

A. Colagrossi, G. Colicchio
CNR-INSEAN, Rome, Italy
CESOS, NTNU, Trondheim, Norway

G. Graziani
University of Rome "Sapienza",
Rome, Italy

***Abstract*—The SPH model in his weakly compressible formulation is applied to simulate viscous flows at low and intermediate Reynolds numbers ($10 < \mathbf{Re} < 1000$) around a cylinder. It is well know in the literature that this kind of problem represents a challenge for the proposed numerical model. To overcome these drawbacks, here a proper ghost-fluid technique is proposed to deal with blunt bodies in viscous fluids. The no-slip condition on the body surface is implemented combining the technique proposed by Takeda et al. (1994) with the one proposed by De Leffe et al. (2011). The global loads on bodies are efficiently evaluated in the ghost fluid framework. Making use of a weakly compressible approach, additional issues have to be addressed for the sound waves. An in-depth validation of the model is performed comparing the numerical outcome with experiments data from the literature and other numerical references. In particular the solver is validated on the prediction of: drag and lift coefficients, wake length and shape and angle of separation (low Reynolds number), shedding frequency and minimum and maximum angles of separation (intermediate Reynolds number). The influence of the domain size is discussed in order to avoid wall side effects and at the same time to limit the computational costs. Convergence of the numerical solutions have been checked for both global and local quantities choosing appropriate Reynolds-cell number for the different test cases.**

I. Introduction

The vortex shedding from blunt bodies is of practical importance in many engineering applications. For bridge columns laying on sandy bottoms, the released vorticity interacts with the sediments eventually leading to the dangerous scouring of the basement [25]. In offshore engineering, the oil rigs can undergo severe oscillations induced by the alternate vortex shedding when storms approach the extraction site. Neglecting this phenomenon in the design stage can cause economic loss, either the rigs get damaged or have to be temporarily removed during storms [29]. Similar issues occur in several other fields. For this reason, several academic studies address the problem of the simple vorticity generated by a cylinder in a stream (see e.g. [28] for a review). In SPH, this problem presents several issues. One of them is the choice of a suitable no-slip boundary condition. A common way is to use a ghost fluid technique. Macia et al. [14] showed that, using the Monaghan & Gingold [19] viscous operator together with the Takeda et al. [26] mirroring procedure, the viscous stress on the body is correctly modelled.

Recently, some authors (e.g. [13]) showed that the weakly-compressible SPH is not suitable to simulate the above mentioned flows while the incompressible variants seem to be more effective. Nevertheless, Takeda et al. [26] in 1994 obtained good results for the flow around a circular cylinder at Re = 40, however limiting its analysis only to low Reynolds numbers. This success is due mainly to three reasons: the choice of an appropriate equation of state, direct density computation from particles masses and positions, an accurate enforcement of no-slip condition on solid boundaries. Conversely, if the density is evaluated through integration of the continuity equation, De Leffe et al. [7] showed that the ghost fluid properties need to be different for the momentum equation and for the continuity equation. The use of a unique ghost fluid, leads to model inconsistencies that cause strong numerical instabilities. Finally, in [15] a way to extend the classical particles ghost technique to curved and edge surface is presented an validated extensively.

The present work proposes a ghost fluid approach formulation which takes into account the above considerations. The model is used to study the flow around cylinders. Detailed analyses of the flow have been done for a wide range of Reynolds numbers and several parameters, as features of the loads acting on the body and also of the velocity field, are taken into account. Satisfactory results have been obtained by only introducing: accurate modelling of no-slip boundary conditions, appropriate conditions to get rid soon of the unwanted weakly compressibility effects, efficient evaluation of the global loads. These prescriptions are valid for each WC-SPH formulation even though, here, they have been adopted for the δ-SPH model proposed by [1]. The analysis has been developed into two parts. In the first one, the flow is confined and the cylinder moves with a prescribed function and the results are compared with a Finite Difference Navier-Stokes solver (FDNS) described in [5]. Later, more complexity is added to the problem introducing open boundaries. These have been treated with the inflow/outflow algorithm proposed by [12]. For this case the obtained results are compared with experiments and with other numerical data available in the literature.

II. The adopted model

In the present work we adopt the δ-SPH scheme proposed by [1]. In this framework, the fluid is assumed to be barotropic and weakly-compressible and the reference equations are the

Turbulent Coherent Structures under Breaking Water Waves

Rozita Jalali Farahani
Civil Engineering Department
Johns Hopkins University
Baltimore, USA
rozita@jhu.edu

Robert A. Dalrymple
Civil Engineering Department
Johns Hopkins University
Baltimore, USA
rad@jhu.edu

Alexis Hérault
Conservatoire National des Arts et Métier
Paris, France
alexis.herault@cnam.fr

Giuseppe Bilotta
Università degli studi di Catania
Catania, Italy
bilotta@dmi.unict.it

Abstract—**The nature of wave breaking at beaches and the subsequent three-dimensional turbulence under the waves are of great interest in coastal engineering. The breaking waves were modelled on a slope using the 3-D open-source GPUSPH code of Hérault, Bilotta and Dalrymple (2010). Then, different coherent structure detection methods were employed, including vorticity, Q criterion, and λ_2 criterion. These last two criteria are based on the velocity gradient tensor and its symmetric and anti-symmetric components. The evolution of the turbulence under a plunging wave is discussed. For plunging waves, the plunger touch-down, the jet penetration, and subsequent splash-up are all sources of vorticity.**

I. INTRODUTION

The characteristics of wave breaking in the surf zone has been of great interest for coastal engineers and fluid dynamicists. As ocean waves propagate towards shore they undergo shoaling, shortening in length and increasing in height. As the waves enter the diminishing water depths close to the beach, waves break, leading to splash-up, and white water fronts or spray depending on the nature of the waves. There are four different generic breaker types: spilling, plunging, collapsing and surging waves, depending on the initial steepness of the waves and the steepness of the beach slope [1].

- In spilling waves, when the wave crest becomes unstable, it spills forward and creates white water in front of the wave and the wave energy is dissipated in this turbulent white water. Spilling waves usually occur on gradual beach slopes.
- In plunging waves, the crest of the wave becomes very steep--almost vertical, then it overturns as a jet and falls into the water in front of the wave, followed by a splash-up, which is sometimes higher than the original wave. Depending on the position of the first plunging point, a sequence of plunging and overturning jets appear in front of the initial wave. This splash-up cycle produces vortex structures propagating towards the shore.
- In collapsing waves, the lower part of the wave gets steeper and curls over, following by bubbles and foam.
- For surging waves, which appear on very steep beaches, the lower part of the wave moves rapidly forward and the wave crest does not curl over. There is no turbulent breaking process.

Several laboratory and numerical studies demonstrated that the turbulent flow under the breaking wave could be characterized by large-scale flow structures, which are called coherent structures. Nadaoka, Hino and Koyano [2] performed a set of laboratory experiments to examine the breaking wave dynamics and the generated turbulence structure. They studied the three-dimensional eddies found under and behind the wave crest, called obliquely descending eddies. Ting [3] studied a solitary wave experimentally and captured coherent structures under the breaking wave, which were the sources of turbulent energy. Watanabe, Saeki and Hosking [4] modelled the large-scale vortex structures under spilling and plunging waves using large eddy simulation (LES) method. Three dimensional vortex structures were captured in both plunging and spilling waves. They studied the obliquely descending eddies, which have an important role in sediment transportation and beach erosion.

In this study, generation and evolution of Lagrangian three-dimensional coherent structures under the plunging water waves are numerically modelled using SPH (Monaghan, 1994), specifically the GPUSPH model developed by Hérault, Bilotta and Dalrymple (2010) [5], [6]. The Navier-Stokes equations are numerically solved using a turbulence closure model to consider small-scale eddies as well.

II. COHERENT STRUCTURES

In this section, we briefly outline some definitions related to the vortex and coherent structures in fluid mechanics. Study of spatially coherent, time-depending vortex structures has an important role on the understanding of turbulent flow physics. Several researchers have

A SPH model for incompressible turbulence

X. Y. Hu and N. A. Adams
Lehrstuhl für Aerodynamik und Strömungsmechanik, Technische Universität München
85748 Garching, Germany
Xiangyu.Hu@tum.de

***Abstract*—A coarse-grained particle model for incompressible Navier-Stokes (NS) equation is proposed based on spatial filtering by utilizing smoothed particle hydrodynamics (SPH) approximations. This model is similar to our previous developed SPH discretization of NS equation (*Hu X.Y. & N.A. Adams, J. Comput. Physics*, 227:264-278, 2007 and 228:2082-2091, 2009) and the Lagrangian averaged NS (LANS-α) turbulence model. Other than using smoothing approaches, this model obtains particle transport velocity by imposing constant σ which is associated with the particle density, and is called SPH-σ model. Numerical tests on two-dimensional decay and forced turbulences with high Reynolds number suggest that the model is able to reproduce both the inverse energy cascade and direct enstrophy cascade of the kinetic energy spectrum, the time scaling of enstrophy decay and the non-Guassian probability density function (PDF) of particle acceleration.**

I. Introduction

The smoothed particle hydrodynamics (SPH) method is a fully Lagrangian, grid free method. Since its introduction by Lucy [14] and Gingold and Monaghan [6], SPH has been applied to a wide range of macroscopic and mesoscopic flow problems [19] [10]. Though the SPH method was originally developed for astrophysical problems involving compressible fluids, it has been extended to problems involving incompressible fluids by using either a weakly compressible model of the fluid [17], or by algorithms designed to solve the full incompressible equations [3] [11] [12].

Many of the incompressible flow problems, such as flood and coastal flows, to which SPH has been applied are turbulent. Since the direct numerical simulation of these problems is not always feasible, turbulence modeling is required for the computational more efficient coarse-grained numerical simulation. One straightforward approach of SPH turbulence modeling is applied the turbulence models originally developed for Euelrian methods directly [7] [25] [27].

Monaghan [18] first noticed the similarity between the version of SPH called XSPH [16] and the Lagrangian averaged Navier-Stokes (LANS) turbulence model [8] [9] on the relation between the velocity determined from particle momentum (momentum velocity) and the transport velocity, and proposed a turbulence model specifically for the SPH method. In this model, the SPH particle moves with the transport velocity smoothed from momentum velocity by an iterative algorithm and a dissipation term is introduced to mimic the standard large eddy simulation (LES) model originally developed for Eulerian methods. A further modification of this model (SPH-ϵ) is to obtain transport velocity directly by the XSPH method with a parameter ϵ [20]. On the other hand, we have noticed the importance of SPH particle moving with the velocity different from its momentum velocity when simulating flows beyond small Reynolds number in our previous developed incompressible SPH method [11] [12]. In this method, other than using the XSPH method or smoothing approaches, the Eulerian incompressibility condition (zero velocity divergence) and the Lagrangian incompressibility condition (constant density) are used respectively to determine the momentum velocity and the transport velocity.

In this paper, we propose a coarse-grained particle system for turbulence simulation based on spatial filtering the Navier-Stokes (NS) equation by utilizing SPH approximations. Since the resulting particle equations are similar to those of the above mentioned incompressible SPH method, except for an additional effective stress term introduced by moving particle with transport velocity, the same numerical method is applied. The numerical tests show that, while achieving good accuracy for resolved flow, the present model can recover the spectral and statistical properties of the two-dimensional decay and forced turbulences with high Reynolds number.

II. Model

We consider the incompressible isothermal NS equation in Lagrangian form

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

$$\frac{d\rho}{dt} = 0 \quad \text{or} \quad \nabla\cdot\mathbf{v} = 0, \quad (2)$$

where ρ, p and $\mathbf{v}$ are fluid density, pressure and velocity, respectively, and $\nu = \eta/\rho$ is the kinematic viscosity. Note the two expressions (constant density and zero velocity-divergence) in Eq. (2) are formally equivalent.

In the next section, we propose a coarse-grained particle system based on filtering the NS equation with numerical techniques in the SPH method. This approach is different from the SPH-ϵ model, which is devised from the SPH discretized form of Eckart's Lagrangian [4].

A. Coarse-grained NS equation and SPH method

Assume that the incompressible flow field is coarse-grained into a particle system with spatial filtering, the variables on particles are obtained by

$$\psi_i = G_i * \psi = \int \psi W(\mathbf{r}-\mathbf{r}_i, h)d\mathbf{r}, \quad (3)$$

SPH simulations of 2D turbulence driven by stirrers

Alireza Valizadeh and Joe Monaghan
School of Mathematical Sciences
Monash University
Clayton VIC 3800, Australia
alireza.valizadeh@monash.edu and joe.monaghan@monash.edu

***Abstract*—In this paper we extend our previous study of decaying turbulence in two dimensions using SPH by simulating driven turbulence. Previous studies of driven turbulence have used stochastic forces to maintain the turbulence, presumably because it was thought that the simulation of mechanical stirrers was too complicated. However, mechanical stirrers can be simulated easily with SPH and we do it in this paper for cylindrical stirrers moving on specified trajectories. The turbulence is generated by the vortices that are produced behind the cylinders as they move through the fluid. To confirm that our algorithm generates these vortices accurately we applied it to the vortex induced vibration of a cylinder in channel flow finding very satisfactory results. To drive the turbulence one or more stirrers move on a trajectory that is a Lissajous figure with a 3:1 frequency ratio. We show that the spectrum and correlation functions agree with theory and with other simulations. The motion of the cylinders, the formation of the vortices, and their subsequent interaction with the motion of the stirrers provides very interesting examples of fluid dynamics with moving bodies.**

I. Introduction

Recent studies of decaying turbulence in two dimensions ([24], [17], [27]) have shown that SPH simulations converge and give results in good agreement with those of other authors ([5], [6]). However, most turbulence simulations consider driven turbulence rather than decaying turbulence since steady turbulence is often assumed in theoretical work. In the laboratory, driven turbulence is due to an input flow, or to some sort of moving mechanical device, but the simulation of such a mechanism has been considered too difficult to implement. Instead, the turbulence has been initiated and maintained by stochastic forces ([13]. In the present paper we exploit the flexibility of SPH and consider turbulence driven by cylindrical stirrers moving on Lissajous figures.

A key feature of a cylindrical stirrer is that it leaves a wake of vortices and these vortices subsequently intereact with themselves and with the moving cylinder. The resulting turbulence is driven at a length scale comparable to the length scale of the vortices which is the diameter of the cylinder. In order to be confident that our algorithm generates the vortices correctly we tested it by simulating a cylinder in a steady flow and a cylinder attached to a spring in a channel flow. In particular, our results for the oscillating cylinder are in good agreement with those of [21] and [7].

In this paper the spectrum and correlation functions for the fluid in a square container with no-slip walls are calculated both when there is one stirrer and when there are two. In both cases the spectrum and correlation functions have a similar form though, as expected the energy and enstrophy are greater when there are two stirrers than when there is one stirrer.

II. SPH model

We consider a weakly compressible fluid with pressure P a function of density ρ. Surface tension is neglected. The reader is assumed to be familiar with standard SPH as described in the reviews by [14], [15]. In the following, the labels a and j are used for SPH fluid and boundary particles respectively, and η is used when a summation is over both fluid and boundary particles.The SPH form of continuity equation is

$$\frac{d\rho_a}{dt} = \rho_a \sum_\eta \frac{m_\eta}{\rho_\eta} \left(\mathbf{v}_a - \mathbf{v}_\eta\right) \cdot \nabla_a W_{a\eta}, \tag{1}$$

where the mass, position, velocity, density and pressure of particle a are m_a, r_a, $\mathbf{v}_a$, ρ_a, and P_a, respectively. The summation is over all particles The function $W_{a\eta} = W(|\mathbf{r}_{a\eta}|, h)$ is the SPH kernel, $|\mathbf{r}_{a\eta}|$ is the distance between particle a and particle η, and $h = (h_a + h_\eta)/2$ is the average smoothing length. In the calculations to be described the kernel is the Wendland function which vanishes at $r = 2h$ according to a fourth order zero ([29]). We choose $h_\eta = 1.5\delta$, where δ is the initial particle spacing. The gradient taken with respect to the coordinates of particle a is denoted by ∇_a. The pressure of fluid particle a is given by

$$P_a = \frac{\rho_0 c_s^2}{7}\left(\left(\frac{\rho_a}{\rho_0}\right)^7 - 1\right), \tag{2}$$

where ρ_0 is the reference density of the fluid. The speed of sound c_s is χ times the maximum speed of fluid V_{max} which is calculated at each time step from the velocity field and χ is between 10 and 15. The result is that although the speed of sound is time varying the Mach number remains constant. The boundary particles have zero pressure.

The acceleration equation for the SPH particle a is

$$\frac{d\mathbf{v}_a}{dt} = -\sum_\eta m_\eta \left(\frac{P_a}{\rho_a^2} + \frac{P_\eta}{\rho_\eta^2} - \Pi_{a\eta}\right)\nabla_a W_{a\eta} + \sum_j m_j \mathbf{f}_{aj}. \tag{3}$$

The first summation in (3) is over all particles. The viscosity is determined by $\Pi_{a\eta}$ for which we use the form

$$\Pi_{a\eta} = -\alpha \frac{\bar{c}}{\bar{\rho}_{a\eta}} \frac{\mathbf{v}_{a\eta} \cdot \mathbf{r}_{a\eta}}{|\mathbf{r}_{a\eta}|}, \tag{4}$$

Advective and Diffusive Turbulent Mixing

J. J. Monaghan
School of Mathematical Sciences
Monash University, Clayton
Melbourne, Australia
joe.monaghan@monash.edu

J. B. Kajtar
School of Mathematical Sciences
Monash University, Clayton
Melbourne, Australia
jules.kajtar@monash.edu

***Abstract*—Recent studies of a turbulence model for SPH [1], [2] show that, for two dimensional turbulence in a square container with no slip walls, it gives results that are in good agreement with those from a high accuracy spectral method. A key feature of the model is that the fluid is advected with a velocity $\widehat{\mathbf{v}}$ obtained by smoothing the velocity $\mathbf{v}$. The equations of motion are obtained from a Lagrangian in which the kinetic energy per unit mass is $\frac{1}{2}\mathbf{v}\cdot\widehat{\mathbf{v}}$, and the movement of an SPH particle is given by $d\mathbf{r}/dt = \widehat{\mathbf{v}}$. The linear and angular momentum and a discrete form of the circulation are conserved. The simulation of turbulent flow in a square container shows that the velocity correlation function for low resolution is close to that calculated using a resolution twice as fine. The reason for the improvement is that the noise in the velocity field is greatly reduced by the smoothing. However, the smoothing of the velocity field affects the mixing because the velocity variations on a short length scale are removed, and there are clear differences in the flow. Some of the difference can be attributed to the fact that the turbulent flow is chaotic and the smoothing makes initial small changes to the advection and these changes are amplified. However, part of the difference is due to the effect of smoothing on the small length scale motion. In this paper we analyse the time variation of the mixing at different length scales and we show the results obtained by including a diffusion equation which describes the diffusion of material due to small velocity fluctuations. This equation takes the same form as that given by Monaghan [3] in connection with the freezing of binary solutions. A key feature of the formulation is that the difference between $\widehat{\mathbf{v}}$ and $\mathbf{v}$ determines the velocity fluctuation, and this determines the coefficient of diffusion.**

I. Introduction

When a laminar flow becomes turbulent mixing occurs. In particular the linear and angular momentum are mixed and this changes the stresses in the fluid. Any scalar quantity the fluid carries is also mixed. The mixing occurs on all scales but it varies with the length scale with the small scales being less easily mixed than large scales. If turbulence is simulated using high resolution, the turbulent flow is found to eventually involve thin sheets and ribbons that decay by viscosity. To simulate these structures correctly it is necessary to resolve down to the Reynolds length ℓ_R which is given in 2D by $\ell_R = L/\mathcal{R}^{1/2}$ where L is the macroscopic scale and the Reynolds number is $\mathcal{R} = VL/\nu$ where V is the large scale velocity and ν the kinematic viscosity coefficient. These thin structures are seen in the direct SPH simulations of 2D turbulence [1], [2], [4]. Unfortunately, in many simulations, the resolution required to describe the turbulence correctly cannot be afforded, even with large parallel clusters. In the case of finite difference calculations the way of escape is to use a turbulence model where the original fluid equations are smoothed in space. The method is called Large Eddy Simulation (LES) and the grid scale is larger than grid spacing of ℓ_R. The use of a coarse grid is partly compensated by including a sub-grid stress that must be guessed.

It has been shown that a recently proposed turbulence model for SPH has a number of desirable properties [1]. These include conservation of linear and angular momentum and circulation (approximately) together with the ability to recover the correlation functions of direct numerical simulations while using a much coarser resolution. The equations of motion are derived in a similar way to that used in a continuum Lagrangian turbulence model called the Lagrangian Averaged Navier-Stokes alpha model (LANS-α) due to Holm and his colleagues [5], [6], and for further references see Lunasin et al. [7]. LANS was originally developed from a lengthy consideration of turbulent fluctuations. However, the end result is simple. A smoothed, or regularised, velocity $\widehat{\mathbf{v}}$ is calculated by a linear operation on the un-smoothed velocity $\mathbf{v}$, and then the Eulerian equations of motion are determined from a Lagrangian with kinetic energy per unit mass $\frac{1}{2}\mathbf{v}\cdot\widehat{\mathbf{v}}$. The particles are moved according to $d\mathbf{r}/dt = \widehat{\mathbf{v}}$, and the combination of $\mathbf{v}$ and $\widehat{\mathbf{v}}$ guarantees that circulation is conserved in the absence of dissipation. The overall result is that, in driven turbulence in the absence of viscosity, the energy in small length scales is redistributed to larger scales. Combined with a viscous term, the equations give a very satisfactory description of turbulence (in periodic domains [6], gyres relevant to oceanography [8], and mixing [9]).

II. Governing Equations

The smoothed velocity $\widehat{\mathbf{v}}_a$ and the un-smoothed velocity $\mathbf{v}_a$ for an SPH liquid particle a are related by [1]:

$$\widehat{\mathbf{v}}_a = \mathbf{v}_a + \epsilon \sum_b \frac{m_b}{M}(\mathbf{v}_b - \mathbf{v}_b)K(|\mathbf{r}_a - \mathbf{r}_b|, \ell). \qquad (1)$$

Here m_b is the mass of particle b and M is a mass closely related to the typical mass of a particle and $0 \leq \epsilon < 1$. The function K is a smoothing function with typical length scale ℓ. It is similar to a Gaussian. The integral of K over the space of the simulation is equal to ℓ^d where d is the number of dimensions. The reader familiar with SPH will recognise this smoothing as the XSPH variant of SPH [10]. It can be shown that, if the velocities are expanded in a Fourier series, the

On the use of numerical diffusive terms in weakly-compressible SPH schemes

M. Antuono
CNR-INSEAN
Rome, Italy

A. Colagrossi
CESOS
NTNU, Trondheim, Norway

S. Marrone
CNR-INSEAN
Rome, Italy

***Abstract*—An in-depth discussion on the use of numerical diffusive terms in SPH models has been performed. These terms are generally added inside the continuity equation in order to reduce the spurious numerical noise that affects the density and pressure fields in weakly-compressible SPH schemes. Specific focus has been given to the theoretical analysis of the diffusive term structure and to the choice of the integration scheme and Courant-Friedrick-Lewy number. The most widespread formulations, that is, those by Ferrari et al. [1], Molteni & Colagrossi [2] and Antuono et al. [3], have been studied in details, highlighting the main benefits and drawbacks.**

I. Introduction

In the SPH literature, two principal approaches are adopted to model liquids: one is based on the smoothing of the Navier-Stokes equations and on the solution of a Poisson equation for the pressure field, the other relies on the assumption that the fluid is weakly-compressible and barotropic.

From a numerical point of view, the main differences between the weakly-compressible and incompressible approaches is that the former requires small time steps constrained by the speed of sound, while the later needs to solve an algebraic system with a sparse matrix, allowing for larger time steps but rather complex for an efficient parallelization. Further, weakly-compressible schemes are generally more suited for free-surface flows since the boundary condition along the free surface is implicitly satisfied (see, for example, Colagrossi et al. [4]) and this avoids an explicit detection of the free surface during the flow evolution. The latter issue can be critical in 3D simulations of violent flows since the Poisson equation may strongly depend on the free surface configuration and small errors in the free-surface detection can lead to different flow dynamics.

Unfortunately, the weakly-compressibility schemes have as a major drawback the generation of spurious numerical oscillation in the pressure and density fields. Over the years, different solutions have been proposed to remove/reduce the spurious numerical noise that affect the pressure field in SPH model. Among them, one is to use proper diffusive terms. For example, Ferrari et al. [1] used a Rusanov flux and built a numerical diffusive term to be added inside the continuity equation. This helped reduce the numerical noise inside the density field and, consequently, inside the pressure field through the state equation (we recall that the fluid is assumed to be barotropic). The use of a numerical diffusive term inside the continuity equation has been also proposed by Molteni & Colagrossi [2]. Their term gave good results but, unfortunately, was inconsistent with the hydrostatic solution. The authors avoided this issue by introducing a threshold density so that the diffusive term only worked when the pressure field exceeded the hydrostatic field. Unfortunately, this strategy led to a drastic reduction of the diffusive term action. To go round this issue, Antuono et al. [3] proposed a correction to the diffusive term of Molteni & Colagrossi [2]. This proved to be compatible with the hydrostatic solution and to properly smooth out the numerical spurious oscillations from the pressure and density fields.

The aim of the present work is to shed light on the use of numerical diffusive terms in SPH. Specifically, we focus on the diffusive term of Ferrari et al. [1], Molteni & Colagrossi [2] and Antuono et al. [3] and show their benefits and drawbacks.

II. SPH scheme with numerical diffusive terms

In this section we study the general structure of a SPH scheme with a numerical diffusive term inside the continuity equation. Specifically, we assume the fluid to be weakly-compressible and barotropic. Under these hypotheses, the density variations are small and it is possible to linearize the state equation in the neighborhood of the reference density value to get a linear dependence of the pressure on the density field. Finally, the artificial viscous term by Monaghan & Gingold [5] is added inside the momentum equation for stability reasons. In any case, we underline that the theoretical analysis on the role of the diffusive term is completely general and can be applied to SPH schemes with different features.

Under the hypotheses above, the governing equations for the SPH scheme at hand are:

$$\begin{cases} \dfrac{D\rho_i}{Dt} = -\rho_i \displaystyle\sum_j (\boldsymbol{u}_j - \boldsymbol{u}_i)\cdot\nabla_i W_{ij}\, V_j + \delta\, h\, c_0\, \mathcal{D}_i \\ \dfrac{D\boldsymbol{u}_i}{Dt} = -\dfrac{1}{\rho_i}\displaystyle\sum_j (p_j + p_i)\,\nabla_i W_{ij}\, V_j + \\ \qquad + \boldsymbol{f}_i + \alpha\, h\, c_0 \dfrac{\rho_0}{\rho_i}\displaystyle\sum_j \pi_{ij}\,\nabla_i W_{ij}\, V_j \\ \dfrac{D\boldsymbol{r}_i}{Dt} = \boldsymbol{u}_i \qquad\qquad p_i = c_0^2(\rho_i - \rho_0) \end{cases} \tag{1}$$

where ρ_i, V_i, p_i are respectively the density, the volume and the pressure of the $i-$particle while $\boldsymbol{r}_i$ and $\boldsymbol{u}_i$ are its position

SPH-ALE for simulations of rotor-stator interactions

M. Neuhauser, F. Leboeuf
Laboratory of Fluid Mechanics and Acoustics
Ecole Centrale de Lyon, University of Lyon
Ecully, France
magdalena.neuhauser@ec-lyon.fr

J.-C. Marongiu, M. Rentschler, E. Parkinson
Hydraulic research department
ANDRITZ Hydro
Vevey, Switzerland
jean-christophe.marongiu@andritz.com

***Abstract*—This paper shows the potential of a weakly compressible SPH-ALE method to simulate transient flows in hydraulic machines. The novelty of the approach is to use the properties of SPH-ALE in order to simulate rotor-stator interactions without a rotor-stator interface. Due to the ALE formalism, the particle velocity is a free parameter and can be chosen independently of the flow velocity. Instead of a rotor-stator interface, we have blocks of particles with different particle velocities. To validate the results, the flow field around a static airfoil and the pressure coefficient on the profile are compared to the results of an in-house Euler solver which is an inviscid finite volume code. Results of transient simulations prove the capability of the method to detect unsteady pressure waves and emphasize its applicability to study global phenomena in multistage machines.**

I. Introduction

The simulation of transient flow configurations in hydraulic pumps is still a challenge for standard numerical methods, especially off-design conditions as start up and shut down of units. In particular, mesh based methods face difficulties at mesh interfaces between rotating and static parts of the machine. SPH on the contrary is a mesh-less method and does not have the problem of mesh interfaces. It has also proved to be advantageous if free surfaces and multiphase flow exist as for example during the filling process of a hydraulic machine. This work aims at showing the potential of a weakly compressible SPH-ALE method for these applications. This is done on the basis of a 2D testcase of two symmetric NACA airfoils, where one (the rotor) is moving with a given velocity and the other one (the stator) is static.

II. The SPH-ALE formalism

We consider the inviscid compressible Euler equations together with Tait's equation of state for weakly-compressible fluids. Vila proposed in [10] to write a weak form of the conservation equations in an arbitrarily moving frame of reference. Discretizing the equations by the SPH operators leads to the following discrete set of SPH equations in Arbitrary Lagrange Euler (ALE) formalism,

$$\begin{cases} \frac{d}{dt}(x_i) = v_0(x_i,t), \\ \frac{d}{dt}(\omega_i) = \omega_i \sum_{j\in D_i} \omega_j (v_0(x_j) - v_0(x_i)) \nabla_i W_{ij}, \\ \frac{d}{dt}(\omega_i\rho_i) + \omega_i \sum_{j\in D_i} 2\rho_{ij}^E \left(v_{ij}^E - v_0(x_j,t)\right)\cdot \nabla_i W_{ij} = 0, \\ \frac{d}{dt}(\omega_i\rho_i v_i) + \omega_i \sum_{j\in D_i} \omega_j 2\left[\rho_{ij}^E v_{ij}^E \otimes (v_{ij}^E - v_0(x_{ij},t)) \right. \\ \qquad \left. + p_{ij}^E\right]\cdot \nabla_i W_{ij} = \omega_i \rho_i g, \end{cases}$$

where $(\rho_{ij}^E, v_{ij}^E)^T$ is the upwind solution of the moving Riemann problem at the midpoint between pairs of particles i and j. In order to calculate this solution of the Riemann problems the field variables are extrapolated at the interface following the MUSCL procedure. Then a linearized approximate Riemann solver calculates a mean of these extrapolated states and expresses the corresponding solution analytically. For the MUSCL procedure, gradients of the field variable are needed, whose accuracy have an influence on the numerical diffusion of the scheme. More details on the formalism can be found in [8]. The frame of reference moves with an arbitrary velocity v_0. This particle velocity can be chosen independently of the fluid velocity. This property of the SPH-ALE formalism will be used in the following for the simulations of rotor-stator systems.

III. Flow around a static NACA airfoil

Before simulating rotor-stator systems, the flow around a two-dimensional symmetric airfoil developed by the National Advisory Committee for Aeronautics (NACA) was computed and validated. The solid geometry of the 4-digit NACA profile is described by

$$\begin{aligned} y &= \frac{t}{0.2}c\left[0.2969\sqrt{\frac{x}{c}} - 0.1260\frac{x}{c} - 0.3516\left(\frac{x}{c}\right)^2\right. \\ &+ \left. 0.2843\left(\frac{x}{c}\right)^3 - 0.1015\left(\frac{x}{c}\right)^4\right], \end{aligned} \quad (1)$$

where c denotes the chord length and t the maximum thickness as a fraction of the chord (see [1], p. 113-114). Figure 1 shows the geometry set up of the simulation. Non-reflecting boundary at infinity conditions inspired by [5] are applied to the inlet situated on the left hand side and the outlet on the right hand side with $v_\infty = (0.1, 0)^T$ m/s and $p_\infty = 0$. In the z-direction periodicity is applied. A chord length $c = 0.1m$ was chosen and the thickness t is 20% of the chord length. The blade channel distance is one chord length. All simulations are in Eulerian mode; that means that particles do not move. In consequence, an initial particle distribution was needed which was obtained by Bouscasse's packing algorithm [2]. This algorithm minimizes $\nabla\Gamma_i = \sum_{j\in D_i} \omega_j \nabla W_{ij}$ and yields a regular particle distribution. The term $\nabla\Gamma_i$ plays an important role in the accuracy of the SPH scheme. It should be zero in order to reproduce correctly the gradients of a constant field. Since particles do not move, this regular initial distribution

FPM Simulations of a 3D Impinging Jet on a Flat Plate Comparison with CFD and Experimental Results

C. Vessaz, E. Jahanbakhsh and F. Avellan
EPFL, Laboratory for Hydraulic Machines
École polytechnique fédérale de Lausanne
Av. de Cour 33 bis, Lausanne, Switzerland
christian.vessaz@epfl.ch, ebrahim.jahanbakhsh@epfl.ch, francois.avellan@epfl.ch

***Abstract*—The present paper reports the development of Finite Particle Method (FPM) in the framework of 3D impinging jet simulations. The FPM kernel has been proposed recently in the literature to improve the consistency of standard SPH. Moreover, a new particle shifting method is proposed to mitigate the particle clustering issue. The influence of particle spacing, maximum CFL number and different particle shifting methods are investigated in the case of a water jet impinging perpendicularly on a flat plate. The results are compared to available measurements and grid-based CFD simulations.**

I. Introduction

The Smoothed Particle Hydrodynamics (SPH) method is well adapted to simulate free surface flows. Indeed, the Lagrangian approach of the SPH enables to simulate complex free surface flows such as in the case of an impinging jet on a flat plate. This test case has been often used in 2D to validate SPH simulations with the analytical solution of Taylor e.g. [1], [2] and [3]. The validation of the simulations is important to validate the SPH solver before extending the fluid analysis to more complex flows. For instance, the simulations of a Pelton turbine [4] and [5] which require an accurate solver to compute the instantaneous torque in the buckets.

The standard SPH method described by Monaghan [6] suffers from a lack of accuracy compared to the grid-based numerical simulations. However, recent studies summarized by Liu and Liu [7] have proposed new developments, which improve the accuracy of the particle-based methods. In the present study, the FPM kernel from Fang et al. [8] is used to improve the consistency of standard SPH. The numerical oscillations are reduced by the mass equation correction proposed by Fatehi and Manzari [9]. Finally, the temporal integration scheme of Molteni and Colagrossi [10] is implemented to increase the overall accuracy and stability of the simulations.

During the SPH simulations, the contraction of the streamlines as well as the tensile instability described by Monaghan [11] result in the clustering of particles. This particles clustering increases the spatial discretization error, which decreases the overall accuracy of the simulation. In the present study, a particle shifting method is applied at the end of each time step to restore a better particles distribution. The shifting methods of Xu et al. [12] and Jahanbakhsh et al. [1] are investigated in the case of the simulations of the impinging jet. Moreover, a new particle shifting strategy which combines the benefits of these two methods is proposed.

The FPM simulations of the 3D impinging jet on a flat plate are performed with the code SPHEROS developed by Jahanbakhsh et al. [13]. The accuracy of these simulations are validated with the numerical simulations and experimental results from Kvicinsky et al. [14]; the numerical simulations being based on the grid-based volume of fluid method implemented in the commercial software ANSYS-CFX®.

II. Governing Equation

A. Restoring Consistency

The standard SPH formulation of a function (1) and its derivatives (2) is based on a kernel approximation and a decomposition of the continuous matter into N particles.

$$f^{(i)} = \sum_{j}^{N} f^{(j)} W^{(ij)} V^{(j)} \tag{1}$$

$$f_{\alpha}^{(i)} = -\sum_{j}^{N} f^{(j)} W_{\alpha}^{(ij)} V^{(j)} \tag{2}$$

Each particle i (or j) has a mass m, a density ρ and a volume V. The position of the particle is defined by the vector $\boldsymbol{X}$ whose components of the Cartesian coordinate system are X_α (or X_β) with $\alpha = [X, Y, Z]$. The approximated function f and the kernel W are expressed as (4) and (5) to simplify the notation. In the present work, the quintic Wendland kernel (6) used by Fatehi and Manzari [9] is chosen. This kernel is used with a constant smoothing length $h = 2.6X_{ref}$ where X_{ref} is the reference particle spacing.

$$V^{(i)} = \frac{m^{(i)}}{\rho^{(i)}} \qquad \text{and} \qquad X_{\alpha}^{(ij)} = X_{\alpha}^{(i)} - X_{\alpha}^{(j)} \tag{3}$$

$$f^{(i)} = f\left(\boldsymbol{X}^{(i)}\right) \qquad \text{and} \qquad f_{\alpha}^{(i)} = \frac{\partial f^{(i)}}{\partial X_\alpha} \tag{4}$$

$$W^{(ij)} = W\left(\boldsymbol{X}^{(ij)}, h\right) \qquad \text{and} \qquad W_{\alpha}^{(ij)} = \frac{\partial W^{(ij)}}{\partial X_\alpha} \tag{5}$$

Remeshed Particles: a robust and efficient method for multiphysics simulations

Wim M. van Rees and Petros Koumoutsakos*
Chair of Computational Science, ETH Zürich
Zürich, Switzerland
*petros@ethz.ch

***Abstract*—We present our recent advances for particle method simulations of continuum and discrete systems. For continuum systems, the approach is based on a Lagrangian formulation of the governing conservation laws, combined with the remeshing of distorted particles on a regularized grid to ensure accuracy and convergence. The remeshing step detracts from the grid-free character of the method, but enables advances such as fast computations of derivatives, fast grid solvers for elliptical equations and the use of multiresolution techniques. In a discrete setting, the approach relies on interacting particles to simulate complex system behavior through the formulation of deterministic or stochastic discrete rules of interaction that underly the system.**

In fluid dynamics, as an example of the continuum approach, we use remeshed particles to solve the incompressible Navier-Stokes equations in velocity-vorticity formulation. The accuracy and efficiency of our results for high Reynolds number vortical flows are validated in a comparison with a pseudo-spectral method. Combining the remeshed particle method with a volume penalization technique to handle no-slip boundary conditions on arbitrary geometries allows us to solve fluid-structure problems, including the simulation of single and multiple three-dimensional self-propelled swimmers in viscous flows.

We also discuss recent advances in particle methods for the simulation of biological systems at the mesoscopic and the macroscale level. We present results from remeshed particle method simulations of reaction-diffusion on deforming surfaces, tumor growth and angiogenesis. For cell biology systems a subgrid-scale modeling approach based on discrete particles, the Subcellular Element Model, is used. Here each cell is modeled using a collection of soft particles which can be seen as a coarse-grained representation of a cell's cytoskeleton. The particles interact according to intra-cellular and inter-cellular potentials which, together with the internal Brownian dynamics and empirical rules on cell growth, division and migration, determine the system's behavior.

Finally, we demonstrate how we exploit scale-separations in physical systems by integrating our remeshed particle methods in a wavelet-based multi-resolution framework. We will report on the implementation details of the approach and show the performance improvements obtained for problems of flow-structure interaction.

I. Introduction

Simulations of interacting particles is a unique, deceivingly simple, yet robust and accurate method for exploring physical systems ranging from cellular ion channels to galaxy formation [1]. Particle methods formulate physical systems as interactions between evolving particles. They were the first method used to describe the simulation of physical processes (in the 1930's hand made calculations by Rosenhead of the evolution of a vortex sheet [2]) and they have been advocated for efficient simulations of multiphysics phenomena in several fields of science ranging from astrophysics to fluid and solid mechanics (see the review papers [3]–[6] and references therein).

Particle methods are unique, in that they can be used to simulate phenomena ranging from the atomistic scale (as in Molecular Dynamics) to the mesoscale (as in kinetic models of complex physics) and the macroscale (as in fluid, solid mechanics and astrophysics). In addition they can be readily formulated to describe discrete and continuous processes as well as deterministic and stochastic models. In recent years starting from the development of particle methods for the simulation of three-dimensional vortical flows [7], these techniques have been extended to the simulation of continuous and discrete processes in biological systems [8].

For continuum simulations, where particles interact and adapt according to a convection velocity field, the non-uniform distortion of the computational elements prevents the convergence of the method [9], [10]. Hence particles evolve while conserving moments of the field they aim to discretize, albeit inconsistently with the equations that govern their evolution. This observation is often overlooked in simulations using particles but we consider that particle distortion and the ensuing inaccuracy of the method are inherently linked to the Lagrangian description of particle methods. In order to correct for this inaccuracy, we advocate particle regularization by remeshing the particles periodically on grid nodes [11]. Remeshing detracts from the grid-free character of particles but enables advances such as grid-based derivative operators, fast Poisson solvers and multiresolution.

In this work we present an overview of the developments in particle methods for multiphysics simulations. The paper is structured as follows. In section II we recapitulate the fundamental equations and ideas behind the methods. Section III will discuss a number of applications of the continuum methods, particularly related to fluid dynamics and biology, and an application of the discrete method from the field of cell biology. Section IV will discuss recent developments to combine the Lagrangian adaptivity of particle methods with wavelet-based multi-resolution representations of multi-scale physical systems, and their implementation on modern

Use of SPHERA Code To Investigate Local Scouring Effects Induced By Fluvial Structures Downstream a Barrage

Giordano Agate, Roberto Guandalini
Environment and Sustainable Development Dept.
RSE – Ricerche Sistema Energetico SpA
Milan (Italy)
giordano.agate@rse-web.it;roberto.guandalini@rse-web.it

Sauro Manenti, Stefano Sibilla, Mario Gallati
Hydraulics and Environmental Eng. Dept.
University of Pavia
Pavia (Italy)
sauro.manenti@unipv.it; stefano.sibilla@unipv.it

Abstract— **The paper reports the development and application of the SPHERA code (SPH based) towards the simulation of the local scouring problems induced by fluvial structures. The study is motivated by the traditional difficulties of SPH approaches to mimic the sediment transport and local scouring downstream a barrage. In the frame of the Italian Energetic System Research Projects a work devoted to develop innovative numerical models, based both on a classic CFD and on a new SPH approaches, has been carried out with the goal of improving the accuracy of the results and to support very detailed analyses. The paper shows the application of the two models to simulate the local scouring effects induced by a barrage, comparing the accuracy of the results with experimental measurements and discussing advantages and limits of each model. The results allow to conclude that both the CFD model and SPH innovative model can be applied successfully to investigate local scouring problems when fixed structures are involved and they can be used in conjunction with the physical models to investigate safety aspect and to support optimization design of fluvial structures.**

I. INTRODUCTION

In the proximity of the structures built on rivers, as barrage, pile of bridge, intake works, etc., the fluent water erosion action can induce a local scour in the riverbed. This action depends from the combined effect of the properties of the bed material (dimension, type of sediment) and from the hydraulic characteristics of the current (velocity, turbulence) induced by the presence of the hydraulic works.

In the frame of the structures that are of interest of electric system, the river barrages of hydroelectric plants, can lead an erosion process at a local scale in the riverbed downstream the barrages. It is to note that these effects can be amplified by other generalize erosion phenomena at a scale of the river (maneuver of regulation of the riverbed). If the local erosion action of the riverbed is not reduced, it can lead to a partial excavation around the foundation of the civil works and at the base of the river banks, with the consequence to put in danger the static stability of the structures and of the bank. For this reason both for the new works and for the existing ones, it is important to analyze this problem in order to take action to adjust and to protect the hydraulic works and/or the riverbed.

II. THE PHYSICAL MODELLING APPROACH

The evaluation of the erosion of the riverbed importance and the evaluation of the action needed to oppose it (i.e. layout modification of the hydraulic work or covering the riverbed), can be faced with the support of scaled physical models. These models are right for comparing different solutions and for highlight the configuration that minimize the riverbed erosion. The Figure 1 shows the experimental facilities that reproduce a real barrage in a river at a scale of 1:35.6 considered in the study as reference.

The sediment in the riverbed, placed downstream the middle outlet of the barrage, is composed of mono-granular sand with d50=500 μm and ρ_s=2650 kg/m^3. The barrage has five outlet, but only the middle one is opened. Upstream the barrage a pump system puts into the model a constant flow rate of 100 l/s (0.1 m^3/s). The experimental measurements are:

1. flow rate at inlet;
2. water level upstream and downstream the barrage in different points;
3. the bathymetry measured at the end of simulation and after removal of water.

The tests have pointed out that the water current modify the bathymetry causing two print excavated on the bed extended along the water flow current direction and separated from a zone with less erosion/deposition. It is to note that the erosion activity induced by water excavates the bed downstream the barrage in proximity of piles. The considered test takes 30 minutes of simulation.

III. THE NUMERICAL MODELLING APPROACHES

The numerical models, compared to the physical one, allow to study the problems without operation of scale and to analyse a great number of layout and/or boundary conditions, but the development and the validation of the

SPH Modelling of Propeller Induced Harbour-Bed Erosion by a Container Vessel

Christian Ulrich and Thomas Rung
Institute for 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 applicability of Smoothed-Particle-Hydrodynamics (SPH) for simulations of ship propeller induced scours in shallow water settings such as harbour basins. Respective erosions represent unpleasant phenomena, especially if they occur close to quay walls, and generate cost intensive counter measures. These measures are usually based on a rather weak background knowledge. SPH simulations can help to analyse the erosional processes and to understand the interaction between ship, water, soil and structure. In the present research, a body-force propulsor model based on open water characteristics is used to represent the ship's propeller. Vessel motions are captured by a 6DOF motion solver. The evolution of the liquid and granular phase particles is obtained from an SPH-integration of the continuity and momentum equations. The fluid is considered to be Newtonian and the viscosity of the soil-phase is modelled in line with the Mohr-Coulomb yield stress criterion. Water and soil particles interacting in a suspension layer are assigned to a viscosity that is derived from a Chézy-relation between the shear stress and the local flow velocity. A variable particle resolution strategy is applied to handle large domains, in which the areas around the ship hull demand a fine resolution. A complex full-scale application example included refers to the starting sequence of a container ship propeller.**

I. Introduction

The interaction of water, soil and structures poses problems to different areas of marine and hydraulic engineering. In the field of port-engineering, erosions of the harbour bottom, especially close to quay walls, represent unpleasant phenomena. In particular, scours may significantly weaken the structural support of wharf constructions and generate costly counter measures. The formation of such erosional phenomena usually follows from flows induced by the ships' manoeuvering and propulsion devices as illustrated in Fig. 1. Our research aims at the development of a numerical SPH-based procedure which is capable of accurately predicting the scour formation process. While previous studies [15], [17], [19] focused on the impact of transverse thrusters, the present work is devoted to simulations of propeller flow induced erosion. The latter is not only a typical problem in harbours but generally occurs in waterways that provide restricted water depths. Figure 2 shows a large scour generated by a ship propeller in an inland water way during in-situ experiments performed by Felkel and Steinweller [2]. The picture outlines that scours can reach significant depths and may seriously damage the channel/harbour bed. As it can handle the transient dynamics of multiple

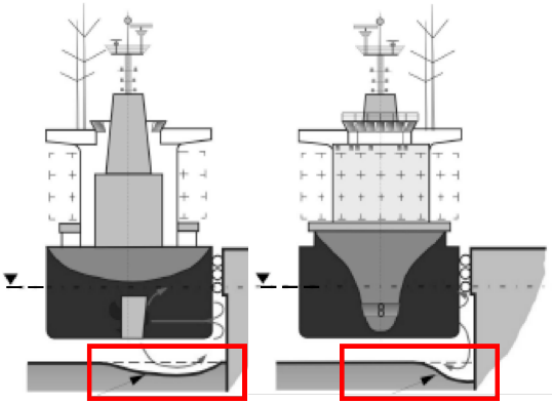


Fig. 1. Scours close to quay wall caused by ship propeller (left) and transverse thruster (right). Picture from [10].

interacting continua featuring large relative motions, the SPH method forms an ideal candidate to simulate the respective complex phenomena.

The paper outlines the application of the massively-parallel hydrodynamic SPH-code GADGET-$^{H}2^{O}$ [16] to study the erosional mechanisms induced by starting ship propellers. The GADGET-$^{H}2^{O}$-procedure is a modification of Springel's [13] cosmological TreeSPH-Code GADGET-2. GADGET-$^{H}2^{O}$ is seen to provide linear speed up for several hundred CPU-cores when applied to hydrodynamic flow simulations [16] using many ten-million particles. Concerning the physical modelling, fluids are assumed to be Newtonian and turbulence is modelled by means of an LES approach. The soil model considers the granular material as a fluid with a variable viscosity which is evaluated in line with the Mohr-Coulomb yield-stress criterium for cohesive or cohesionless materials. Water/soil suspensions are taken into account by a concentration based approach to mimic the stresses inside a fictitious suspension layer which is derived from a Chézy-relation between the shear stresses and the local flow velocity as proposed e.g. by Fraccarollo and Chapart [3]. The employed motion modeller for floating bodies is based on unit-quaternions and does not exhibit a limitation of the rotative motion as displayed by traditional Eulerian-Angles techniques. The ship propeller is modelled by a semi-empirical actuator disk propeller model. In addition

SPH simulations of bow waves dynamics

Bouscasse B.
CNR-INSEAN
Rome, Italy

Marrone S.
CNR-INSEAN
Rome, Italy

Colagrossi A.
CNR-INSEAN, Rome, Italy
CESOS, NTNU, Trondheim, Norway

Broglia R.
CNR-INSEAN
Rome, Italy

***Abstract*—The present study aims to provide further understanding of the dynamics of the ship bow waves. From a physical point of view, it is important to understand how the bow wave breaking process develops and how it modifies the wake field. To tackle the problem the ship bow is modeled with simple wedge geometries moving through the water in stationary forward motion at different speeds. For this geometry, experimental measurements are available in the literature (see Waniewski [4]). The problem has been simulated using the $\delta - SPH$ model and an in depth analysis of the 3D wave pattern has been done for different bow shapes and for different Froude numbers. The latter has been chosen to get overturning bow waves generating intense splash-up cycles. A preliminary 2D+t analysis has been performed to identify the main characteristic lengths of the bow wave plunging jet. In order to be able to carry intensive computations an SPH hybrid MPI-OpenMP programming model has been used. The physical characteristics of the bow breaking wave are investigated and the outcomes are compared with both experimental and numerical results from Finite Volume Level Set (FVM-LS) calculations.**

I. Introduction

A. State of the art

Bow wave dynamics have been for a long time a subject of theoretical and analytical research in the naval hydrodynamic field (Ogilvie (1972) [1], Noblesse (1991) [2]). All along the years, impressive progress have been made in the numerical simulation of these kind of flows (see e.g. Miyata [3], Wyatt [6], Iafrati [7]). At present, some of the most used numerical models for free-surface flows are Finite Volume schemes with level-set or volume of fluid algorithms. These solvers are able to deal with 3D breaking wave phenomena and, specifically, they can simulate the evolution of bow waves dynamics (see *e.g.* Carrica [8] and Di Mascio [9] [10]).

Recently, SPH method has been used to study the dynamics of breaking waves proving to be a reliable numerical method for this kind of phenomena (see *e.g.* [12], [13]). Colagrossi et al. [14] proposed a 2D+t SPH model that has been proved to provide a good qualitative description of the breaking phenomenon. A further inspection of this model has been provided in [15] where the minimum discretization required to catch the breaking inception has been determined.

In [16], a 3D SPH solver was described and tested in order to model the 3D wave pattern generated by a ship in steady-state forward motion, and a first attempt was done on a simplified geometry. The subject is further developed in the present paper. To avoid complexities linked to the use of a variable-h SPH scheme a constant discretization was used for the whole fluid domain. Further, the domain has to be large enough to avoid wave reflections at the boundaries. As a consequence, the number of particles used in the simulations is of the order 10^7. For this reason, the hybrid MPI-OpenMP programming model, developed in [16], has been adopted.

B. Scaling of the experimental data from model test

The bow wave dynamics has been historically studied using simplified geometry as a wedge. Several experimental campaigns on wedges have been made by Miyata [3], Waniewski [4] and Karion [5] while a numerical study have been done in Broglia [11].

Generally, for this kind of flows, viscosity and surface tension effects can play a significant role. Therefore, together with the Froude number, Reynolds and Bond (or Weber) numbers should be taken into account when scaling experimental results from model test. The experimental results performed by Waniewski [4] clearly show that measurements of different model scales in Froude analogy present non-negligible discrepancies. In particular, the analysis confirms a substantial influence of the Froude number on the patterned morphology and highlights the importance of surface tension for small-scale bow ships (e.g. small-scale models tested) in inhibiting the plunging jet formation and evolution.

In such kind of analysis the Froude number is generally defined respect the ship draft H and therefore it reads as $Fn_H = U/\sqrt{gH}$ where U is the forward ship speed and g the gravity acceleration. Reynolds number and Bond number are defined as $Re = U\,L/\nu$ and $Bo = \rho g L_j^2/\sigma$, where L is the typical ship length (here the extension of the bow region), ν the kinematic viscosity, ρ the density of fresh water, L_j is the characteristic curvature radius of the plunging and σ the water surface tension. Regarding the Bond number, L_j is of the same order of H in the problem studied in this work.

Miyata [3] noted that the wave system generated by a traveling wedge is quite similar to the Free Surface Shock Waves characteristic of shallow water regime. In particular, he shows that the wave system in an open channel flow with a current in supercritical regime ($Fn_H > 1$) is quite similar to the one generated by the bow of a ship. Following this analogy, Waniewski [4] shows that the ratio between the model draft and the water depth has no significant effect on the bow wave profile. The bow waves profiles on a wedge model towed in a tank (where the water depth is by far larger than the wedge draft) are similar to those obtained in a stationary open-channel where the wedge is fixed to the bottom and, therefore, the water depth coincides with the wedge draft.

Using SPHysics to Simulate a Wigley Hull in Head Waves

Mark Pearce, Dr. Glyn Thomas, Dr. Dominic Hudson
School of Engineering Sciences
University of Southampton
Southampton, United Kingdom
mgp104@soton.ac.uk, T.G.Thomas@soton.ac.uk, dominic@soton.ac.uk

***Abstract*–This paper describes a novel methodology, using the open source SPH code SPHysics, that is capable of simulating a Wigley hull in a wave tank. To achieve this capability a number of modifications made and new features added. These are a Wigley hull geometry modelled as a floating body, a new inflow condition based on the Airy wave theory and a two-part outflow condition that acts as a non-reflecting open boundary. To verify the capability of the new methods a series of test cases have been conducted. These are 2D and 3D wave tank simulations to test the inflow and outflow conditions and finally a Wigley hull is simulated in a wave tank to test all of the new methods simultaneously.**

I. INTRODUCTION

The methods most frequently used to conduct computer simulations of ship motion are the older, more mature methods; including Strip Theory and Reynolds Averaged Navier-Stokes (RANS). However other methods are becoming more popular, including SPH. In this paper we use the open source SPH code SPHysics [1] to simulate a Wigley hull subjected to head waves in a wave tank. Firstly a number of modifications and new methodologies have been introduced to allow SPHysics to model the simulation correctly. Secondly a number of test cases have been conducted. This work follows on from the results presented in [2].

The first set of test cases discussed involve testing the inflow and outflow conditions to ensure that the waves generated can propagate correctly and then can leave the computational domain without any unphysical wave reflection. The second set of test cases bring all of the new features together to simulate a ships' hull in a wave tank; for this a source of validation data is required. The results from a number of Wigley hull wave tank experiments have been detailed in [3,4]. In these experiments the Wigley hull is subject to regular head waves where the wavelength, wave amplitude and Froude number *(Fn)* are varied. [3,4] give the results from a number of different experiments where the hull is either restrained or free to heave and pitch.

II. FORMULATIONS AND METHODOLOGY

The latest version of serial SPHysics (V2.2.001) does not have all the features required to simulate a ship's hull in a wave tank. To enable the simulations to be conducted a number of modifications to the code have been made and new features have been added.

A new inflow condition has been implemented that is based on the Airy wave theory that allows a mean flow speed, wavelength and wave amplitude to be prescribed. A novel outflow condition has been created to compliment the inflow condition, this is split into two parts. The first part uses a sponge layer to damp out waves so that any wave that is reflected off the outflow boundary will be damped out before it reaches the main domain. The second part uses a density based system to regulate outflow to ensure that the average mass flux across the boundary remains constant. Finally the floating body feature in SPHysics has been modified so that it can model a more complex geometry such as a Wigley hull.

A. SPHysics

A full account of the features available in SPHysics version 2.2.001 can be found in [5], in this section only the original formulations that have been altered for this project will be discussed.

The repulsive boundary condition used in SPHysics models a solid boundary by using a series of boundary particles which impart a repulsive force on any fluid particle that approaches the boundary. The force is defined by a delta function and is further modified by a fluid depth and particle velocity correction factor.

The wavemakers included in SPHysics use moving walls that act like pistons or paddles and are designed to generate waves in a static volume of fluid. This is not appropriate for wave tank tests where there is a free stream flow velocity, such as in [3,4]. Therefore a new wavemaker method that can include a free stream velocity has been developed.

B. Altered and Novel Methodlogies

In this section the alterations and additions made to the original code will be discussed.

The delta function used by the repulsive boundary condition can result in very large forces being applied to fluid particles if the particle is very close to the boundary. This can cause problems during a simulation and may require an decrease in the time step to maintain stability. The distance between the fluid particle and the boundary particle is normalized by the smoothing distance such that:

Efficient parallelisation of 3D SPH schemes

David Guibert, Matthieu de Leffe
Hydrocean
Nantes, France
David.Guibert@hydrocean.fr

Guillaume Oger
LHEEA laboratory
Ecole Centrale de Nantes / CNRS
Nantes, France
Guillaume.Oger@ec-nantes.fr

Jean-Guillaume Piccinali
CSCS-Swiss National Supercomputing Centre
Lugano, Switzerland
jgp@cscs.ch

***Abstract*—The SPH-flow code developed by Ecole Centrale de Nantes and HydrOcean relies on SPH methods to model complex free surface problems involving fast dynamics, such as the water entry of 3D complex geometry objects. SPH methods suffer from high computational costs, which increase dramatically in 3D engineering applications. To increase the use of SPH methods in industrial studies, large simulations must be solved in a reasonable restitution time. Such simulations involve hundreds of millions of particles and, therefore, require important computational resources and an efficient parallelization.**

The parallelization strategies adopted in this work takes advantage of a domain decomposition method, classically used in mesh-based methods. They solve the two main bottlenecks of SPH codes by introducing a spatial distribution of particles based on a sampled Orthogonal Recursive Bisection (ORB) and non-blocking MPI communications after identifying border and inner particles.

Very good results have been obtained up to 32192 cores with 3 billions particles, and industrial studies have been performed up to 100 million particles.

I. Introduction

The objective of the present work is to improve the performance of the HPC code SPH-flow developed jointly by Ecole Centrale de Nantes and HydrOcean, to make it efficient in research as well as in industrial contexts. For this purpose, the underlying objective is to keep the presence of all the main existing SPH formulations in this code, in order to ensure the accuracy of the results and their applicability in the industrial field. The implemented methods are for example the standard SPH scheme [11], but also the δ-SPH scheme [10], or the Vila scheme [15] based on Rieman solver. Similarly the renormalized scheme, the ability to resolve 2-D or 3-D multifuids flow [6], the Fluid-Structure interaction (FSI) based on coupling SPH with a finite element code [5], advanced boundary conditions [3], [7], [9] and viscosity [8] are maintained in this HPC code. It is important to keep all of these schemes in mind, since it guides the algorithmic choices made in this paper.

The presented work has been developped into SPH-flow code but could be easily reused elsewhere. The main difficulties of the implemented models of this work arise from variable kernel supports of the particles, from interaction lists that are built for several time steps in Verlet list framework, and from boundaries that could be defined by ghost particles or with surfacic terms.

In Section II, the main aspects of the parallelisation of SPH methods are introduced. The main phases of SPH methods will be presented, relying on the load balance of the particles throughout processes, the parallel computation of the flux terms and the time advance procedures. In Section III, parallel results will be shown. Several tests have been performed: strong scalability tests where the problem size is fixed while the number of processes increases, and weak scalability tests where the problem size is fixed per process. Finally, industrial test cases involving massively parallel SPH simulations will be presented in Section IV.

II. Overview of the parallelisation of SPH methods

Generally parallel SPH methods rely on three main steps

1) definition of the domain of each processes and distribution the particles accordingly to the domain decomposition.
2) positioning of the particles in a grid and eventually build the interaction list.
3) computation of the flux terms and perform the time advance.

When the discrete SPH model is defined with constant or pseudo constant kernel support size (namely constant-H), steps 1 and 2 could be implemented as finite difference methods. It means that an underlying regular grid is built and particles are then placed in its cells. Then the domain decomposition could be performed on the data structure defined by these cells.

But in SPH-flow, the SPH models could be solved using variable-H, which introduces much more difficulties. The next two subsections will describe a load balance procedure and flux computations with overlapping communications. Both approaches are compatibles with variable-H SPH formulations (for more details see Section IV).

A. Load balance procedure

To parallelize explicit codes, such as SPH methods, the basic approach is to spread the discrete points, here the particles, among the processes. Hence the computation of fluxes can be performed in parallel whilst the neighbour particles are exchanged.

This method is mainly called domain decomposition. The whole computational domain is divided into subdomains. Each subdomain is affected to a single process. To obtain good

New OpenMP-MPI-CUDA implementation for parallel SPH simulations on heterogeneous CPU-GPU clusters

Jose M. Domínguez, Alejandro J.C. Crespo, Moncho Gómez-Gesteira
Environmental Physics Laboratory (EPHYSLAB)
Universidade de Vigo
Ourense, Spain
jmdominguez@uvigo.es

Daniel Valdez-Balderas, Benedict D. Rogers
Modelling and Simulation Centre (MaSC),
School of Mechanical, Aerospace and Civil Engineering (MACE)
University of Manchester
Manchester (U.K.)

***Abstract*—A massively parallel SPH scheme using heterogeneous clusters of CPUs-GPUs is developed. The new implementation has been carried out starting from the single-GPU DualSPHysics code that has been proven to be powerful, stable and accurate. A combination of different parallel programming languages is merged to exploit not only the different cores of one device (CPU or GPU) but also the combination of different machines. The communication among devices is carried out through a new MPI implementation. The proposed implementation tries to address some of the well-known drawbacks of MPI by including a dynamic load balancing and the overlapping between data communications and computation tasks. The efficiency and scalability obtained with the new DualSPHysics code are analysed for different numbers of particles and different number of GPUs. Last, an application with more than 170 million particles is performed to show the capability of the code to handle simulations that could only be carried out using big CPU cluster machines or supercomputers.**

I. INTRODUCTION

One of the main drawbacks of the SPH method is its high computational cost when real engineering problems must be studied using a huge number of particles. It is imperative to develop parallel implementations of SPH capable of combining the resources of multiple machines allowing simulations of millions of particles at a reasonable runtime. The use of graphic processing units (GPUs) has become an affordable option to accelerate SPH with a very low economic cost (compared to traditional CPU clusters). However, the use of a single GPU card is not enough when several million particles are involved since the execution times are high and the available memory space is insufficient. Therefore, for large simulations is essential to gather the performance of multiple GPUs.

This work presents different parallelisation approaches to accelerate the SPH codes. On the one hand, Open Multi-Processing (OpenMP) and Compute Unified Device Architecture (CUDA) programming frameworks are implemented to take advantage of the different cores of one CPU and one GPU respectively. And on the other hand, a new parallel implementation using the Message Passing Interface (MPI) is used to combine different machines (CPUs and/or GPUs) making possible the execution of SPH on heterogeneous clusters.

These parallel implementations are available at the open-source code DualSPHysics (www.dual.sphysics.org) that is based on the SPH code named SPHysics [1]. The single GPU code has been shown to achieve speedups of up to two orders of magnitude compared to the CPU code [2]. Thus, the new MPI implementation was designed starting from the already optimised DualSPHysics code for CPU and GPU [3].

The efficiency and performance of the new OpenMP-MPI-CUDA implementation are presented and analysed in this work. This new heterogeneous version allows a more efficient use of different machines with both multi-core CPUs and/or different GPU cards. The simulation of hundreds of millions particles are now possible in small clusters of CPUs and GPUs.

II. PARALLEL IMPLEMENTATIONS

This section describes the different parallel implementations applied to the DualSPHysics code. All of them consist of dividing the execution time among the different processing units and adapting to the specific features of each hardware.

The implementation of the SPH method consists of the iteration of three main steps: generation of a neighbour list, calculation of forces by computing particle interactions and updating all physical magnitudes of the particles of the system. The most expensive step is the force computation since it takes more than 90% of the total runtime using DualSPHysics on a single CPU . Therefore, all efforts will be mostly focused on accelerating the particle interaction stage.

A journey from single-GPU to optimized multi-GPU SPH with CUDA

E. Rustico, G. Bilotta, G. Gallo
Dipartimento di Matematica e Informatica
Università di Catania
Catania, Italy
{rustico,bilotta,gallo}@dmi.unict.it

A. Hérault
Conservatoire des Arts et Métiers
Département Ingénierie Mathématique
Paris, France
alexis.herault@cnam.fr

C. Del Negro
Istituto Nazionale di Geofisica e Vulcanologia
Sezione di Catania
Catania, Italy
ciro.delnegro@ct.ingv.it

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

***Abstract*—We present an optimized multi-GPU version of GPUSPH, a CUDA implementation of fluid-dynamics models based on the Smoothed Particle Hydrodynamics (SPH) numerical method. SPH is a well-known Lagrangian model for the simulation of free-surface fluid flows; it exposes a high degree of parallelism and has already been successfully ported to GPU. We extend the GPU-based simulator to exploit multiple GPUs simultaneously, to obtain a gain in speed and overcome the memory limitations of using a single device. The computational domain is spatially split with minimal overlap and shared volume slices are updated at every iteration of the simulation. Data transfers are asynchronous with computations, thus completely covering the overhead introduced by slice exchange. A simple yet effective load balancing policy preserves the performance in case of unbalanced simulations due to asymmetric fluid topologies. The obtained speedup factor closely follows the ideal one and it is possible to run simulations with a higher number of particles than would fit on a single device. efficiency of the parallelization.**

I. Introduction

The numerical simulation of fluid flows is an important topic of research with applications in a number of fields, ranging from mechanical engineering to astrophysics, from special effects to civil protection.

A variety of computational fluid-dynamics (CFD) models are available, some specialized for specific phenomena (shocks, thermal evolution, fluid/solid interaction, etc) or for fluids with specific rheological characteristics (gasses, water, mud, oil, petrol, lava, etc). The Smoothed Particle Hydrodynamics (SPH) model, initial developed by Ginghold and Monaghan [1] and Lucy [2], has seen a growing interest in recent years, thanks to its flexibility and the possibility of application to a wide variety of problems.

The flexibility of SPH comes at the cost of higher computational costs compared to other methods (e.g. mesh methods like finite differences or finite volumes). However, since it exposes a high degree of parallelism, its implementation on parallel high-performance computing (HPC) platforms is conceptually straightforward, significantly reducing execution times for simulations.

Among the many possible parallel HPC solutions, an approach that has emerged lately is the use of GPUs (Graphic Processing Units), hardware initially developed for fast rendering of dynamic three-dimensional scenes, as numerical processor for computationally-intensive, highly parallel tasks.

Although initial attempts to exploit the computational power of GPUs go back to the introduction of the first programmable shaders in 2001, the break-through for GPGPU (General-purpose Programming on GPU) was the introduction in 2007 of CUDA, a hardware and software architecture released by NVIDIA with explicit support for computing on GPUs [3].

While typically running a lower clock rates, a single GPU features a large number of compute units (for more recent cards, in the order of thousands of cores per GPU) and much higher memory bandwidth than what is found on standard desktop or server motherboards.

Although serial execution does not gain much from GPU execution, its large multi-core structure makes it the ideal computing platform for algorithms that exhibit a high level of parallelism on a fine data granularity, such as SPH. For such problems, a well-tuned GPU implementation can easily achieve two orders of magnitude in speed-up of standard single-core CPU implementations.

The cost-effectiveness and the ease of utilization of modern GPUs have led to a widespread usage of GPU computing even outside the commercial and academic world, leading to what some to claim to be the *GPU Computing Era* [4]. It should be mentioned, however, that some have criticized the enthusiasm for GPGPU as being 'excessive', showing that a well-tuned CPU implementation, optimized for execution on recent multi-core processors, often reduces the flaunted $100\times$ speedup reported by many works [5].

Parallelisation of a Finite Volume Particle Method code

Mihai Basa, Libor Lobovský, Nathan J. Quinlan
Mechanical and Biomedical Engineering
National University of Ireland, Galway
Galway, Ireland
nathan.quinlan@nuigalway.ie

***Abstract*—We present the parallelisation of a Finite Volume Particle Method code using MPI, employing overlapped computation and communication. We pay particular attention to the difficulties that may be encountered in developing and implementing a parallelisation strategy, and discuss the chosen solutions.**

I. Introduction

The Finite Volume Particle Method (FVPM) is a mesh-free method in which particles represent discrete overlapping volumes of fluid. Interactions between particles are based on flux functions weighted by the overlap between particle supports [1]. FVPM allows simple and accurate treatment of boundaries and has attractive conservation and consistency properties.

A principal drawback of the FVPM method is its high computational cost, even relative to SPH, another costly method. The processing capabilities of a single CPU are not sufficient for realistic simulations of large problems, hence the necessity for parallelisation. We chose MPI for the parallelisation due to its versatility in use on both shared memory and distributed memory systems.

II. Domain decomposition

The aim of domain decomposition is to divide and balance the computational cost between available processes as uniformly as possible, while minimising inter-partition communication costs and keeping the cost of the decomposition itself low.

Assuming a uniform distribution of particles, computational cost is roughly proportional to the area of each partition, while communication costs are proportional to the length of the boundaries between them. A better approximation that does not assume uniform particle distribution relates the computational load to the number of particles in a partition, and communication costs to the number of particles within interaction distance of partition borders. This model is still just an approximation. Other factors remain, that are measurable but hard to balance against, such as differences in the number of neighbours per particle (significant especially when dealing with non-uniform distributions), the presence and complexity of geometric boundaries or differences in particle types.

In the current implementation, the square-celled search grid is partitioned into a rectangular tiling, with as many tiles as processes. For simplicity, we chose a Cartesian tiling, although the implementation is in principle able to handle any arrangement of rectangular partitions. This simple space decomposition is sufficient for particle distributions that are approximately uniform and constant.

The number of tiles in the x, y (and z) directions are chosen such that the tiles' aspect ratio is closest to 1, in order to minimise boundary lengths. Because the number of tiles is equal to the number of processes, the Cartesian grid arrangement means that poor tile aspect ratios may be obtained for certain numbers of processes, such as prime numbers, or those with few and uneven factors. A more versatile partitioning method can be implemented without any fundamental change to the rest of the parallelisation.

The neighbour search algorithm used can handle grid cells with calibrated pitches, unconstrained by maximum smoothing lengths. The grid resolution can therefore be made fine enough for accurate balancing of loads between processes even in the presence of particles with large occasional smoothing lengths, as might occur in variable smoothing length simulations.

III. Data dependencies

Particles that interact with those of another partition form a border region around the inside of a partition's boundary. It is necessary to identify which particles belong to this region for two reasons:

- to prepare them for sending to all the necessary neighbouring partitions
- to avoid making any calculations on such particles until information about particles from other partitions has fully arrived (if overlapped communication and calculation is desired)

The second point implies that calculations performed on particles must happen in two stages: firstly on particles in the inner region, away from boundaries, and secondly (after communications have finalised) over particles in the border region.

A few complications appear which are specific to FVPM. The first is that in order to reconstruct function values at particle-particle interfaces, FVPM uses gradients calculated

Use of Complex Inlet Boundary Conditions For Accelerated Studies of Green Water Events

Csaba Pakozdi and Carl Trygve Stansberg
Offshore department

MARINTEK SINTEF
Trondheim, Norway
Csaba.Pakozdi@marintek.sintef.no

SINTEF SCORE-team: A. Lavrov, B. Lund, E.W. Bjønnes
F.O. Bjørnson, J.E. Olsen, J.O. Busklein, J.S. Seland
K.O. Lye, P. Skjetne, P. Klebert, T. Coudert, W. Yang
SINTEF
Trondheim, Norway

***Abstract*—Traditionally model tests play an important role in estimating hydrodynamic loads during so called green water events (i.e. when ocean waves break onto a ship decks or deck house). In order to predict the correct loads it is important that the characteristics of the incoming wave are realistic [1], [2]. In this work we present an approach to modeling green water events using SPH based on a complex velocity inlet boundary condition. In naval hydrodynamics practical engineering tools have been developed based on both experimental data and nonlinear random wave modeling including 3D linear diffraction theory which allows for estimation of the realistic waveforms for green water events. We use one such tool, KINEMA2 [1], for the purpose of prescribing the spatial and temporal variation of the wave as it breaks along the deck perimeter. This approach greatly reduces the number of degrees of freedom (SPH particles) needed to solve the problem, thus rendering the analysis feasible on a laptop computer. We compare SPH results for the resulting water elevation and velocity on deck with CFD and experimental time series from model test.**

I. INTRODUCTION

Severe storms have gained more attention in recent years. Improved metocean data have lead to new insight into severe wave conditions for marine design. Green water on deck and slamming on bow flares may be critical to FPSO integrity ([4]). This has also been confirmed from observation through model test studies ([5], [6])

A complete technical description of this strongly nonlinear processes is quite complex and it is hard to model properly by "classical" (panel methods based on potential theory) analysis methods. Therefore, there exists an industrial demand for fast and accurate numerical tools to estimate the hydrodynamic loads during e.g. green water events. In the last decade, several efforts have been made to overcome this problem ([3], [7] and [1]). These methods (WaveLand [3], KINEMA2 [1]) group the green water events into three different phases (Fig. 1):

1) Relative wave elevation and kinematics at bulwark locations
2) Green water shipping and propagation on deck
3) Slamming loads on vertical structure on deck

The first problem is solved by combining nonlinear wave kinematics and local steepness in the time domain with the actual linear ship motion and linear ship-induced wave diffraction in all above mentioned methods. In [7] the water propagation across the deck is modeled by a shallow-water approach based on- and extended from [8] using a Finite Differential scheme. This method gives very accurate description of the wave propagation but depends highly on the quality of the Finite Differential mesh used which makes this method not robust and simple enough for an engineering-type method. To overcome this lack of robustness [1] implement a new and even simpler analytical prediction, which are established based upon classical hydraulic theory ([9]). The approach is adjusted to take into account special effects such as dynamical wave input conditions and incident water velocities. This method is accurate and robust enough as a simplified tool for engineering design analysis purposes. It has its shortcomings to handle with complex geometry on the deck.

Methods with fully nonlinear descriptions of the hydrodynamics, such as BEM ([10]) and Reynolds-Averaged Navier-Stokes Equations (RANSE) - Volume Of Fluid (VOF) methods ([7]) are able to handle complex geometry but there are not fast and robust enough.

The model tests indicated that the incident free-surface particle velocity in critical waves may have the same magnitude as the velocity that may be released after 20m from a pure dam-breaking effect around the bulwark. This observation opens for the following simplified numerical models for simulations of the flow field on the deck:

- a 3D dam-break model, initially surrounding the forecastle deck of a ship
- a relative wave, with dynamic height and velocity surrounding the forecastle deck of a ship as boundary conditions

However, it was shown [3] that the incident water velocity is seen to be an important parameter in addition to the incident amplitude and therefore the 3D dam-break may be a too coarse assumption. This kind of simulation can be accomplished by the Smoothed-particle hydrodynamics (SPH) method in a very effective, robust and accurate way [11].

In this paper a development version of SCORE is used. SCORE is a SPH-simulator developed by Score Development Team in SINTEF.

Absorbing inlet/outlet boundary conditions for 2D SPH turbulent free-surface flows

Omar MAHMOOD, Damien VIOLEAU and Christophe KASSIOTIS
Laboratoire d'Hydraulique Saint-Venant
Joint research team EDF / Ecole des Ponts / CETMEF
Chatou, France
damien.violeau@edf.fr

Benedict D. ROGERS
School of MACE
Modelling and Simulation Centre (MaSC)
The University of Manchester
Manchester, UK

Martin FERRAND
MFEE department
EDF R&D
Chatou, France

***Abstract*—Our purpose is to provide SPH with a technique allowing absorbing inlet/outlet boundary conditions (BC), i.e. avoiding numerical wave reflection at open boundaries. We base our model on the inviscid 2–D Navier–Stokes equations, written in conservative form as propagation equations for appropriate Riemann invariants. We then use the latter to correct the boundary conditions in such a way that the proper velocity and density are prescribed. We use the SPH renormalized discrete interpolation to approximate the invariants coming from the interior of the fluid domain, while the invariants coming from the outside are simply prescribed by the user. This process is similar as in Lastiwka et al. [3], but extended to 2D conditions applied to laminar or turbulent flow (here with a *k*–ε turbulent closure and consistent wall BC as presented by Ferrand et al., 2010). We propose two validations: a laminar steady flow in a closed pipe and a turbulent steady flow in an open channel.**

I. INTRODUCTION

The question of inflow/outflow boundary conditions is SPH is crucial when one wants to simulate a local flow with open boundaries. Most authors use a crude technique where the velocity of particles is simply prescribed according to the desired distribution, the water depth being regulated through the pressure profile (see e.g. [1] and [2] for open-channel flows). However, this technique is known to yield spurious waves which can alter the computed fields or even result in a blowing up. Recently, a more sophisticated approach was proposed by Lastiwka et al. (see [3] and [4]) based on Riemann invariants for 1-D flows, avoiding artificial waves. The fields prescribed near the open boundaries are then a combination of velocity and density (the so-called Riemann invariants) in agreement with the physical information travelling across the boundaries. This 'absorbing boundary conditions' technique was successfully applied in [3] to laminar flows around rigid bodies. Here we extend this method to 2D turbulent flows combined with recent improvements in solid boundary conditions [6] and apply it to a closed pipe and a free-surface channel.

II. THEORY

A. Governing equations

We consider a turbulent weakly compressible free-surface flow. The velocity vector, density, pressure, turbulent kinetic energy and energy dissipation rate are denoted by $\mathbf{u}$, ρ, p, k and ε, respectively. Velocities and pressure are Reynolds-averaged, and the effects of turbulent fluctuations are modelled through the concept of eddy viscosity μ_T, estimated from the k–ε model [5].

The Lagrangian form of the Reynolds-averaged Navier–Stokes (RANS) and k–ε equations read

$$\begin{aligned}
\frac{d\rho}{dt} &= -\rho \operatorname{div}\mathbf{u} \\
\frac{d\mathbf{u}}{dt} &= -\frac{1}{\rho}\mathbf{grad}\,\tilde{p} + \frac{1}{\rho}div\left(\mu_m \mathbf{grad}\,\mathbf{u}\right) + \mathbf{g} \\
\frac{dk}{dt} &= P - \varepsilon + \frac{1}{\rho}div\left(\mu_k \mathbf{grad}\,k\right) \\
\frac{d\varepsilon}{dt} &= \frac{\varepsilon}{k}\left(C_{\varepsilon 1}P - C_{\varepsilon 2}\varepsilon\right) + \frac{1}{\rho}div\left(\mu_\varepsilon \mathbf{grad}\,\varepsilon\right)
\end{aligned} \tag{1}$$

where g is the gravitational acceleration. In these equations, the modified pressure $\tilde{p}$ and production of turbulent energy P are given by

$$\begin{aligned}
\tilde{p} &= p + \frac{2}{3}\rho k \qquad & p &= \frac{\rho_0 c_0^2}{\xi}\left[\left(\frac{\rho}{\rho_0}\right)^{\xi} - 1\right] \\
P &= \frac{2\mu_T}{\rho}\mathbf{S}:\mathbf{S} \qquad & \mathbf{S} &= \frac{1}{2}\left[\mathbf{grad}\,\mathbf{u} + \left(\mathbf{grad}\,\mathbf{u}\right)^T\right]
\end{aligned} \tag{2}$$

c_0 being the speed of sound at the reference density ρ_0, ξ = 7, and S the (mean) rate-of-strain tensor. Lastly, the dynamic viscosities are given by

On the boundary condition enforcement in SPH methods.

L.M. González
Naval Architecture Department (ETSIN)
Technical University of Madrid (UPM)
Madrid, Spain
leo.gonzalez@upm.es

J.L. Cercós
Naval Architecture Department (ETSIN)
Technical University of Madrid (UPM)
Madrid, Spain

F. Macià
Naval Architecture Department (ETSIN)
Technical University of Madrid (UPM)
Madrid, Spain

***Abstract*—The implementation of boundary conditions is one of the points where Smoothed Particle Hydrodynamics SPH methodology still has room to improve. In the SPH framework, different methodologies have been used to impose these boundary conditions. Namely, by using boundary forces-type models [1], [2]; by modifying the kernel structure in the boundaries neighborhood [3]; by creating virtual particles inside the solid boundary domain through mirroring techniques [4] and by adding normalized boundary integrals [5]. The latter being the main focus of the present work. In this paper a consistent implementation of classical boundary conditions in the presence of typical discretized and smoothed differential operators is analyzed. The calculation of gradients, divergences and Laplacians are important steps that must be consistently performed in explicit SPH formulations such as weakly compressible SPH (WCSPH). When the divergence free condition is enforced in the incompressible version of the SPH equations (ISPH), the solution of the Poisson equation plays a crucial role in pressure field calculations. This analysis studies the convergence of typical problems where differential operators are involved and the discretization SPH parameters tend to the continuum. In this paper we show: first, the lack of consistency of the SPH solutions if boundary conditions are not appropriately implemented, and consequently, the inconsistency of the calculated pressure field in the ISPH formulation. Second, it is also shown that the inclusion of the boundary integrals is an effective way of satisfying boundary conditions and obtaining a consistent formulation of the SPH methodology without using ghost or extra particles. As a continuation of the work presented in [6], rows of fixed dummy particles were used to impose the no slip boundary condition and severe inconsistencies near the boundaries were found when a second-order differential operator was applied. Nonetheless, when the boundary integral methodology is applied, the expected analytical result is recovered when both h and dx/h simultaneously tend to zero.**

I. Introduction

The SPH scheme is a Lagrangian model based on a smoothing of the spatial differential operators of the fluid-dynamics equations and on their subsequent discretization through a finite number of fluid particles. The smoothing procedure is performed at the continuum level using a compact support kernel function whose characteristic length is the smoothing length h. The resolution of the discrete SPH scheme is a function of the smoothing length h and the mean particle distance dx. In this framework, the (continuous) equations of fluid-dynamics should be recovered as both h and dx/h simultaneously tend to zero [7]. The SPH simulations in engineering usually involve solid boundary conditions (BC) for both the velocity and pressure fields. In the SPH framework, these conditions have been implemented in the past in a different number of ways: by using boundary forces-type models [1]; by modifying the structure of the kernel in the neighborhood of the boundaries [3]; by creating virtual particles inside the solid boundary domain through mirroring techniques; by renormalizing the boundary terms that explicitly appear in the integral SPH formulation [5], [17]. This latter approach is the main focus of the present work. In an incompressible fluid, density is not altered by pressure changes and the only local condition to obtain mass conservation is to have a divergence free velocity field. When incompressible fluids are simulated, different formulations depending on how the density and the incompressibility constrain are treated, can be found in SPH literature. On one hand, the WCSPH uses an explicit formulation where the pressure gradient together with the divergence and Laplacian of the velocity field are calculated applying discrete and smoothed versions of these operators to a discretized field. Physically, the pressure plays a thermodynamic derived role through the state equation. An interesting discussion about the influence of the truncation of differential operators close to the boundaries can be found in [8].

On the other hand, in the ISPH [9] and the MPS [10] approximations, the zero divergence condition is enforced solving a Poisson equation for the pressure field in every time increment. Here, the pressure acts as a Lagrange multiplier to satisfy the incompressible hypothesis. In this incompressible formulation, pressure moves as a wave that propagates with an infinite sound speed. Although no real fluid is completely incompressible, it is generally accepted that the incompressible assumption is a good approximation when the Mach number $M = v/c < 0.3$ where v is the characteristic fluid speed and c is the speed of sound. Many techniques presented in literature [11]–[14] solving the incompressible Navier-Stokes system of mass and momentum conservation are based on projection methods. This technique is often referenced in literature as 'fractional step', 'semi-implicit' or 'pressure-Poisson equation' method. The projection method decouples

A diffusion based shifting algorithm for incompressible smoothed particle hydrodynamics: Validation with cases involving slamming bodies and cylinder exit.

Alex Skillen, Benedict D. Rogers, Peter K. Stansby
The Modelling and Simulation Centre,
School of MACE,
The University of Manchester, UK

Steven Lind
School of Computing, Mathematics, and Digital Technology,
Manchester Metropolitan University,
Manchester, UK.

***Abstract*—When simulating an incompressible fluid, traditional weakly compressible SPH techniques have disadvantages including noisy pressure predictions and high computational costs. As an alternative, incompressible SPH simulations can be achieved by employing a projection method to enforce a divergence free velocity field. Such techniques are receiving considerable interest in the literature due to the potential for noise free pressure predictions, with acceptable computational costs. The method does however suffer from stability issues when particles bunch together or are stretched apart from one another, which is generally the case if particles are restricted to following streamlines, as is dictated by the Navier-Stokes equations. To overcome this issue, particles can be shifted away from one another, across streamlines, according to some algorithm whose job it is to determine a suitable displacement vector. The primitive variables are then corrected to account for their change in position via interpolation.**

Here, we follow on from earlier work in the authors' group and employ Ficks's law to shift the particles. We present validation cases involving cylinder and wedge slamming into initially calm water. We also consider cylinder exit from initially calm water. These cases are particularly relevant to the design of ships and wave energy extraction devices. The results we present show effectively noise free pressures and very high accuracy. To our knowledge, this is the first time such accuracy has been achieved via SPH techniques for such complex cases.

I. Introduction

SPH is a particularly attractive tool in simulating flows involving free-surface deformation due to the Lagrangian nature of the discretisation. Free surface tracking is not required and hence arbitrarily complex free-surface deformations can be handled with relative ease. For this reason, SPH has received considerable attention in simulating complex hydrodynamic problems. In virtually all such cases, the working fluid (i.e. water) can be considered as incompressible without loss of rigour.

Traditional methods of simulating incompressible fluid flow via SPH have tended to use weakly compressible SPH formulations (WCSPH). In the weakly compressible approach, the pressure is treated as a thermodynamic variable and is set via an artificial equation of state. The sound speed is set to be sufficiently high so as to limit density variations to within a small fraction of the actual fluid density (i.e. by reducing the Mach number). In practice, this high sound speed places a severe limitation on the maximum permissible time-step size via the CFL constraint. In addition, for a WCSPH simulation, sound waves tend to reflect from solid surfaces, and hence it is generally necessary to place tank walls far away from areas of interest so as to not introduce spurious interference (this is particularly true given the high sound speed). The combination of a large domain size and small time step can impose a severe penalty in terms of computational costs. The predicted pressure field in a WCSPH simulation also tends to be noisy since a small perturbation in the local density will yield a large variation in the local pressure.

To overcome these limitations of a WCSPH formulation, a truly incompressible approach can be adopted. Chorin [1] describes a projection method for the numerical solution of the incompressible Navier-Stokes equations. The method decouples the momentum and continuity equations and is used to enforce a divergence free velocity field, thereby satisfying the continuity condition for an incompressible fluid. Cummings and Rudman [2] apply the Chorin projection method to SPH. The method initially solves the discritised Navier-Stokes momentum equations with a null pressure gradient in order to generate an intermediate velocity field. A Poisson equation for the pressure is then derived such that the divergence of the intermediate velocity field is balanced by the divergence of the pressure gradient term. The final velocity field at the end of the time step is therefore divergence free. Cummings and Rudman demonstrate their method for internal test-cases involving vortex spin-down and Rayleigh-Taylor instability. It is found that satisfactory agreement with finite difference simulations can be attained. They do however note error accumulation in the density field, becoming manifest via particle clumping and/or stretching. This anisotropic particle distribution tends to lead to instability in the algorithm, often prohibiting a solution from being obtained at all [3].

Shao and Lo [4] propose an alternative formulation in

SPH multi-fluid model with interface stabilization based on a quasi-buoyancy correction

A.C.H. Kruisbrink, F.R. Pearce, T. Yue, K.A. Cliffe and H.P. Morvan
University of Nottingham
University Park
Nottingham
arno.kruisbrink@nottingham.ac.uk

Abstract—**In this paper a concept is introduced for the stabilization of the interface between two fluids. It is shown that the change in pressure gradient across the interface between two fluids leads to a force imbalance, which is attributed to the SPH particle approximation. To stabilize the interface a pressure gradient correction is proposed. For this purpose the pressure gradient is related to (gravitational and fluid) accelerations. This leads to a quasi-buoyancy correction for stagnant flows, which is extended to accelerating flows. The approach results in a simple density correction, where no parameters or coefficients are involved, which is included as extra term in the SPH momentum equation.**

The concept for the stabilization of the interface is explored in three case studies. The first case is the stagnant flow in a tank. Simulation shows that the interface remains stable up to density ratios of 1000 (typically for water and air). The second case is the Rayleigh Taylor instability, where a reasonable agreement with a level-set model is achieved. The third case is an air flow across a water surface up to density ratios of 100, artificial wave speeds of water higher than that of air, and high air velocities.

It is demonstrated that the quasi-buoyancy correction may be used to stabilize the interface up to high density ratios and more realistic wave speed ratios.

I. Introduction

Many fluid flows involve more than one fluid. In the case of liquid-gas flows the density ratio may be very high. For water-air flow under atmospheric conditions the density ratio is about 800.

Several multi-fluid models are available in the literature for incompressible fluids (e.g. Cummins and Rudman, 1999; Hu and Adams, 2007 and 2008; Xu et al., 2008) as well as for weakly compressible fluids. This paper focuses on weakly compressible SPH only. The most important multi-fluid models in literature and their main features are described below.

Ritchie and Thomas (2001) suggest a summation of the particle averaged pressure (not density), to deal with large density gradients. However, the method is based on specific energy and does not satisfy mass conservation.

Colagrossi and Landrini (2003) used a density renormalization at intermediate time steps, a large artificial surface tension and high wave speed for the low dense fluid, and a smoothing of the velocity field. Apart from these unphysical aspects, very small time steps are required.

Flebbe er al. (1994) introduced the "particle number density" in astrophysics. Ott and Schnetter (2003) and Hu and Adams (2006) applied it to estimate the density for multi-fluids. Colagrossi et al (2008) applied it to develop an equation of motion for multi-fluids. The main restriction of this method is that the particle volume (V) and the change of volume ($\Delta V/V$) due to compression must remain the same for different fluids. The latter means that the choice of the wave speed ratio is rather limited, usually resulting in a higher wave speed for the low dense fluid. For water and air, with a density ratio of 1000, a wave speed ratio of 1/14 would be required. Another restriction is that the method, usually based on the standard SPH summation density, cannot be applied to free surface flows.

Grenier et al. (2009) use a density renormalization with a variant of the Sheppard correction, in which the particle volume is obtained from the continuity equation. In addition they use a repulsive force between particles of different fluids, similar to that suggested by Monaghan (2000), to stabilize the interface.

Monaghan (2011) uses the continuity equation in a form suited to multi-fluid flows. He also uses a repulsive force to stabilize the interface between two fluids, similar to that used by Grenier et al. (2009). The concept is applied to high density ratios up to 1000. However, the wave speed of the low dense fluid (read: gas) is still a factor 5 to 7 higher than that of the high dense fluid (read: liquid).

The application of the above multi-fluid models for weakly compressible SPH is limited by density ratios and/or wave speed ratios. None of these models deal with the particle instability, due to the change of the pressure gradient across the interface, except Grenier et al. (2009) and Monaghan (2011). In this paper we aim for water-air applications with density ratios up to about 1000, and more physically realistic wave speed ratios. For this purpose a novel model based on a pressure gradient correction is introduced.

A Multiphase Incompressible-Compressible Smoothed Particle Hydrodynamics Method

S. J. Lind
School of Computing, Mathematics, and Digital Technology
Manchester Metropolitan University
Manchester, UK
s.lind@mmu.ac.uk

P. K. Stansby, B. D. Rogers
Modelling and Simulation Centre
School of Mechanical, Aerospace, and Civil Engineering
University of Manchester
Manchester, UK
p.k.stansby@manchester.ac.uk
benedict.rogers@manchester.ac.uk

***Abstract*—The projection-based incompressible smoothed particle hydrodynamics (ISPH) method has been shown to be accurate and stable for a range of internal and free-surface flows [1], [2]. Although single-phase ISPH provides valuable predictions, many of the flows of greatest interest to engineers, such as wave breaking and wave impact problems, are inherently multiphase. It is well known that the air phase can play a significant role in such problems, either through its entrainment in the water or through its role as an air cushion that dissipates wave impact pressures [3]. While water can be accurately modelled as incompressible, the air phase must be modelled as compressible for a complete description of the physics. With this in mind, a novel multiphase SPH method has been developed to model two-phase fluid systems where one phase is strictly incompressible, and the other phase is compressible. The ISPH and standard compressible SPH formulations are employed, and are fully coupled in a physically and mathematically consistent manner. The method permits physically realistic density ratios (1000:1) and, unlike many multiphase SPH methods, uses values of the speed of sound appropriate to each phase. The method is validated against known analytical solutions for multiphase flows, such as standing gravity waves, before being applied to some popular test cases and the problems of interest. To the authors' knowledge, this is the first multiphase SPH method that couples both the incompressible and compressible formulations. It is an important step in the development of a modelling tool capable of completely describing wave breaking, impact and slamming phenomena.**

I. Introduction

Many of the flows of greatest interest to civil and coastal engineering, such as wave breaking and impact problems, are inherently multiphase. In the vast majority of cases, the density and viscosity of air is such that its influence on the water phase is assumed negligible in comparison, and the problem is treated wholly as a free-surface flow. In many cases this assumption is justified and can provide very accurate descriptions of flows which, in actuality, are multiphase [4]. However, there are also a substantial number of situations where one cannot neglect the air phase [3], [5]. For wave breaking, wave impact and slamming problems, the air phase can play a significant role in the dynamics - either through its entrainment as small air bubbles (millimetre length scale), or through the entrapment of large air pockets (wave-height length scale). Undeniably, to obtain a complete description of such problems, the air phase needs to be retained and modelled. To a high degree of accuracy, as the fluid velocities of the problems in question tend not to exceed the speed of sound in water, the water phase is incompressible. In the air phase however, where the speed of sound is an order of magnitude less than in water, the Mach number for these flows is not negligible and compressible effects can be important. It is then necessary that, while the water phase can be treated as incompressible, the air phase be modelled as compressible.

In this paper a novel multiphase SPH method (referred to herein as ICSPH) is presented which can be used to model such incompressible-compressible multiphase flows. There are, of course, a multitude of applications for such a method (such as cavitation erosion and spraying/atomization modelling in medicine and industry), but, in this paper, the focus will be on the coastal engineering processes of interest to the authors. In the water phase, the incompressible Navier-Stokes equations are solved using the incompressible ISPH method of Lind et al. [2]. This method follows on from the work of Xu et al. [1], in which a shifting algorithm was employed which maintained regular particle distributions at each time step, preventing instability. The method of Lind et al. [2] is almost identical but for the employment of a more general shifting approach based on Fick's law of diffusion. Results have shown that incompressible SPH, with shifting, produces very accurate, almost noise-free, predictions for both pressures and velocities for a range of internal and free-surface flows. This method is strictly incompressible in the sense that a divergence-free velocity field is maintained at each time step through a projection method. Therefore it cannot be employed in the air phase where compressible effects may be important. In the air phase, the conventional weakly compressible SPH method is employed, which solves the compressible governing equations, given some functional (thermodynamic) relationship between density and pressure. These are two quite distinct numerical approaches which also have to be (fully) coupled in a way that is mathematically

SPH for two-phase fluid flow including cavitation

P. Groenenboom
ESI Group Netherlands
Radex Innovation Centre, room 4.57
Rotterdamseweg 183 C, 2629 HD, Delft, The Netherlands
Email: pgr@esi-group.com

***Abstract*—The SPH module of PAM-CRASH has been extended by a model for a mixture of liquid and gas/vapour. For this two-phase model it is assumed that both components may co-exist in individual particles and that they are advected according to a single velocity field. It is furthermore assumed that the liquid in the mixture is incompressible and that the gas/vapour is an ideal gas at adiabatic conditions. Numerical results for wedge impact on aerated water will be discussed. Evaporation and condensation phenomena according to a cavitation model [1] have also been implemented. For some simple test cases the well-known phenomenon of very high pressure spikes due to the collapse of cavitation bubbles has been observed. Preliminary results for ditching of the NACA-2929J fuselage model, including cavitation, will be discussed.**

I. INTRODUTION

Flow of a mixture of two or more types of fluids is of great relevance to many industrial applications and natural phenomena. Many of the commercially available CFD codes are able to simulate flow of multiple species (miscible or immiscible) or multiple phases, but may experience problems in the case of fluid interfaces having large deformations such as may occur for violent free surface motion or fluid-structure interaction. The Smoothed Particle Hydrodynamics (SPH) method is well-suited to include such interfaces, but most formulations only allow for a single-phase solution. In this paper, we consider the extension of SPH to handle multi-phase flow. With an appropriate choice for the SPH solution of the conservation equations for mass and momentum, SPH is can solve flow of immiscible species, i.e. as in PAM-CRASH/SPH [2]. For miscible fluids or multi-phase mixtures it is usually more appropriate to consider a mixture to be present in the discretized volumes (i.e. in individual particles). Obvious examples for such a description are wave propagation in aerated water and ditching of aircraft. In both cases, it is not feasible to perform simulations using particles with a size of the (microscopic) bubbles of vapour (or air) in the water. For this reason, a two-phase flow model has been implemented in the SPH solver of PAM-CRASH. When the pressure drops below the vapour pressure, cavitation will be initiated and the density of the mixture will reduce. In case cavitation bubbles collapse, high pressure spikes are generated. The importance of these counteracting effects on ditching has been recognized, but experiments are difficult to perform and until recently numerical simulation was not possible. Numerical simulations and sub-scale testing of ditching has been conducted [3], but thus far without using two-phase models. Below a description of the two-phase model including cavitation for SPH will be presented. Some validation tests and a preliminary ditching application will be discussed.

II. NUMERICAL MODEL

A. Basic Mixture Model

For numerical simulation of the flow of a mixture several aspects need to be accounted for. Firstly, the mass of each species has to be conserved (and, as a consequence, the total mass). Secondly, it has to be decided whether the mixture should be transported according to individual velocity fields or by a single one. Since for SPH it is assumed that particles will be displaced according to a unique velocity field it is logical to assume that for a mixture (defined by volume fraction within each particle) the species move with the same velocity (no-slip condition). The alternative of using separate velocities for each species may be covered by the exiting option with SPH for different material to be treated by disjunct particles within a single simulation. Finally, the conditions at which the various species co-exist must be defined.

In the following we will consider only two species (gas and liquid), without diffusion or chemical reactions, with a single velocity field. We will assume either one phase (liquid) to be incompressible or pressure-equilibrium between the two materials in a single volume. It will be assumed that the gas is adiabatic (or isentropic), there is no need to consider internal energies to obtain a solution.

According to the no-slip model and neglecting diffusion, conservation of gas and liquid is defined by:

$$\frac{\partial \hat{\rho}_g}{\partial t} + div(\hat{\rho}_g \vec{u}) = 0 \qquad (1)$$

And,

$$\frac{\partial \hat{\rho}_l}{\partial t} + div(\hat{\rho}_l \vec{u}) = 0 \qquad (2)$$

With the following definitions for a two-phase mixture,

$$\hat{\rho}_g = \alpha.\rho_g \ , \ \hat{\rho}_l = (1-\alpha).\rho_l \ , \text{ and}$$

$$\rho = \hat{\rho}_g + \hat{\rho}_l \qquad (3)$$

A Consistent Particle Method for Simulation of Multiphase Flows with High Density Ratios

Abbas Khayyer
Dept. of Civil and Earth Resources Eng.
Kyoto University
Kyoto, Japan
khayyer@particle.kuciv.kyoto-u.ac.jp

Hitoshi Gotoh
Dept. of Civil and Earth Resources Eng.
Kyoto University
Kyoto, Japan
gotoh@particle.kuciv.kyoto-u.ac.jp

***Abstract*—A novel scheme is presented for evaluation of density at a phase interface in particle-based simulations of multiphase flows characterized by high density ratios. The scheme is founded on a Taylor series-based approach and has been shown to be efficient in providing mathematically consistent and physically sound results. The effectiveness of the newly proposed scheme is verified through the simulation of a bubble rise in water, oscillating concentric elliptical regions and a violent sloshing flow.**

I. INTRODUTION

Multiphase flows are ubiquitous in a wide range of engineering and industrial processes at different length scales and flow regimes. Some typical applications of multiphase flows include air-water flows such as violent aerated wave impacts and sloshing flows, gas-liquid flows in pipelines, sediment transport, thermal-hydraulic design of nuclear reactors, etc. Multiphase flows are also well-known to manifest themselves as unsteady processes characterized by inherently complicated physics. Hence, numerical simulation of multiphase flows has been one of the most challenging issues in Computational Fluid Dynamics. The challenges in multiphase flow simulations include: large/abrupt density/viscosity drop across the phase interface, presence of moving and highly deforming boundaries, topological transitions such as merging of interfaces, etc.

In the context of particle methods, there have been several attempts to propose stable/accurate multiphase methods. However, in most cases, researchers have used numerical stabilizers (such as artificial viscosity or unphysical surface tension terms, e.g. [1]) and/or reformulated the original schemes by considering the particle volume rather than its mass (e.g. [2, 3]). This consideration results in smoothening of density field at the phase interface and despite being helpful in dealing with the mathematical discontinuity of density, it will result in an unphysical density diffusion. Recently Monaghan and Rafiee [4] proposed a robust SPH algorithm based on the Lagrangian equations and successfully simulated several multiphase flows with high density ratios without consideration of particle volume instead of mass or use of an unphysical surface tension term. Nevertheless, their simulations were performed by using a repulsive pressure force between particles of different fluids as well as an artificial viscosity term.

This paper aims at introducing a consistent particle method for simulation of multiphase flows characterized by high density ratios. The developed method is an enhanced version of the Moving Particle Semi-implicit (MPS [5]) method benefiting from four previously developed schemes [6,7,8] as well as a new one, proposed for accurate modeling of density at the phase interface. A Taylor series-based approach is considered for an accurate, consistent modeling of density at the phase interface. The performance of the developed method is verified through the simulation of a bubble rise in water [3], oscillating concentric elliptical regions [4] and a violent sloshing flow [9].

II. ACCURATE MODELING OF DENSITY AT THE PHASE INTERFACE

One of the challenging issues in simulation of multiphase flows, particularly those characterized by high density ratios, corresponds to the mathematical discontinuity of density at the phase interface. The simplest approach to deal with this discontinuity is to evaluate the calculated density at a target particle i based on a simple spatial averaging by considering the distribution of the mass of neighbouring particles.

$$\rho_i = \sum_{i \neq j} m_j W_{ij} \tag{1}$$

As mentioned by several researchers, e.g. [3], application of Eq. (1) will simply result in a numerical diffusion and accordingly an unphysical smoothening of density. As a result, the sharp variation of density at the interface cannot be represented correctly. Therefore, the key issue in simulation of a multiphase flow is to model the density at the phase interface in a mathematically consistent, physically sound and computationally efficient manner. Here, a Taylor series-based approach is applied for an accurate, consistent modeling of density at the phase interfaces.

Mathematically, a relation between the calculated densities at neighbouring particles j and target particle i can

Development of SPH variable resolution using dynamic particle coalescing and splitting

R. Vacondio
Department of Civil Engineering,
Parma University
Viale G.P. Usberti 181/A,
43100, Parma, Italy
renato.vacondio@unipr.it

B.D. Rogers[1], P.K. Stansby[1], P. Mignosa[2] & J. Feldman
[1]School of Mech., Aero. & Civil Engineering
University of Manchester
Manchester, United Kingdom
[2]Department of Civil Engineering,
Parma University
Viale G.P. Usberti 181/A,
43100, Parma, Italy

***Abstract*—In this paper a novel variable resolution method using particle splitting and coalescing for the SPH numerical schemes of the Navier-Stokes equations is presented. The key idea of the scheme is to dynamically modify the particle sizes by means of splitting and coalescing individual particles. The SPH scheme adopted is variationally derived and this guaratees that both mass and momentum are conserved including particles with different smoothing lengths. To prevent highly anisotropic distributions of the particles, a generalized shifting procedure which can address also domains discretized with variable mass particles is included. The algorithm has been tested against Poiseuille flow showing that the error introduced by the splitting and coalescing is negligible. The capability of the numerical scheme for increasing efficiency is also shown: the SPHERIC test case of the moving square in a box has shown that the particle refinement procedure is able to increase the efficiency while maintaining the same level of accuracy, as a uniform distribution with the most refined resolution.**

I. Introduction

In classical Eulerian computational models, adaptive structured [18] or unstructured grids [9] have been successfully used to obtain variable resolution and to simulate multiscale flows while retaining computational efficiency. In meshfree numerical schemes there have been some early attempts to introduce variable resolution by either remeshing, and particle insertion/removal techniques [7], [19], [20], [22]. Recently dynamic particle refinement which conserves mass and momentum has been applied to Shallow Water Equations [23], [24], [26]. This has been obtained by particle splitting and coalescing procedures which can respectively increase and decrease the spatial resolution and this has made the simulations of real flooding test cases possible [23]–[25].

The same dynamic particle refinement algorithm is applied in this work to Navier-Stokes equations. Since this procedure generates particles with different sizes, a consistent SPH discretization scheme which can accurately discretize the Navier-Stokes equations in presence of variable smoothing length is needed. Bonet and Rodríguez-Paz [1] proposed a momentum-conservative weakly compressible formulation which takes into account variable smoothing length. However, this is neither accurate in the presence of a free surface nor computationally efficient due to multiple sub-iterations required.

A novel, variationally consistent and efficient SPH formulation is herein derived to address both of these issues. This scheme assures momentum conservation also in presence of particles with different smoothing length, and moreover it addresses the efficiency and accuracy problem highlighted in [1].

Some additional improvement to the formulation is also introduced to increase the accuracy of the scheme. In the framework of projection-based incompressible SPH schemes Xu et al. [27] proposed a particle shifting algoritm which is able to prevent instabilities due to highly disordered particle distributions. In this work the algoritm is generalized for particles with different masses.

This paper is organized as follows: in Section II the derivation of the particle splitting and coalescing algorithm is presented. In Section III the variationally consistent SPH discretizations of Navier-Stokes equations is briefly reported and the particle shifting algoritm is also described, whereas the complete derivation of the formulation is presented in Appendix A. In Section IV the numerical scheme for Navier-Stokes equations is tested against analytical and reference solutions.

II. Methodology

A. Particle splitting

To increase the resolution in certain areas of the domain one particle is split into M daughter particles. The mass m_k, position x_k, velocity vector $\mathbf{v}_k$, and the smoothing length h_k for any of the $k = 1...M$ refined particles has to be defined, therefore the total number of degrees of freedom is 6 for each k-th daughter particle. To reduce the degrees of freedom of the problem, the number of new particles M and their relative positions are given by using a fixed refinement pattern which defines the relative position of daughter particles and their masses. The hexagonal refinement pattern plotted in Figure 1 is adopted in this work since it is a good balance between

Development of the finite volume particle method for internal flow with rigid body dynamics

Nathan J. Quinlan, Libor Lobovský, Mihai Basa, Rúairí M. Nestor
Mechanical and Biomedical Engineering
National University of Ireland Galway
nathan.quinlan@nuigalway.ie

***Abstract*—Particle methods offer an interesting approach for simulation of flow around moving bodies in confined ducts, where conventional methods would suffer extreme mesh distortion. However, it is difficult to maintain a reliable distribution of particles while internal flow passages rapidly change in cross-sectional area, or open and close completely, as in mechanical heart valve. We present some developments of FVPM to address this problem. A simple fluid-rigid-body interaction algorithm is described and validated for flow over an elastically mounted cylinder with transverse vortex-induced vibration. A hybrid Lagrangian-ALE particle motion scheme is proposed, with ALE particles distributed over the moving body and moving with it, and Lagrangian particles flowing through most of the domain. This has the advantage of ensuring particle cover near the body at all times, and also gives an opportunity for refined particle distribution around the body. Some body-following particles may pass completely or partially through the domain boundary. Some special treatment is required for particles that have very small volume remaining inside the domain to prevent numerical error from dominating. The hybrid particle motion approach is demonstrated for flow around a body with prescribed motion impinging on a wall, and for an idealised two-dimensional mechanical heart valve.**

I. Introduction

In this paper, we report progress towards the modelling of flow in devices such as mechanical heart valves, which involve rigid bodies moving near or impacting on duct walls. These siutations are prone to the development of unfavourable particle distribution. To gain a measure of control of particle distributon in critical regions, a combination of Largangian and arbitrary Lagrangian-Eulerian (ALE) techniques is proposed.

A mechanical heart valve (MHV) consists of one or more occluders which prevent unwanted blood flow by responding passively to pressure gradient. The capability to model flow through these devices can guide design to enhance bulk hydrodynamic characteristics (the pressure drop-flow rate relationship) and device longevity, while minimising blood damage caused by abnormal magnitude and duration of mechanical loading on blood cells. Modelling by conventional mesh-based approaches is difficult, however, because of the large geometric and topological changes due to motion of the occluder. Particle methods avoid the need for mesh deformation and/or remeshing. An additional benefit of particle methods for haemodynamic applications is the possibility to track histories of particles representing individual blood cells.

In the present work we describe progress towards modelling a MHV using the finite volume particle method (FVPM). In particular, this paper is concerned with the interaction between the fluid and the rigid body or bodies that make up the valve occluder. We begin by briefly reviewing the finite volume particle method. We then describe the implementation of couple rigid-body dynamics in FVPM and present validation for vortex-induced vibration of a cylinder. In section IV, the concept of body-following particles is introduced and demonstrated for a simple problem with prescribed body motion. Finally, this approach is applied to modelling of an idealised mechanical heart valve in section V.

II. The finite volume particle method

The finite volume particle method (FVPM) created by Hietel *et al.* [1] may be understood as a generalisation of the mesh-based finite volume method, in which the finite volumes are allowed to overlap. The finite volumes are advected with an arbitrary Lagrangian-Eulerian velocity field and referred to as particles. An infintesimal volume element is considered to be distributed among the particles that cover it according to the relative magnitude of the kernel-like weight functions $W_i(\mathbf{x})$ of the particles i. Following this logic, particle volume can be defined as

$$V_i = \int_\Omega \frac{W_i(\mathbf{x})}{\sum_j W_j(\mathbf{x})} \mathrm{d}\mathbf{x} \,, \tag{1}$$

where i and j are particle indices, $\mathbf{x}$ is position and Ω is the spatial domain. The weight functions $W_i(\mathbf{x})$ and $W_j(\mathbf{x})$ are compactly supported. Therefore, the only non-zero terms in the sum come from particles j which overlap with the support of i. This FVPM definition of neighbourhood differs from the SPH version. Domain boundaries truncate the support of particles, and must be accounted for in the calculation of volume.

The partition of unity in volume leads to the derivation of a particle interaction vector which is precisely analogous to the interface area between finite volume mesh cells. It is defined as $\boldsymbol{\beta}_{ij} = \boldsymbol{\gamma}_{ij} - \boldsymbol{\gamma}_{ji}$, where

$$\boldsymbol{\gamma}_{ij} = \int_\Omega \frac{W_i(\mathbf{x}) \nabla W_j(\mathbf{x})}{\left(\sum_k W_k(\mathbf{x})\right)^2} \mathrm{d}\mathbf{x} \,. \tag{2}$$

The details of FVPM are derived by replacing cell interface area in the finite volume method with $\boldsymbol{\beta}_{ij}$. The resulting semi-

Third-generation RSPH in 3D

Børve, S.
Norwegian Defence Research Establishment
Kjeller, Norway
Steinar.Borve@ffi.no

***Abstract*—This paper is a progress report on the development of a three-dimensional version of third-generation RSPH. It focuses on challenges met when transforming the method from handling two to three dimensions. We will describe how template particle distributions utilized by the method is generated, and what the intrinsic discretization errors in using these template particles are. We show how the discretization errors increase in three dimensions compared to two dimensions. We discuss to some detail possible approaches to correcting these errors. Stability properties are also considered. In the simple case of a plane Sod shock, we show that corrections can give improved accuracy and prevent instabilities from occurring. The final section of this paper is devoted to describing briefly the current status of free surface handling in three-dimensional RSPH.**

I. Introduction

At SPHERIC 2010, a modified version of the Regularized SPH (RSPH) method was presented [1]. RSPH was originally developed as an extension to SPH [5], [6] with the aim of providing flexible methods for variable resolution in SPH. In contrast to standard SPH, RSPH is not restricted to resolution being a function of the initial particle distribution and the subsequent, time-dependent flow pattern, and RSPH is therefore able to maintain high resolution in regions of interest, for example near shock wave structures, and low resolution elsewhere [4]. However, the integration of these methods into existing SPH codes has previously not been straightforward. This comes from the fact that SPH summations give poor accuracy when the particle size and/or particle mass is varied substantially within the interaction range.

The third-generation RSPH method gives the same flexibility with regards to variable resolution as its predecessors but in addition, it also has a better integration with conventional SPH and can therefore be more easily integrated in existing SPH codes. In previous versions of RSPH, the smoothing length (h) profile was piecewise constant with steps in h representing a factor of 2 change in the h-value. This approach was well suited for constructing global h-profiles and corresponding particle distributions, but it made it impossible to achieve acceptable accuracy using SPH summations only. Although third-generation RSPH, as a first approximation, constructs global h-profiles in the same manner as previous RSPH versions, the final global h-profile is piecewise linear rather than piecewise constant. The generation of the corresponding particle distribution is made by using a set of template particle distributions, small building blocks of sub-distributions that can be put together (and reused) in a consistent manner.

The method presented in 2010 was restricted to two dimensions [1]. Although all principles of the method presented in two dimensions can be reused in three dimensions, the increased complexity of adding another dimension requires the method to be reinvestigated. This paper is a progress report on the development of a three-dimensional version of third-generation RSPH, and we will focus on the differences between the two-dimensional (2010) version and the three-dimensional (2012) version.

The generation and use of template cells is reviewed in section II. In section III, we discuss the discretization errors associated with the use of these template cells as indicated by a series of static tests with variable smoothing length. Section IV focuses on dynamical tests of a well known compressible problem, the Sod shock-tube test, both in planar and spherical symmetry [14], [19]. In [1], a substantial part of the paper was devoted to how free surfaces could be accurately handled within the framework of third-generation RSPH. In section V, we briefly review the current status of the work on extending the free-surface methods from two to three dimensions. Finally, a short conclusion is presented in section VI.

II. Generating particle distributions using template particles

The starting point for generating new particle distributions is a global, piecewise-constant h-profile where the local value of h is a binary number multiplied by a global minimum h-value. Assuming a fixed ratio of h to particle spacing, the grid upon which the h-profile is defined can be grouped into blocks which fit exactly M particles (in 1D). Using such a h-profile directly would result in a piecewise uniform particle distribution. The yellow curve and filled circles in Fig. 1 indicate how the particle size and particle positions, respectively, would in this case vary as one move across a one-dimensional step in h.

In third-generation RSPH, we attempt to modify the global h-profile so that any transition in h becomes piecewise linear rather than piecewise constant. The approach is here explained in the one-dimensional case. Let M, as before, be the number of particles which would fit uniformly into a h-profile grid cell if the local h-value is identical to the minimum h-value. The transition from a smaller particle size, Δ_0, to a factor 2 larger particle size, will take place over an interval equal to $3\Delta_0 M$, and the interval should fit exactly $2M$ particles. Since particle 0 and particle $2M-1$ have particle size equal to Δ_0

Development and validation of a SPH model using discrete surface elements at boundaries

Andrea Amicarelli, Giordano Agate, Roberto Guandalini
Environment and Sustainable Development Department -ASV-
RSE (Research on Energy Systems), Milan, Italy
Andrea.Amicarelli/Giordano.Agate/Roberto.Guandalini@rse-web.it

***Abstract*. A 3D SPH model has been developed, according to a particle approximation, which involves both volume and surface discrete elements at boundaries. The model relies on the principles of the 2D formulation of Ferrand et al. ([8] and [9]) and the spatial reconstruction schemes used in SPH-ALE (Arbitrary Lagrangian-Eulerian) modelling. Validations are performed on two test cases: a 2D water jet impact over a plate and a 3D dam break phenomenon. The results are compared with the reference solutions and measurements. Further inter-comparisons are provided by performing the same simulations with the semi-analytic approach (alternative SPH model).**

I. INTRODUCTION

Smoothed Particle Hydrodynamics (SPH) has been successfully developed to model free-surface and multi-phase flows, especially under fast and transitory regimes ([17], [19]). This mesh-less computational fluid dynamics technique has been recently applied even to represent complex and industrial phenomena (as in [12]). At the same time some SPH features are still deeply investigated, as they limit the accuracy or the effectiveness of the technique. They mainly concern the boundary treatment, the spatial accuracy and the computational effort.

In SPH modelling the particles are considered as moving computational nodes. The derivatives appearing in the governing equations of a computational particle (located at $\underline{x}_0$) are evaluated performing a weighted interpolation. This involves the particles ("neighbours") in a sphere of influence (kernel support V_h) around $\underline{x}_0$. The weight is represented by an analytical function, called kernel (W, considered hereafter as defined in [20]).

In particular the SPH boundary treatment concerns the role of the surface term in the integral SPH approximation ("I") of a derivative (of a generic function f):

$$\left\langle \frac{\partial f}{\partial x_i} \right\rangle_{I,\underline{x}_0} = \int_{A_h} f W n_i dx^2 - \int_{V_h} f \frac{\partial W}{\partial x_i} dx^3 \qquad (1)$$

A_h is the kernel support boundary, whose dimensions are related to the smoothing length scale h; $\underline{n}$ is the unity vector locally normal to the boundary.

In order to model the surface term of (1) in a proper way, several SPH techniques have been proposed. They are briefly synthesized in the following.

[10] and [18] formulated repulsive forces, to be introduced in the momentum equation, in order to ensure the non-penetration condition at wall. These forces depend on the relative distance of the computational fluid particle and the boundary. A similar approach has been developed by [21], using the so-called "boundary particles".

One of the most common approaches refers to the "ghost particles" ([4]). Each kernel support, which remains truncated by boundaries, is artificially filled up with virtual (ghost) particles in its truncated part. The position of these particles is derived by the distribution of the fluid particles inside the interior domain, according to symmetry considerations. The parameters of the ghost particles are then set, in order to impose the desired boundary conditions. A similar approach has been derived by [22], using "mirror particles", which are instead fixed and regularly distributed.

[24] and [5] formulated and validated the semi-analytic approach. Thanks to Green's theorem and the kernel properties, the surface term in (1) is replaced by a volume integral. This is partitioned in a sum of analytical integrals, all over the truncated part of the kernel support. This technique represents the integral version of the "ghost particle" technique and seems to provide more reliable results under generic boundary conditions.

SPH-ALE models ([6], [14], [15], [16]) probably provide the most accurate solution, in order to treat complex and moving boundaries. The differential governing equations are integrated over a finite volume (the kernel support itself), furnishing a weak (global) formulation. As resulting from this integration, SPH-ALE models automatically and explicitly treat the surface terms. Further the spatial accuracy is improved by the use of Godunov fluxes (in analogy to the Finite Volume technique) and Riemann solvers (up-flow schemes).

Recently [8] and [9] have developed a 2D model based on a direct modelling of the surface term in (1), solving analytical integrals over surface elements. Further, in order to improve the spatial accuracy of the SPH approximations ([1] and [2]), the governing equations are normalized by an integral version of Shepard's coefficient ([23]). In addition, a thin layer of fluid particles is integrated at the very boundary (in the inner domain), to strength the boundary term effects.

An Improved Consistent 3D Particle Method for Enhanced Wave Impact Calculations

Hitoshi Gotoh
Dept. of Civil and Earth Resources Eng.
Kyoto University
Kyoto, Japan
gotoh@particle.kuciv.kyoto-u.ac.jp

Abbas Khayyer
Dept. of Civil and Earth Resources Eng.
Kyoto University
Kyoto, Japan
khayyer@particle.kuciv.kyoto-u.ac.jp

***Abstract*—A 3D higher order Laplacian model is proposed for enhancement and stabilization of pressure calculation by a projection-based particle method, namely, the Moving Particle Semi-implicit (MPS) method. The Laplacian model is derived by meticulously taking the divergence of a commonly applied SPH gradient model and is then utilized for discretization of Laplacian of pressure corresponding to the Poisson Pressure Equation (PPE). The enhancing and stabilizing effect of the 3D higher order Laplacian model is shown through simulations of designed exponentially excited sinusoidal pressure oscillations and a schematic dam break with an obstacle.**

I. INTRODUTION

The MPS (Moving Particle Semi-implicit) method [1] is a projection-based particle method. It is similar to the SPH method in that both methods provide approximations to the strong form of Partial Differential Equations on the basis of integral interpolants. Nevertheless, the (original) MPS is characterized by simplified differential operator models solely based on a local weighted averaging process without taking the gradient of kernel function. Despite its wide-range of applicability, the MPS method has a few major drawbacks analogous to those corresponding to the SPH method. Non-conservation of momentum [2], unphysical pressure fluctuations [3,4] and numerical instability [5] are among the major drawbacks associated with MPS method.

Through the past years, the authors have been working on enhancement of MPS method by revisiting the derivation of differential operator models and by proposing more accurate, consistent schemes, while trying to maintain the simplicity and robustness of the original method. In 2008 [2], the authors proposed a Corrected version of MPS method, abbreviated as CMPS and characterized by an anti-symmetric pressure gradient model. Later, a Higher order Source term, abbreviated as HS, was derived [3] for enhancement of pressure calculation.

Another step towards enhancement and stabilization of pressure calculation by a projection-based particle method is to apply a more accurate Laplacian model for discretization of Laplacian of pressure in the PPE. Khayyer and Gotoh [4] highlighted the importance of the mathematical consistency of the Laplacian model and discretized source term of the PPE and derived a 2D Higher order Laplacian model, abbreviated as HL, for the MPS method.

In most cases, the above-mentioned enhanced particle methods have been applied to and verified by 2D calculations. On the other hand, most hydrodynamic flows are essentially three-dimensional. Hence, development of 3D accurate particle methods becomes indispensable. In this paper and consistent with our previous work [4], a 3D higher order Laplacian is derived for further enhancement and stabilization of pressure calculation in three-dimensional MPS-based simulations.

The 3D Laplacian model is derived by meticulously taking the divergence of a commonly applied SPH-based gradient model [6]. It will be shown that the Laplacian model derived in a 3D framework differs to that corresponding to a 2D one and that the same approach can be applied for derivation of consistent Laplacian models for 2D and 3D SPH simulations. The enhancing effect of the 3D Laplacian model will be shown by simulating designed exponentially excited pressure oscillations [4,5] and a schematic dam break with an obstacle [7].

II. DERIVATION OF A 3D LAPLACIAN MODEL

By considering the Laplacian at a target particle i as the divergence of the gradient calculated at that target particle and by applying the commonly applied SPH gradient model [6], the Laplacian at a target particle i would be formulated as [4]:

$$\nabla \cdot \langle \nabla \phi \rangle_i = \frac{1}{n_0} \sum_{i \neq j} \left(\nabla \phi_{ij} \cdot \nabla w_{ij} + \phi_{ij} \nabla^2 w_{ij} \right) \tag{1}$$

In 3D Cartesian coordinates, the gradients of ϕ_{ij} and w_{ij} are expressed as:

$$\begin{aligned} \nabla \phi_{ij} &= \frac{\partial \phi_{ij}}{\partial r_{ij}} \frac{\partial r_{ij}}{\partial x_{ij}} \boldsymbol{i} + \frac{\partial \phi_{ij}}{\partial r_{ij}} \frac{\partial r_{ij}}{\partial y_{ij}} \boldsymbol{j} + \frac{\partial \phi_{ij}}{\partial r_{ij}} \frac{\partial r_{ij}}{\partial z_{ij}} \mathrm{k} \\ \nabla w_{ij} &= \frac{\partial w_{ij}}{\partial r_{ij}} \frac{\partial r_{ij}}{\partial x_{ij}} \boldsymbol{i} + \frac{\partial w_{ij}}{\partial r_{ij}} \frac{\partial r_{ij}}{\partial y_{ij}} \boldsymbol{j} + \frac{\partial w_{ij}}{\partial r_{ij}} \frac{\partial r_{ij}}{\partial z_{ij}} \boldsymbol{k} \end{aligned} \tag{2}$$

where $\phi_{ij} = \phi_j - \phi_i = -\phi_{ji}$ and $\nabla \phi_{ij} = \nabla \phi_j - \nabla \phi_i$. From Eq. (2):

Experimental and Numerical Modeling of the Impulsive Dynamics of an Underwater non-Cohesive Sediment Deposit subjected to a Gaseous Jet

Sauro Manenti, Stefano Sibilla, Mario Gallati
Dept. of Civil Engineering and Architecture
University of Pavia
Via Ferrata, 1 – 27100 Pavia, Italy
sauro.manenti@unipv.it

Giordano Agate, Roberto Guandalini
Environment and Sustainable Development Dept.
RSE s.p.a.
Via Rubattino, 54 – 20134 Milan (Italy)
giordano.agate@rse-web.it

***Abstract*— This paper shows an advanced application of the SPHERA code, an SPH-based numerical model, to simulate the impulsive dynamics of a cold CO_2 jet injected from the bottom of a 2D laboratory tank containing water and a sand bed at initial rest conditions. Since the gas induces sediment motion and resuspension in the water column, this study aims at investigating an innovative technique that could be applied to increase the effectiveness of sediment removal at the bottom of an artificial reservoir by means of the combined use of explosions and flushing maneuvers. The experimental tests were simulated by introducing some severe approximations in order to handle heterogeneous materials. Even if some improvements of the model are required, the results show good qualitative agreement with the experimental frames; the SPH model can therefore lead to a first evaluation of the scouring effect when gas injection and flushing maneuvers are combined.**

I. Introdution

This paper illustrates the recent studies belonging to a wider research project that aims at the development of an innovative technique for improving the effectiveness of sediment removal at the bottom of an artificial reservoir by means of the combined use of explosions and flushing maneuvers.

Since the use of an explosive charge in a laboratory experiment requires special and expensive safety measures, at this stage of investigation its effects are mimed by a cold jet of inert gas; this represents a severe simplification since: thermodynamic effects are neglected; the involved pressure gradients are considerably smaller and the time length-scale of the phenomenon is significantly longer with respect to an underwater explosion inside non-cohesive material [1].

Despite these differences, the physics shows some analogies (inertial effects dominate) and this study, being more simple to handle, allows understanding and solving the problems encountered both in the experimental campaign and in the numerical modeling; furthermore it represent a first step toward experimental and numerical simulation of underwater explosion inside a non-cohesive sediment deposit [2].

This work consists of two parts:

1) Experimental Study: a laboratory campaign was carried out to investigate the two-dimensional dynamic behavior of underwater non-cohesive sediment layer (sand) at initial rest condition and subjected to an impulsive jet of carbon dioxide (CO_2), with known volume, temperature and pressure, from the bottom of the tank;

2) Numerical Study: the two-dimensional modeling of the sediment removal that can be obtained combining the impulsive gas jet with a flushing water flow was examined through the SPH technique.

The first stage is illustrated in section II, while section III deals with the numerical approach. Section IV contains the final conclusions.

II. Experimental Study

In this section, the experimental study carried out in the hydraulic laboratory of the Civil Engineering Department at the University of Pavia is described.

The laboratory facility is briefly described and then the experimental campaign is illustrated. The use of image processing for the detection of the interface evolution over time is finally discussed.

3-D coastal inundation simulation using a shallow-water solver

J. Zhao, D. Le Touzé, L. Gentaz, P. Ferrant
LUNAM Université, Ecole Centrale Nantes, LHEEA Lab. (UMR CNRS), Nantes, France
Jie.Zhao@ec-nantes.fr

Abstract— The application of shallow-water SPH solvers to non-linear coastal flows in 2-D shows promising capabilities to model complex free surface flows, see e.g. [1, 2]. In the present paper the shallow-water SPH numerical model is extended to 3-D real flooding events such as tsunamis, inundations following a dam break, etc. In case of very large spatial field, the domain of interest is restricted in the solver to a certain region for accurate depiction and reduction of computational costs. Boundary conditions are then carefully introduced to permit the input signal to enter the solution domain and prevent any non-physical reflections. The developed 3-D shallow-water SPH solver is then validated on flood inundation test cases of the literature, showing a close agreement of the model with results in reference.

I. INTRODUCTION

Human beings are regularly hit by major floodings resulting from tsunamis, storms or even dam breaks and recent deadly events (Tsunami in Japan: 2011, Indonesia: 2004, New Orleans flooding: 2005, France Xynthia tempest: 2010) remind us that such hazards are seriously destructive to the civilized environment. Despite large human and economic costs due to those disasters, potential dangerous areas (coastal zones and river plains prone to flooding) are still being urbanized as the population grows. The numerical simulation of floods is a crucial tool to predict their impact on land and cities. The results could be a great help to define areas unsuitable for building, to help developing protective tools (dam, channels), to build warning alert systems or evacuation plans, etc. The realism and accuracy of the numerical models are then a primary condition in order to obtain better understanding of these phenomena, leading to the definition of countermeasures for safer life.

A 3-D SPH numerical model for shallow water equations (SWEs) is presented in this work. The flexibility provided by the mesh-free Lagrangian nature of this method permits to describe any evolution of the water front on the land. This self-adaptability of the method allows more flow simulation possibilities than with grid-based methods in which the dry-wet front is difficult to track in complex geometrical situations. The SPH method is thus especially adapted in the case of large scale inundation events, where the topographies are composed of dry and wet areas of complex shapes.

Several previous studies dealt with the modeling of SWEs within an SPH framework. Chronologically, Rodrigez-Paz and Bonnet [3] developed a correction of the SPH formulation for it to be adapted to shallow water studies. Ata and Soulaimani [4] then improved the stability of the model by introducing a new calculation method based on a Riemann solver. Panizzo et al. [5] developed a model for shallow water equations applied to tsunami floodings. Anisotropic kernel and periodic redistribution allowing the following of large fluid domain expansions for inundation simulations was studied by De Leffe et al. [1]. Vacondio et al. [6] also developed a related procedure using particle splitting in order to ensure a better performance of the method for small depth locations where the smoothing length, which is inversely proportional to the water depth, becomes too large for preserving a good precision. Finally, Zhao et al. [2] developed a Roe solver solution of the SPH-SWEs permitting to improve the accuracy especially in situations involving step-like water beds.

For large field flow events, such as tsunamis, the phenomena of interest are often restricted to a relatively small region (e.g. the coast). The non-reflecting boundary conditions (NRBCs), allowing waves to enter and leave the local computational domain freely and preventing non-physical reflections to affect the flow field of interest, are very important to get a good accuracy at a reasonable computational cost. Giles [7] derived NRBCs in various conditions, based on the linearized Euler equations, for turbo-machinery flows. Vignjevic and Powell [8] produced a NRBC by the correction of elastic waves to realize multi-resolution in SPH. Lastiwka et al. [9] developed the characteristic-based NRBC within the SPH framework.

The present study is the further development of the work reported in [1, 2] with the aim of extending the model to application to inundation events involving complex realistic topographies. Two laboratory benchmark tests were reproduced, showing good agreement with experimental measurements and computational results by other numerical methods. Then a realistic flooding following a dam break was simulated to demonstrate the capability of the present solver in such complex topographical situations.

Improved accuracy in modelling armoured breakwaters with SPH

Corrado Altomare, Xavier F. Gironella
Laboratori d'Enginyeria Maritima (LIM-CIIRC)
Universitat Politecnica de Catalunya (UPC)
Barcelona, Spain
corrado.altomare@upc.edu

Alejandro J.C. Crespo, Jose M. Domínguez,
Moncho Gómez-Gesteira
Environmental Physics Laboratory (EPHYSLAB)
Universidade de Vigo
Ourense, Spain

Benedict D. Rogers
Modelling and Simulation Centre (MaSC),
School of Mechanical, Aerospace and Civil Engineering (MACE)
University of Manchester
Manchester (U.K.)

***Abstract*— The present paper reports the added-value of using SPH in the fluid-structure interaction occurring in the run-up and overtopping phenomena for a the rubble mound breakwater, where the seaward layer is made of grooved cubic blocks. The main goal is the study of the equivalent roughness of the outer layer, defined as the ratio of the maximum water level (run-up) on the armoured structure to the run-up on a smooth similar one. Run-up heights and overtopping rates have also been compared with empirical solutions. The complex geometry of the blocks and the surrounding gaps within breakwaters must be efficiently simulated to obtain a detailed description of the flow. Using novel computing solutions such as the new graphics processing unit GPU technology used in the DualSPHysics code (www.dual.sphysics.org), the fluid-structure interaction is being modelled with many armour blocks that are representative of the real structure. GPU computing enables the simulation of millions of particles leading to potentially better accuracy of the numerical run-up and overtopping estimates to obtain the value for the structure roughness.**

I. INTRODUCTION

Coastal defences are built to protect the world population that lives in coastal zones. These structures defend not only houses, seawalks, beaches, recreational area, but also harbours and ports where the maritime shipping plays a key role in the world economy. These defences protect infrastructures against storm surge and large waves that may cause run-up and overtopping of such structures like breakwaters, dikes, seawalls, etc., leading to potential damage and flooding of the area behind the structure.

In particular, this work is focused on the design of breakwaters such those shown in Figure 1. The study and analysis of the better design of a breakwater can be modelled physically in laboratory facilities or computationally using a numerical model. Many examples can be found in the literature [1, 2, 3] examining the optimization and design of different breakwaters.

Figure 1. Picture of the antifers breakwater in Molfetta (Italy) and rubble mound breakwater in Muxia (Spain).

In breakwater design, the wave run-up and overtopping rate are key parameters to define the breakwater crest height. The resilience of these structures against the waves can be represented by means of the slope equivalent roughness coefficient. This coefficient depends on the armour units layout, shape and size, and hence the porosity of the seaward layers. An appropriate value of roughness based on detailed experimental and numerical validation can lead to a significant reduction of overtopping flows and run-up height without unnecessary expensive designs.

Within the European research project CLASH (*Crest Level Assessment of coastal Structures be full-scale monitoring, neural network prediction and Hazard analysis on permissible wave overtopping*, www.clash-eu.org), field measurements were carried out at various locations worldwide along with laboratory tests. One of the main goals of CLASH project was the creation of a large database containing more than 10,000 test results on wave overtopping to develop a generic prediction method for overtopping at coastal structures. One of these locations in Europe with complete series of field measurements is a rubble mound breakwater armoured with antifer blocks at Zeebrugge, Belgium. In the proposed work, the Zeebrugge breakwater will be used to validate the numerical results. A further objective of the CLASH project also focused on

Simulation of dam-break flow in channel expansion with coupled 2-D/3-D SPH model

Elvira Džebo, Dušan Žagar, Matjaž Četina
Faculty of Civil and Geodetic Engineering
University of Ljubljana
Ljubljana, Slovenia
edzebo@fgg.uni-lj.si, dzagar@fgg.uni-lj.si,
mcetina@fgg.uni-lj.si

Gregor Petkovšek
HR Wallingford
Wallingford, United Kingdom
g.petkovsek@hrwallingford.com

Abstract—**The paper presents the simulation of a dam-break flow in a channel expansion with a coupled 2-D/3-D SPH mathematical model Tis Isat. The model was calibrated and validated against the experiments of Martin and Moyce [1]. Overall agreement between the Tis Isat model and other available models was good while local behaviour at the solid boundaries was better [2]. The novelty of the presented approach is in the application of a coupled 2-D/3-D SPH model. The proposed method is applicable to systems where the flow transits from 2-D to 3-D state. The proposed approach significantly shortens the computational time of the SPH simulations, while the precision of results is still satisfactory.**

A laboratory experiment of the wave after instantaneous collapse of a dam [3] was used for calibration. The downstream channel slope was 0.087% and the channel width expanded instantaneously from 0.4 m to 1.2 m. The results of the coupled 2-D/3-D mathematical model Tis Isat were compared to measurements on this physical model, results of a 1-D finite difference model and results of the fully 3-D mathematical model Tis Isat. The overall agreement of the results was at least as good as achieved with other models.

I. INTRODUCTION

Numerical models for simulating water flow are an increasingly widespread and indispensable tool for analyzing engineering problems. Mathematical models can be classified into two major groups: Eulerian and Lagrangian models. The Eulerian approach divides the computational domain into grid cells and the Lagrangian approach keeps track of individual fluid elements as they move through an area. Eulerian models are most often used to analyze hydrotechnical problems because the numerical mesh methods are less demanding on hardware capabilities. However, they also have some weaknesses, such as numerical diffusion. The disadvantages of mesh methods can be completely avoided by using the Lagrangian computational model. The Lagrangian approach requires longer computational times than the Eulerian approach. This is not surprising because the Lagrangian approach uses a large number of particles and short time steps. However, computational time can be decreased by greater hardware capabilities and new methodologies (e.g. the method for coupling 2-D and 3-D models).

Smoothed particle hydrodynamics (SPH) is one of the most successful and popular Lagrangian methods for free surface flow simulations. The downside of the SPH method is the longer computational time. Reducing the computational time is a challenge for the SPH method. It can be reduced by coupling with computationally less demanding models. Some authors have proposed different techniques for coupling the SPH model with mesh methods: e.g. with DEM [5 and 6], with FEM [7, 8, 9 and 10], and with a 1-D Boussinesq-type wave model [11].

In areas where the hydrodynamics can be described as a width-averaged flow (e.g. in rectangular channels), less demanding 2-D models are often used. Three-dimensional models are used for simulations of fluid flow in a changeable domain. In this work we propose a simple method for coupling two models with different dimensionalities. This option is applicable in systems where the flow transits from 2-D to 3-D and as far as we know, coupling of models with different dimensionalities (2-D and 3-D SPH models) is a new approach in the SPH method that is here used for the first time.

An upgraded version of the model Tis Isat, developed at the University of Ljubljana, was used in our research. Tis Isat was in relatively good agreement with the experiment of Martin and Moyce [1] and with other available models, while the behaviour near the boundaries was better [2]. With a simple coupling algorithm it is possible to combine both of the Tis Isat models, 2-D and 3-D. We used the coupled 2D/3D model to simulate water flow in a channel with expansion and compared the results with measurements and other models. We expected that the new coupling procedure would significantly shorten the computational time without decreasing the quality of results.

SPH benchmarking: a comparison of SPH variants on selected test cases within the NextMuSE initiative

D. Le Touzé, D.A. Barcarolo, M. Kerhuel
LHEEA Lab.
LUNAM Université, Ecole Centrale de Nantes
Nantes, France
nextmuse@ec-nantes.fr

F. Leboeuf, J. Caro
LMFA
Ecole Centrale de Lyon
Lyon, France

N. Quinlan, L. Lobovsky, M. Basa
National University Galway
Galway, Ireland

A. Colagrossi, S. Marrone
CNR-INSEAN
Roma, Italy

J.-C. Marongiu
Andritz Hydro
Vevey, Switzerland

M. de Leffe, P.-M. Guilcher
HydrOcean
Nantes, France

***Abstract*—In this paper are presented comparisons of SPH variants on academic test cases classically used to validate numerical fluid dynamics software. These comparisons are extracted from NextMuSE FP7 project activities which will be published more extensively in near future. One of the goals of this project was to better understand the SPH method to leave the path to its establishment within CFD methods. An important work load was thus dedicated to benchmark SPH variants on selected test cases.**

A number of results and conclusions of this comparative study are presented in this paper. The studied variants are: standard weekly-compressible SPH, δ-SPH, Riemann-SPH, incompressible SPH, and FVPM. The majority of the test cases also present a reference solution, either experimental or computed using a mesh-based solver. Test cases include: wave propagation, flow past a cylinder, jet impact, floating body, bubble rise, dam break on obstacle, etc.

Conclusions may help SPH practitioners to choose one variant or another and shall give detailed understanding necessary to derive further improvements of the method.

I. INTRODUCTION

The ultimate goal of the FP7 European project NextMuSE is to make the proof of concept of a new generation of simulation environment, interactive and immersive, based on the SPH technology (http://nextmuse.cscs.ch). To achieve this objective the better understanding and further improvement of the SPH method was a key task. To that purpose project participants dedicated an important work load to benchmark SPH variants on selected test cases.

Part of the results and conclusions of this comparative study are presented in this paper. The studied variants of SPH are the ones mostly used in the literature, including the Finite-Volume Particle Method (FVPM): standard weekly-compressible SPH, δ-SPH, Riemann-SPH, explicit incompressible SPH, and FVPM. These variants are not recalled here; the reader may refer to the different articles on the topic written by the authors of the present article in journals and in the present and past issues of the SPHERIC workshop proceedings. Many of these papers are listed on the NextMuSE website [1].

Almost all the test cases chosen present a reference solution, either experimental or computed using a mesh-based solver. Test cases include: wave propagation, flow past a cylinder, jet impact, floating body, bubble rise, dam break on obstacle, etc. Only some results could be presented here due to the length of the paper.

This careful investigation of SPH variants permits to draw interesting and contrasted conclusions which may be of help to the community of SPH practitioners in its future choices of research.

II. WAVE PROPAGATION: VERY-SLOW DYNAMICS INVISCID TEST CASE

A. *Test case choice*

This test case concerns the propagation of a regular gravity wave in 2D wave basin. It is generated by a paddle wavemaker located on the left end. Surface tension effects are negligible. This test case involves a problem which, in terms of physics, is not at all in the scope of the SPH method application to free-surface flows which concerns fast dynamics flows with strongly-nonlinear free-surface deformations. However, this test case is very interesting to validate the method in the sense that it enables to monitor numerical dissipation (non-breaking ocean waves are quasi-inviscid at standard scale) in a free-surface problem. Besides, wave propagation can be of interest to be able to address violent wave-body interactions.

Results are compared to the inviscid potential-flow BEM (Boundary Element Method) results of [2]. δ-SPH and Riemann-SPH variants are compared. Standard WCSPH is not

Author Index

Adami, S. 18
Adams, N. A. 18 179
Agate, G. 229 369 381
Altomare, C. 395
Amicarelli, A. 369
Antuono, M. 7 107 163 200
Avellan, F. 214
Ayliffe, B. A. 131
Bakir, F. 78
Barcarolo, D. A. 99 409
Basa, M. 282 355 409
Bierwisch, C. 59
Bilotta, G. 171 274
Børve, S. 361
Bouscasse, B. 244
Breinlinger, T. 14
Broglia, R. 244
Bubel, J. 38
Caro, J. 409
Cercós, J. L. 303
Četina, M. 403
Chan, E. S. 157
Chang, K.-H. 144
Chang, T.-J. 144
Cliffe, K. A. 317
Colagrossi, A. 7 244 107 163 200 409
Colicchio, G. 7 163
Connolly, A. 65
Crespo, A. J. C. 266 395
Dalrymple, R. A. 171 274
Dao, M. H. 157
de Leffe, M. 259 409
de Vuyst, F. 99
del Negro, C. 274
Domínguez, J. M. 266 395
Džebo, E. 403
Farzaneh, S. 78
Feldman, J. 347
Ferrand, M. 149 296
Ferrant, P. 387
Gallati, M. 229 381
Gallo, G. 274
Gentaz, L. 387
Geyer, T. 31
Gironella, X. F. 395
Gomez-Gesteira, M. 266
González, L. M. 303
Gotoh, H. 340 375
Grabe, J. 38
Grant, J. 84
Graziani, G. 163
Grenier, N. 7
Groenenboom, P. 333
Guandalini, R. 229 369 381
Guibert, D. 259
Guilcher, P.-M. 409
Hashibon, A. 14
Hérault, A. 171 274
Hu, X. Y. 18 179
Hudson, D. 252
Iannucci, L. 65
Jahanbakhsh, E. 214
Jalali Farahani, R. 171
Kajtar, J. B. 195
Kassiotis, C. 296
Kerhuel, M. 7 409
Khayyer, A. 340 375
Khelladi, S. 78
Khorasanizade, Sh. 92
Kordilla, J. 31
Koumoutsakos, P. 221
Kraft, T. 14
Kruisbrink, A. C. H. 317
Lagger, H. 59
Laibe, G. 131
Le Touzé, D. 7 99 387 409
Leboeuf, F. 208 409
Leonardi, M. 137
Lind, S. J. 311 324
Lobovský, L. 282 355 409
López, Y. R. 71
Maciá, F. 303
Mahmood, O. 296
Manenti, S. 137 229 381
Marongiu, J.-C. 208 409
Marrone, S. 244 163 200 409
Mayrhofer, A. 149
Mignosa, P. 347
Minatti, L. 51
Monaghan, J. J. 23 195 187
Morfa, C. R. 71
Morvan, H. P. 317
Moseler, M. 59
Nestor, R. M. 355
Neuhauser, M. 208
Nutto, C. 59
Oger, G. 259
Pâkozdi, C. 287
Pan, W. 23
Paris, E. 51
Parkinson, E. 208
Pearce, F. R. 317
Pearce, M. 252
Petkovšek, G. 403
Piccinali, J.-G. 259
Pinto, J. F. 92
Prakash, M. 84
Price, D. J. 115 123 131
Quinlan, N. J. 282 355 409
Rentschler, M. 208
Riviere, S. 78
Rogers, B. D. 149 266 296 311 324 347 395
Roose, D. 71
Rung, T. 43 236
Rustico, E. 274
Semercigil, S. E. 84

Sibilla, S. 1 137 229 381
SINTEF SCORE-team 287
Skillen, A. 311
Sousa, J. M. M. 92
Souto-Iglesias, A. 107
Stansberg, C. T. 287
Stansby, P. K. 311 324 347
Stefanova, B. 38
Tartakovsky, A. M. 23 31
Tcharkhtchi, A. 78
Thomas, G. 252
Tkalich, P. 157
Torti, E. 1
Tricco, T. S. 115
Turan, O. F. 84
Ulrich, C. 43 236
Vacondio, R. 347
Valdez-Balderas, D. 266
Valizadeh, A. 187
van Rees, W. M. 221
Vessaz, C. 214
Violeau, D. 149 296
Xu, H. 157
Yue, T. 317
Žagar, D. 403
Zhao, J. 387
Zuzio, D. 7