TUHH

6th International SPHERIC Workshop

8-10 June 2011

Organised by

Inst. Fluid Dynamics and Ship Theory
Hamburg University of Technology

Edited by

Thomas Rung
Christian Ulrich

Sponsored by

Deutsche Forschungsgemeinschaft

DFG

Fluid Dynamics
and Ship Theory

6th International SPHERIC Workshop
First Edition, Hamburg, Germany, June 2011

ISBN 978-3-89220-658-3

Editors:
Prof. Dr.-Ing. Thomas Rung, Hamburg University of Technology
Christian Ulrich, Hamburg University of Technology

Published by

SCHRIFTENREIHE SCHIFFBAU
Schwarzenbergstraße 95c
21073 Hamburg, Germany
http://www.tuhh.de/vss

Printed in Germany by Elbepartner Breitschuh & Kock GmbH, Hamburg.

Foreword to the 6th Int. SPHERIC Workshop

Dear Delegate,

The Institute for Fluid Dynamics and Ship Theory of the Hamburg University of Technology (TUHH) is honoured and pleased to organise the 6th international SPHERIC workshop. It is a great pleasure for us to welcome you to the exciting and cosmopolitan city of Hamburg and we look forward to sharing a successful and enjoyable meeting with you.

The SPHERIC workshop has taken a root as the primary annual event on Smoothed-Particle Hydrodynamics (SPH). It is organised under the aegis of the *SPH EUROPEAN RESEARCH INTEREST COMMUNITY* (SPHERIC) which forms part of the widely recognised ERCOFTAC network. The successful concept of SPHERIC is due to the strong methodological focus in an interdisciplinary application environment, integrating the know-how of physicists, mathematicians, IT experts and engineers from academia and industry.

The SPHERIC community plays an important role in the advancement and dissemination of SPH, an exciting scientific-computing method which has considerably matured over the past five years. The range of SPH applications is continuously growing and involves tsunami and landslide simulations, cosmic-structure formation and galaxy collisions, tank sloshing, pedestrian-crowd simulation, virtual-reality simulations, wave-energy generation, slamming loads on offshore structures, flooding of ships sections, decay of tip vortices behind an aircraft and lung-respiration simulations amongst others. Due to the large computational effort, a significant branch of SPH research is concerned with high-performance computing on the most recent hardware technologies.

Similar to previous years, a record number of abstracts was submitted (eighty-three in 2011) which clearly indicates the sustained increase in interest in the workshop. Dedicated to the aim of stimulating an enhanced direct exchange of ideas, we decided to continue with the workshop ethos of the event and the Scientific Committee had to confine the selection of presentations in order to allow everyone to view all presentations. In line with the constant growth of the workshop, the organisation becomes more demanding. This event would not be possible without the generous financial support provided by Deutsche Forschungsgemeinschaft (DFG) which is greatly acknowledged.

We are confident that the workshop series continues in its friendly and creative spirit and that you will leave Hamburg with new ideas and inspirations.

Thomas Rung, Christian Ulrich
Local Organising Committee of the 6th International SPHERIC Workshop 2011

Committees

SPHERIC Steering Committee (Workshop Scientific Committee)

Dr David Le Touzé (Ecole Centrale de Nantes, France)
Damien Violeau (Electricité de France, LNHE, Chatou, France)
Dr Nathan Quinlan (National Univ. of Ireland, Galway)
Dr Ben Rogers (Univ. of Manchester, UK)
Prof. Moncho Gómez-Gesteira (Univ. de Vigo, Spain)
Prof. Stefano Sibilla (Univ. of Pavia, Italy)
Dr Jean-Christophe Marongiu (ANDRITZ Hydro, Switzerland)
Dr Paul Groenenboom (ESI BV, The Netherlands)
Dr Jason Hughes (Univ. of Plymouth, UK)
Dr Andrea Colagrossi (INSEAN, Italy)
John Biddiscombe (CSCS, Switzerland)
Dr Xiangyu Hu (Technical University of Munich, Germany)
Prof. Rade Vignjevic (University of Cranfield, UK)

Local Organising Committee – Hamburg Univ. of Technology

Prof. Dr.-Ing. Thomas Rung – Chair
Christian Ulrich

Workshop Office – TuTech Innovation GmbH

Gerlinde Löbkens

Contents

Day 1: Wednesday, June 8th, 2011

Day 2: Thursday, June 9th, 2011

Day 3: Friday, June 10th, 2011

Non-Reflecting Boundary Conditions and Multi-Resolution SPH

R Vignjevic
Applied Mechanics
School of Engineering, Cranfield University
Bedfordshire, MK43 0AL, UK
v.rade@cranfield.ac.uk

S Powell
Applied Mechanics
School of Engineering, Cranfield University
Bedfordshire, MK43 0AL, UK

Abstract— **For many problems in computational continuum mechanics an accurate treatment of non-reflecting boundaries that allows incoming waves to propagate out of a finite domain is important. One application is mesh-refinement; high frequency components of a solution on a fine mesh cannot be transmitted into a more coarsely meshed region and will reflect back in a non-physical manner. A robust non-reflecting boundary condition (NRBC) can therefore allow high spatial resolution to be used only where necessary without the need for a graduated transition region.**

We propose a simple non-reflecting boundary formulation for SPH, efficacy is investigated in 2d and the NRBC will be applied to a multi-resolution simulation in 1d.

I. INTRODUTION

There are two distinct approaches explored by researchers thus far for increasing spatial resolution in SPH. The particle splitting approach; where the particle distribution is refined adaptively by splitting particles into a set of child particles when some criterion is met. Or starting with an initial multi-resolution discretisation; where different particle density is used in zones where it is known *a priori* to be needed.

An adaptive SPH algorithm has to address the following issues:

- When to re-mesh; the adaptivity criterion or error estimate.
- How to re-discretise the zone where the adaptivity criterion is satisfied; the new particle distribution.
- How to interpolate field variables onto the new particles.

Dealing with each item in turn: [1] add particles in regions with high velocity gradients, removing them when the gradient is low. Eschewing calculation, in [2] fluid particles are split whenever they enter a refinement zone. In the examples given in the paper the particles are refined just before they leave a variety of tanks. For astrophysical simulations [3], using a variable smoothing length, refine a particle whenever its smoothing length exceeds twice the average smoothing length and [4] adapt based on the Jeans condition. The last two methods are rather specific to astrophysics.

The reproducing kernel particle method [5] is a mesh-free method closely related to SPH. To identify sub-domains in need of refinement the authors use the properties of the kernel function as a filter to find regions dominated by high-frequencies and therefore it high-gradients.

Increasing resolution is most often done by splitting a particle into a number of smaller child particles. As long as the parent particle's mass is distributed evenly between the children conservation of mass is guaranteed. The spatial distribution of the child particles affects the density distribution and the splitting procedure can introduce extra errors in this way. Generally a particle is split by having one particle in the same position as the parent and placing the others evenly around it. Assuming the particle is split into equally sized child particles and that the are placed symmetrically around the original particle's position, it is only left to decide on the new particles' smoothing lengths and separation, l, from the centre particle. Kitsionas *et al.* [4] split the particle into 13 child particles and decide that the new smoothing length will be $h_{new} = 13^{-1/3} h_{old}$. After some experimental analysis the authors settle for a $l = 1.5 h_{new}$ as the particle separation. In a similar way [2] define a density error function depending on two parameters, the new smoothing length and the separation, the minimisation of this function is used to find approximately optimal pairs of values. Alternatively [1] simply insert an extra particle where needed. The difficulty with this approach is that the new particle's position and the masses and smoothing lengths of its neighbours are calculated to approximate the original density distribution instead of being pre-determined by a splitting algorithm. This procedure is more complex and mass conservation is no longer automatic. The benefit is that it avoids the difficulties associated with abrupt steps in the smoothing lengths that may arise when simply splitting particles.

The final step is interpolating the field variables on to the refined particle distribution. The interpolation procedures vary from a simple weighted average; in [6]the child particle's velocities are given by,

$$v_{i'} = \sum_{k \in N_i} v_k W_{ik}\left(h_{ik}\right) \Delta V_k \quad ,$$

Characterisation of SPH noise in simulations of protoplanetary discs

S.E. Arena, J-F. Gonzalez and E. Crespe
Université de Lyon, Lyon, F-69003, France;
Université Lyon 1, Observatoire de Lyon, 9 avenue Charles André , Saint-Genis Laval, F-69230, France;
CNRS, UMR 5574, Centre de Recherche Astrophysique de Lyon; École Normale Supérieure de Lyon, Lyon, F-69007, France
serena.arena@ens-lyon.fr

***Abstract*—The aim of this work is the characterisation of the noise in SPH simulations of gaseous viscous discs and it is motivated by the study of the dynamics of the dust component in protoplanetary discs where the dust is coupled to the turbulent gas component.**

We perform a statistical analysis of the properties of the gas fluctuating velocity field, considering both its magnitude and its structure, and compare the results to some know features of turbulence.

We have found that the properties of SPH noise are determined by the global effect of the numerical noise and the model of artificial viscosity. In particular, the considered artificial viscosity model produces effects similar to a subgrid scale turbulence model. In fact, for several combinations of the involved parameters turbulence-like behaviour has been observed. However, it turns out that the combinations that better reproduce turbulence are associated with the less intense fluctuation fields. Therefore, the artificial viscosity model is able to reproduce the effects of diffusion and gas viscosity produced by turbulent eddies, but it cannot resolve them.

I. Introduction

Protoplanetary discs (abbreviate in PPD) are discs composed mainly by gas and dust in quasi-keplerian rotation around young stars. They are belived to be the birth place of planets. The theory of planet formation is complex because several scales (from μm to hundreds of thousand km), several forces (e.g. electrostatic, magnetic, gravitational, drag) and several processes (e.g. turbulence, chemical and thermodynamical transformations, instabilities) are involved.

One of the more poorly understood stages is the evolution of μ-meter size dust grains up to km size objects, called planetesimals.

Among the several mechanisms the dust is subject to (e.g. radial drift and vertical settling), turbulence is expected to be of relevant importance because it can have two competitive effects mediated by gas drag: stir up and diffuse dust particles, contrasting their agglomeration, or trap them inside eddies, favoring their agglomeration. Current observations have not reached yet the resolution necessary to directly detect turbulence and study its effects. However, the measured values of mass accretion rates onto the central star ($\dot{M} \approx 10^{-8}$ M$_\odot$; Hartmann et al. 1998 [1], Andrews et al. 2009 [2]) and the estimated life time of discs (around 10^7 yr) imply a kinematic viscosity much larger than the molecular viscosity in PPD (20 m^{2}s^{-1}). Shakura & Sunyaev (1973) [3] noted that turbulence can provide an effective viscosity able to justify these data.

Given the complex interplay between dust and gas, the problem can be only addressed by numerical simulations.

Here we focus on the modeling of the gaseous disc with particular attention to the fluctuating velocity field. We have performed 3D Smoothed Particle Hydrodynamics (SPH) simulations of gaseous accretion discs. Our aim is to clarify if the numerical noise intrinsically present in SPH simulations of accretion discs can mimic the effects of turbulence and to what extent.

II. The method

We have performed a set of SPH simulations of a gaseous disc changing both numerical and physical parameters. For each simulated disc we have studied both the magnitude and the structure of the fluctuations present in the velocity field. The resulting properties of such fluctuations have been compared to the typical behaviour of turbulence and to results from grid based simulations available in the literature.

A. The disc model

A typical T Tauri disc of mass $M_{\rm disc} = 0.01 M_{\rm star}$ orbiting around a one solar mass star ($M_{\rm star} = M_\odot$) is considered. It extends from 20 to 400 AU (Astronomical Units), it is characterised by a surface density radial profile given by $\Sigma(r) = \Sigma_0 (r/r_0)^{-p}$, it is locally isothermal with a sound speed radial profile $c_s(r) = c_{s_0}(r/r_0)^{-q}$. The semi-thickness H of the disc is related to the sound speed and to the angular velocity Ω by $H(r) = c(r)/\Omega(r)$. Note that the sound speed coefficient c_{s_0} and the sound speed exponent q determine respectively the semi-thickness of the disc and its radial dependence $H(r) \propto r^{3/2-q}$.

The reference values we adopt in the following are $p = 3/2$ and $q = 3/8$ typical of the Minimum Mass Solar Nebula. The disc is slightly flared with $H(r)/r = 0.05$ at $r = 100$ AU.

The evolution of the disc is followed for about 10 orbits (at 100 AU) after numerical thermalisation has been reached.

B. The code

We use the two-phase SPH code described in Barrière-Fouchet et al. (2005) [4]. The two phases represent gas and dust that interact via aerodynamic drag. The gas is described

A Stabilising Diffusion-Based Shifting Algorithm for Incompressible Smoothed Particle Hydrodynamics

S. J. Lind, R. Xu, P. K. Stansby, B. D. Rogers
School of Mechanical, Aerospace, and Civil Engineering
University of Manchester
Manchester, UK
steven.lind@manchester.ac.uk

***Abstract*—The incompressible smoothed particle hydrodynamics (ISPH) method with projection-based pressure correction has been shown to be highly accurate and stable for internal flows and, importantly for many problems, the pressure field is virtually noise-free in contrast to the weakly compressible SPH approach (Xu et al. (2009)). However for almost inviscid fluids instabilities at the free surface occur due to errors associated with the truncated kernels. A new algorithm is presented which remedies this issue, giving stable and accurate solutions to both internal and free-surface flows. Generalising the particle shifting approach of Xu et al. (2009), the algorithm is based upon Fick's law of diffusion and shifts particles in a manner that prevents highly anisotropic distributions and the onset of numerical instability. The algorithm is validated against analytical solutions for an internal flow for Reynolds numbers as large as 10^6, the flow due to an impulsively started plate, and highly accurate solutions for wet bed dam break problems at small times. The method is then validated for progressive regular waves with paddle motion defined by linear theory. The accurate predictions demonstrate the effectiveness of the algorithm in stabilising solutions and minimising the surface instabilities generated by the inevitable errors associated with truncated kernels. The test cases are thought to provide a more thorough quantitative validation than previously undertaken.**

I. Introduction

The classical SPH method is formulated as a weakly compressible fluid with an artificial equation of state defining pressures. Although this predicts some highly transient flows quite well, notably dam break flows, pressures are extremely noisy and the method highly dissipative [1]. An important development was made by Vila [2] who introduced an arbitrary Lagrangian Eulerian formulation with a Riemann solver for each particle interaction, reducing pressure noise markedly. However the artificial equation of state remains. Cummins and Rudman [3] proposed a direct approach for incompressible flow in SPH following the projection method of Chorin [4] for maintaining zero velocity divergence. ISPH can be highly accurate and effectively noise free but instability arises if particle distributions become highly distorted and errors amplify rapidly [1]. This instability can be avoided in various ways [5]–[7], but the stabilisation method of Xu et al. [8] in particular has been shown to be highly accurate and cost effective. The stabilisation method proposed in [8] involves slightly shifting the particles across streamlines, thereby avoiding the extreme stretching and bunching of particles. The hydrodynamic variables are then adjusted through interpolation. The algorithm however has weaknesses for the simulation of free-surface flows. Numerical instabilities can develop on the free surface as a result of the incomplete kernel support and increased kernel interpolation error, at least partly due to errors in the precise definition of the free-surface location. It was found that these free-surface instabilities could be damped by artificially increasing the viscosity of particles on or near the free surface [9]. While this approach produced accurate and stable free-surface profiles, viscosity magnification at the free surface is physically undesirable and inconsistent with the Navier-Stokes equations. In this work a new algorithm is presented which shifts particles according to Fick's law of diffusion. In a similar, but more general manner to [8], the algorithm acts to maintain a regular distribution and prevent the extreme stretching and bunching of particles. Furthermore, it can be applied to both internal and free-surface flows to produce accurate and stable results. Many problems in engineering involve both fluid-structure interaction and violent free-surface motion. Thus, a highly accurate and rigorously validated, incompressible, mesh-free, noise-free method for arbitrary free-surface flows is very attractive. The following validation cases will be considered: a Poiseuille flow, a Taylor-Green flow, an impulsive fluid-structure interaction, a wet-bed dam break problem, and regular wave propagation. To the authors' knowledge, this is the first time ISPH simulations of high Reynolds number free-surface flows have been validated against highly accurate solutions for both pressure and velocity fields. The scheme may be generalized to 3-D motion with complex boundaries through massively parallel processing but this is not considered here.

II. SPH Methodology

The governing equations to be solved are the incompressible Navier-Stokes equations, comprised of the conservation of mass,

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

and the conservation of momentum,

Dimension reduction method for SPH equations

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

Tim D. Scheibe
Hydrology Group,
Pacific Northwest National Laboratory,
Richland, USA
Tim.Scheibe@pnl.gov

***Abstract*—Smoothed Particle Hydrodynamics model of a complex multiscale processe often results in a system of ODEs with an enormous number of unknowns. Furthermore, a time integration of the SPH equations usually requires time steps that are smaller than the observation time by many orders of magnitude. A direct solution of these ODEs can be extremely expensive.**

Here we propose a novel dimension reduction method that gives an approximate solution of the SPH ODEs and provides an accurate prediction of the average behavior of the modeled system. The method consists of two main elements. First, effective equationss for evolution of average variables (e.g. average velocity, concentration and mass of a mineral precipitate) are obtained by averaging the SPH ODEs over the entire computational domain. These effective ODEs contain non-local terms in the form of volume integrals of functions of the SPH variables. Second, a computational closure is used to close the system of the effective equations. The computational closure is achieved via short bursts of the SPH model. The dimension reduction model is used to simulate flow and transport with mixing controlled reactions and mineral precipitation. An SPH model is used model transport at the porescale. Good agreement between direct solutions of the SPH equations and solutions obtained with the dimension reduction method for different boundary conditions confirms the accuracy and computational efficiency of the dimension reduction model. The method significantly accelerates SPH simulations, while providing accurate approximation of the solution and accurate prediction of the average behavior of the system.

I. Introduction

Smoothed Particle Hydrodynamics is a versatile tool to model complex physical processes. For a certain type of processes (mainly multiscale processes), SPH discretization (as any other numerical discretization) may lead to a system of ODEs with an enormous number of unknowns. Furthermore, a time integration of theSPH equations usually requires time steps that are smaller than the observation time by many orders of magnitude. A direct solution of such SPH ODEs can be extremely expensive. This necessitates development of advanced algorithms for model (or dimension) reduction.

Often, we are not interested in the SPH solution itself, but in the average behavior of the modeled system. Equations for the averages can be obtained using a volume averaging of the SPH [1], [2]. Usually the equations for averages contain non-local terms. When scale separation exists, accurate empirical or analytical closures can be developed [3]–[5]. A well known example of a problem with scale separation is hydrodynamics where separation exists between hydrodynamics and molecular scales, and the Navier-Stokes equations are empirical closed-form equations for hydrodynamics averages. In the absence of scale separation, except for special cases, closed-form averaged equations do not exist.

Here we focus on flow and transport in porous media. On the pore scale, transport is described by a combination of Navier-Stokes and advection-diffusion-reaction equations [6]. There are a number of examples (e.g. transport with mineral precipitation and/or microbial growth with mixing controlled reactions [10]–[13]) of reactive transport problems where scale separation does not exist and traditional advection-dispersion equations do not provide an accurate approximation of the average solution of the pore-scale advection-diffusion equations [9]. In such cases, accurate simulation of macroscopic behavior may require explicit simulation of pore-scale processes.

We propose a novel dimension reduction method for a large system of SPH equations describing pore-scale (microscale) flow and transport [11]. The method uses a computational closure for non-local averaged equations, so we name it the Computational Closure Method (CCM). CCM is especially useful when accurate closed form equations for averages do not exist. CCM is gives an approximate solution of the SPH ODEs and provides an accurate prediction of the average behavior. Effective ODEs for evolution of average variables (e.g. average velocity, concentration and mass of a mineral precipitate) are obtained by averaging the SPH ODEs over the entire pore-scale domain. These effective ODEs contain non-local terms in the form of volume integrals of the SPH variables. The computational closure is achieved via short bursts of the SPH model. The effective ODEs have fewer degrees of freedom and can be interated with longer time steps than the SPH equations. The CCM-SPH model is used to simulate flow and transport with mixing controlled reactions and mineral precipitation. Good agreement between direct solutions of the SPH equations and solutions obtained with CCM for different boundary conditions confirms the accuracy and computational efficiency of CCM. CCM significantly accelerates SPH simulations, while providing accurate approximation of the SPH solution and accurate prediction of the average behavior of the system.

II. SPH transport equations

CCM is applicable to a wide range of ODE systems describing particle dynamics or obtained from discretization of

Benefits of using a Wendland kernel for free-surface flows

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

M. Antuono
INSEAN
00128 Rome, Italy.
matteoantuono@gmail.com

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

***Abstract*—The aim of this paper is to discuss the influence of the selection of the interpolation kernel in the accuracy of the modeling of the internal viscous dissipation in free surface flows. Simulations corresponding to a standing wave, for which an analytic solution is available, are presented. Wendland and renormalized Gaussian kernels are considered. The differences in the flow patterns and internal dissipation mechanisms are documented for a range of Reynolds numbers. It is shown that the simulations with Wendland kernels replicate the dissipation mechanisms more accurately than those with a renormalized Gaussian kernel. Although some explanations are hinted, we have failed to clarify which the core structural reasons for such differences are.**

Symbols

WC2	Wendland 5^{th} degree class 2 kernel
RGK	Renormalized Gaussian kernel

I. Introduction

In Colagrossi et al. [1], a theoretical analysis of the two most popular SPH formulations for Newtonian viscous terms, i.e. Monaghan's [2] and Morris *et al.*'s [3], was carried out at the continuous level. It was shown that in the presence of a free surface and under certain conditions, these viscous terms become singular with the inverse of the smoothing length. It was also demonstrated that for the Monaghan's [2] viscous term the singularity does not affect the integral flow quantities. As a consequence, the exact mechanical energy dissipation rate is recovered in the continuous approximation as the smoothing length goes to zero.

While in practical SPH implementations for pure diffusion processes, the viscous terms behave similarly [4], in order to assess the practical implications of the aforementioned results, a viscous free surface flow is simulated with SPH. Such flow is the evolution of a standing wave. This is a classical problem in the scientific literature and is of practical interest since it is strictly related to the propagation of gravity waves. Moreover, an analytic solution for the decay of the kinetic energy is available (see Lighthill [5]). The SPH simulations have been implemented using free-slip conditions along the solid boundary of the tank and using a renormalized Gaussian-Kernel [6]. and a fifth order class 2 Wendland [7] kernel. The aim of the present work is to document the benefits of using such a kernel when dealing with viscous free surface flows.

The paper is organized as follows: first, the physical problem we are interested in is presented. Second, the SPH model, emphasizing the properties of the considered kernels is introduced. Third, the practical problem considered, which has been the dissipation of the kinetic energy in a standing wave, is introduced and the results of the computations carried out, for a range of Reynolds numbers, are presented and discussed. Finally, some conclusions are drawn and future work threads hinted.

II. Governing equations

Free surface 2D laminar Newtonian incompressible flows are treated in this paper. The viscous effects in these flows are proportional to the Laplacian of the velocity field, $\boldsymbol{u}$. The equations that describe these flows are the Navier-Stokes incompressible ones.

In order to close this system of equations it is necessary to specify the boundary conditions (BC). Let the fluid domain be noted as Ω and its boundary as $\partial\Omega$. Such boundary encompasses a free surface boundary $\partial\Omega_F$ and solid boundaries $\partial\Omega_B$. Along the free surface, both a kinematic and a dynamic BC should be in principle fulfilled. They are not explicitly included in the SPH simulation, since the weakly compressible model is considered and such model can be properly shaped in order to being inherently consistent with the kinematic and pressure conditions [8] and simultaneously providing the correct viscous dissipation [1]. Notwithstanding that, the dynamic free surface BC is presented (equation 1 for the normal stress and 2 for the tangent stress) both for the sake of completeness and in order to recall that when viscosity is relevant, pressure is not in principle zero at the interface and is in general discontinuos.

$$p = 2\mu\, \boldsymbol{n} \cdot \partial \boldsymbol{u} / \partial \boldsymbol{n} \tag{1}$$

$$\boldsymbol{\tau} \cdot \mathbb{D} \cdot \boldsymbol{n} = 0 \tag{2}$$

In these equations $\boldsymbol{n}$ is the normal vector to the fluid domain, $\boldsymbol{\tau}$ is an unitary vector lying on the free surface tangent plane and $\mathbb{D}$ is the rate of strain tensor.

A splitting integration scheme for SPH for high viscosity ratio

S. Litvinov, M. Ellero, X.Y. Hu, N.A. Adams
Lehrstuhl für Aerodynamik und Strömungsmechanik,
Technische Universität München,
85747 Garching, Germany

***Abstract*—In this work we combine and further develop the methods of [1], [2]. The resulting algorithm aims to deal with flow problems relevant to geophysics (simultaneously high dissipation and high viscosity gradients). In particular we cosider the so called "falling block benchmark" [3], which provides a challenging problem for SPH in terms of preserving the "block edges geometry" at high $(10^2$–$10^6)$ viscosity ratios.**

I. Introduction

In our previous work [2] an algorithm to solve a density discontinuity at a phase interface has been developed which satisfies mass conservation exactly. The viscous terms in this formulation allow to model also discontinuity in the viscosity by ensuring at the same time continuity of velocity and shear stress across the phase interface.

In [1] we have proposed an implicit numerical scheme for SPH that allows to increase significantly the time-step size thus performing simulations at significantly larger viscosity.

Gerya and Yuen [4] mention important elements for the modeling realistic situations in lithospheric and mantle dynamics. Most of these challenges come from the fact that transport properties of rocks including viscosity, conductivity vary strongly with chemical composition or mineralogy. In this paper we are dealing with one of such challenges namely high viscosity ratios. We apply the combination of the multiphase [2] and splitting algorithm [1] to model the descent of a stiff object into a medium with a lower viscosity ("falling block benchmark" [3]).

II. Method

Let us assume to deal with an isothermal Newtonian solvent described by the Navier-Stokes equations written in a Lagrangian reference frame

$$\begin{aligned} \frac{d\rho}{dt} &= -\rho\nabla\cdot\mathbf{v}, \\ \frac{d\mathbf{v}}{dt} &= -\frac{1}{\rho}\nabla p + \frac{\eta}{\rho}\nabla^2\mathbf{v}, \end{aligned} \tag{1}$$

where ρ is the material density, $\mathbf{v}$ is the velocity, p is the pressure and η is the dynamic viscosity. The SPH discretization of the Navier-Stokes equations is given by

$$\frac{d\mathbf{r}_i}{dt} = \mathbf{v}_i \tag{2}$$

$$\begin{aligned} \frac{d\mathbf{v}_i}{dt} &= -\frac{1}{m_i}\sum_j\left(\frac{p_i}{\sigma_i^2}+\frac{p_j}{\sigma_j^2}\right)\frac{\partial W_{ij}}{\partial r_{ij}}\mathbf{e}_{ij} \\ &+ \frac{\eta_i\eta_j}{2(\eta_i+\eta_j)m_i}\sum_j\left(\frac{1}{\sigma_i^2}+\frac{1}{\sigma_j^2}\right)\frac{\mathbf{v}_{ij}}{r_{ij}}\frac{\partial W_{ij}}{\partial r_{ij}}. \end{aligned} \tag{3}$$

Here, m_i is the mass of a particle with index i, W_{ij} is a kernel function, σ_i is the inverse of the particle volume, $\mathbf{e}_{ij}$ and r_{ij} are the normalized vector and distance from particle i to particle j, respectively. In order to close these equations, an equation of state for the pressure is given as

$$p = p_0\left(\frac{\rho}{\rho_0}\right)^\gamma + b \tag{4}$$

where p_0, ρ_0, b and γ are parameters which may be chosen based on a scale analysis so that the density variation is less than a given value. When a sufficiently stiff equation of state is used (usually $\gamma = 7$, $b = 0$), penetration between particles is prevented, and an almost incompressible situation is reproduced.

The average density of a particle is

$$\rho_i = m_i/\mathcal{V}_i \tag{5}$$

in which m_i is the mass of a particle. The evolution equation for the particle density used here is

$$\rho_i = m_i\sigma(\mathbf{r}_i) = m_i\sum_j W_{ij} \tag{6}$$

where

$$W_{ij} = W(r_{ij}) = W(\mathbf{r}_i - \mathbf{r}_j) \tag{7}$$

This form conserves mass exactly and is similar to the common SPH density approximation

$$\rho_i = \sum_j m_j W_{ij} \tag{8}$$

The difference is that in the current approximation neighboring particles contribute to the particle density only by affecting the specific volume of particle i. Since there is no mass contribution from neighboring particles Eq. (6) allows for density

A comparative study on the accuracy of SPH schemes in DNS simulation of 2D wall–bounded decaying turbulence

Ashkan Rafiee
School of Mechanical Engineering
The University of Western Australia
Perth, WA 6009, Australia
rafiee@mech.uwa.edu.au

Sharen Cummins
Computational Modelling Group
CSIRO Mathematics Informatics and Statistics
South Clayton, VIC 3168, Australia
sharen.cummins@csiro.au

Murray Rudman
Department of Mechanical and Aerospace Engineering
Monash University
Clayton, VIC 3800, Australia
murray.rudman@monash.edu

Krish Thiagarajan
School of Mechanical Engineering
The University of Western Australia
Perth, WA 6009, Australia
krish.thiagarajan@uwa.edu.au

***Abstract*—In this paper, the accuracy and stability of various SPH formulations are compared in simulating 2D wall–bounded decaying turbulence with no–slip boundaries. Specifically, we compare the performance of traditional weakly compressible SPH (WCSPH) to incompressible SPH (ISPH) and the Godunov-type SPH schemes (GSPH). The flow evolution are shown and the decay rates of kinetic energy for these three SPH formulations are compared with the numerical pseudo-spectral simulations of Clercx [1]. In order to better understand how the solution of the SPH pressure equation affects the accuracy of SPH in simulating turbulent flows, the 1D energy spectrum of various SPH schemes at different times is compared. These comparisons highlight the way in which the kinetic energy is distributed over the multiple scales in various SPH schemes and compare the influence of the pressure solution in reproducing the energy cascade in a 2D wall–bounded turbulent flow.**

I. Introduction

While several turbulence models have recently been proposed for the SPH method [2]–[4], few studies have been performed to assess how accurate the SPH method is in simulating turbulence. Recently, Robinson *et al.* [5] has performed a DNS simulation of 2D wall–bounded decaying turbulence using the standard weakly compressible SPH (WCSPH) method [6] and compared his results with published results of Clercx *et al.* [1]. The study concluded that while the results are qualitatively similar to Clercx's, the standard SPH method leads to significantly higher dissipation of the kinetic energy. The study also concluded that a major source of this energy dissipation is the noisy motion of particles at the kernel length scale [5].

In a study on sloshing flows, Rafiee et. al [7] showed that the incompressible SPH formulation [8] and a Godunov–type SPH scheme [9], [10] results in a smoother flow field than WCSPH [7] and hence less noisy motion of particles. Therefore, the purpose of this study is to compare the accuracy of traditional weakly compressible SPH with incompressible and Godunov–type SPH in simulating 2D wall–bounded decaying turbulence. The outcome of this research will provide information on how well various SPH schemes can model turbulent flow without the implementation of a turbulence model and in particular will demostrate how the solution of the pressure (which results in a noisy or smooth flow field) influences the accuracy of the SPH method in modelling turbulence.

II. Weakly Compressible SPH

In weakly compressible SPH (WCSPH) the pressure is calculated through an equation of state. The most common form of equation of state used in SPH has the form [11], [12]

$$P_i = \frac{\rho_{0_i} {c_i}^2}{\gamma}\left[\left(\frac{\rho_i}{\rho_{0_i}}\right)^{\gamma} - 1\right] \tag{1}$$

where ρ_0 is the reference density and γ is the polytropic constant and is usually set to $\gamma = 7$. In order for density fluctuations to be approximately 1%, the Mach number should be $M \approx 0.1$. To ensure this condition, the speed of sound is set to $c = 10V$, where V is the maximum anticipated velocity inside the flow. The density ρ_i is evolved at each time–step using the continuity equation

$$\frac{d\rho_i}{dt} = D_i = \rho_i \sum_j \frac{m_j}{\rho_j}(\mathbf{u}_i - \mathbf{u}_j)\cdot\nabla W_{ij} \tag{2}$$

and the velocity is evolved using the momentum equation,

$$\frac{d\mathbf{u}_i}{dt} = \mathbf{F}_i = -\sum_j m_j \left(\frac{P_i}{{\rho_i}^2} + \frac{P_j}{{\rho_j}^2} + \Pi_{ij}\right)\nabla_i W_{ij} \tag{3}$$

SPH implicit turbulence modeling

Y.L. Shi, M. Ellero and N.A. Adams
Institute of Aerodynamics and Fluid Mechanics
Technische Universität München
Munich, Germany
yilei.shi@aer.mw.tum.de

***Abstract*—In this paper, the use of Smoothed Particle Hydrodynamics (SPH) as a potential tool for turbulent flows modeling will be discussed. Explicit Sub-Particle-Scale (SPS) turbulence models based on generalization of Smagorinsky, k-ϵ [1,2], alpha models [3] or stochastic models [4] were recently applied to SPH with some success. Following the philosophy of Implicit Large Eddy Simulation (ILES) in grid-based methods [5], in this paper we focus on the possible use of numerical dissipative mechanisms implicitly contained in the SPH scheme to simulate turbulent flows. The presence of atomistic implicit viscosities in a standard SPH formulation was recently observed [6], which are related to the particulate nature of the method. The magnitude of these implicit contributions becomes usually dominant over the input viscosity in the limit of large Reynolds number and could in principle interfere with an ad-hoc introduced explicit SPH-SPS model. In the context of ILES it has been shown that the numerical dissipation can be beneficial in some cases and can act as an implicit sub grid scale model under a proper choice of the discretization parameters. The goal of this paper is to analyze the performance of a standard version of SPH in a turbulent framework without any explicit SPS model. In particular, an energy spectrum analysis will be presented and the relation with the classical Kolmogorov -5/3 law will be discussed. Finally, third order structure function and vorticity field will be investigated.**

I. Introduction

Smoothed Particle Hydrodynamics (SPH) is a meshless, Lagrangian method. Due to the random nature of the Lagrangian particle trajectories, the method might be able to capture some properties of hydrodynamic turbulence. It is therefore interesting to investigate its performance under standard turbulent flow conditions.

The idea to use an implicit form of Sub-Grid Scale (SGS) modeling in the context of the 'vortex method' was pioneered by Cottet [7] who in 1996 showed that it is possible to correct the governing equations with additional terms in order to produce a desired artificial dissipation. Preliminary studies of the truncation error indicate that such a theoretical analysis can be performed also in the framework of SPH even if the subject has not been considered yet. Numerical indication that SPH can implicitly realize a SGS model was proven by the existence of an artificial viscosity inherently present in the numerical formalism. This anomalous behaviour of SPH was first identified by Hoover and his collaborators by studying the microscopic intrinsic shear viscosity of an Euler fluid in the middle 90s [8]. By invoking the isomorphism linking the SPH equations describing an inviscid fluid with the ones governing the motion of an atomistic Lucy fluid, they performed Non-Equilibrium Molecular Dynamics (NEMD) simulations of the latter to deduce the spurious transport coefficients of SPH. The presence of two viscosities in a thermostated Lucy fluid, analogous to the microscopic kinetic and potential ones, was reported. Recently, Ellero et. al. made a systematic study of the two spurious viscosities [6]. The results suggest that the the mechanism of the kinetic dissipation, which has a similar formulation to the Reynolds stress tensor and dominates at high Reynolds number flow, might act in principle as an implicit turbulence model. The goal of this work is to analyze the performance of a standard version of SPH in simulating decaying turbulence.

The paper is organized as follows. In section II the basic SPH formulation used in this work is reviewed. In section III, two kind of interpolation methods are introduced (e.g. SPH interpolation and Remeshed interpolation) to map particle properties on a grid. Errors introduced by the two interpolation schemes are compared by performing a spectral analysis of a prescribed velocity field. Results of our simulation in the case of decaying turbulence are presented in section IV where the energy spectrum, third-order structure function and vorticity field are studied in detail. Finally, conclusions are given in Sec. V.

II. SPH method

A. Equations of motion

The basic Navier-Stokes equations on a Lagrangian framework can be formulated as follows:

$$\frac{d\mathbf{v}}{dt} = -\frac{\nabla p}{\rho} + \nu\nabla^2\mathbf{v} + \mathbf{F} \tag{1}$$

where $\mathbf{v}$ and ρ are respectively, the fluid velocity and density, p is the pressure, $\nu = \eta/\rho$ is the kinematic viscosity and $\mathbf{F}$ is an external body force. A simple equation of state relating pressure to density is for example

$$p = \frac{c^2\rho_0}{\gamma}\left[\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right] \tag{2}$$

where ρ_0 is the reference density, c the numerical speed of sound and γ the ratio of specific heats, $\gamma = 7$ for liquid.

B. Density equation

In SPH, the mass density of particle i is evaluated as

$$\rho_i = \sum_{j=1}^{N} m_j W_{ij} \tag{3}$$

Simulation of 2D turbulence in a no-slip box using SPH

A. Valizadeh and J.J. 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 study turbulence in two dimensional boxes with no-slip boundaries using the particle method SPH. A set of gaussian vortices is used as the initial turbulence and the boundaries are specified by boundary force particles. In the case of fixed boxes we recover the results of Clercx and his colleagues, including the change in the fluid's angular momentum. When the box is allowed to rotate freely so that the total angular momentum of box and fluid is constant, the change in the angular momentum of the fluid is very much smaller than is the case for the fixed box. We also simulate the behaviour of the turbulence when the box is forced to rotate with small and large Rossby number.**

I. Introduction

Smoothed particle hydrodynamics is now widely used in computational fluid dynamics especially in problems involving breaking waves, and free surfaces disrupted by the impact of rigid bodies. In many of these systems the flows are initially laminar but develop turbulence as they evolve. In order to have confidence in the predicted evolution of the flow it is necessary to determine how accurately SPH simulates turbulence. The simplest test case is that of two dimensional turbulence in a bounded domain with no-slip boundaries. Not only does this mimic aspects of confined flows, but it has been the subject of comprehensive laboratory and numerical study by Clercx and van Heijst and their colleagues. They have studied the spectra [1], the energy and enstrophy decay [5], and the vorticity distribution [4]. A remarkable phenomenon that has emerged from these calculations is the spontaneous spin-up, with either positive or negative angular momentum, and the sensitivity of the vorticity to the no-slip boundary [21].

The application of SPH to the simulation of decaying and driven turbulence is in its infancy. However, turbulence in a fixed square box in two dimensions, has been simulated [18], [19] finding satisfactory agreement with the results of [2], and establishing some features of the convergence. In the present paper the study of SPH simulation of turbulence will be extended. First, we simulate turbulence within boundaries defined by boundary force particles unlike the layers of fluid particles used by [18], and [19]. The reason for this is that boundary force particles are often used in SPH simulations [14] and the results indicate the sensitivity of the turbulence simulation is to the precise modeling of the boundaries. Second, we make a detailed study of the convergence of the energy decay with time. Third we study decaying turbulence when the box containing the fluid is allowed rotate under the surface stresses produced by the fluid. In this case the total angular momentum of the system of box and fluid is conserved. Fourth we study how the turbulence evolves when the fluid and box are first in rigid rotation, then the turbulence is initiated, after which the combined system is allowed to rotate freely. This problem is related to turbulence in the earth's atmosphere, but a more complete discussion along the lines of the β-plane study of [9], will not be attempted.

II. The equations of motion in two dimensions

We assume an incompressible fluid is moving in two dimensions within a square boundary with no-slip boundary conditions. It is convenient, especially when we give the boundary a mass and allow it to rotate, to refer to the boundary as a box. The acceleration equation for the fluid is

$$\frac{d\mathbf{v}}{dt} = \frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v}\cdot\nabla)\mathbf{v} = -\frac{\nabla P}{\rho} + \frac{\mu}{\rho}\nabla^2\mathbf{v}, \qquad (1)$$

where μ is the dynamic viscosity, P, ρ, and V are pressure, density and velocity of fluid, respectively.

In this paper we also consider turbulence in a two dimensional fluid contained within a no-slip, square boundary rigidly rotating with angular velocity $\Omega\hat{\mathbf{z}}$, and $\hat{\mathbf{z}}$ is a unit vector perpendicular to the plane of the fluid. In this case it is useful to consider the equations of motion in a frame rotating with angular velocity Ω. For this purpose, we let $\mathbf{v}$ denote the velocity in this rotating frame, and $\omega\hat{\mathbf{z}}$ its vorticity. The acceleration equation is then

$$\frac{d\mathbf{v}}{dt} = -2\Omega\hat{\mathbf{z}}\times\mathbf{v} + \Omega^2\mathbf{r} - \frac{\nabla P}{\rho} + \frac{\mu}{\rho}\nabla^2\mathbf{v}. \qquad (2)$$

where d/dt denotes the derivative following the motion in the rotating frame. From this equation the rate of change with time of the energy and the enstrophy is the same as in the non rotating frame. Since these equations form the basis of Batchelor's argument for two dimensional turbulence we believe that the argument also applies to turbulence in the rotating frame.

We also study the case where the boundary is given a mass and a moment of inertia (we then call it a box) and

Simulation of particulate flows using coupled SPH and DEM simulations

T. Breinlinger, A. Wonisch, T. Kraft
Fraunhofer Institute for Mechanics of Materials
Freiburg, Germany
thomas.breinlinger@iwm.fraunhofer.de

***Abstract* — For several processes dealing with complex fluids neither particulate nor fluidic influence is negligible. Thus, for analyzing such problems purely solving the fluid flow equations is not sufficient. In this study we demonstrate the coupling of Smoothed Particle Hydrodynamics (SPH) used to describe fluid motion and the Discrete Element Method (DEM) representing solid particles suspended in the fluid phase. Our method is compared with various results from literature and its application on a real problem is demonstrated.**

I. INTRODUCTION

The production and processing of granular materials in fluid flows is common in many industrial applications of powder production and processing. With a suitable simulation scheme in-depth knowledge of the underlying flow conditions can be obtained. However, as these processes often deal with complex fluids where neither particulate nor fluidic influence is negligible coupling of the two mechanisms is a prerequisite.

In literature many reports about the simulation of particles suspended in gas or fluid may be found. Besides neglecting the solid phase, which may be a reasonable approach for very small solid volume fractions, the most common technique is coupling classic mesh based computational fluid dynamics solvers (CFD) with various kinds of treatment of the solid particles.

Benyahia et al. [1] investigate a circulating fluidized bed by using CFD to solve for the gas flow and the kinetic gas theory for treatment of the disperse phase. Xu and Yu [2] also analyze fluidized beds however they combine locally averaged CFD with the discrete element method (DEM) which keeps track of the individual granules by solving Newton's second law of motion. To reduce the computational costs emerging from the treatment of every single particle, Kloss et al. [3] combined CFD with a discrete phase model (DPM) to solve for the granular flow which was complemented to the DEM describing the single particles motions on a smaller scale. Using spatial domain decomposition, all three models were computed simultaneously.

Being a fully lagrangian approach for solving Navier-Stokes equations, smoothed particle hydrodynamics (SPH) [4], [5] promises several advantages for the treatment of particulate suspensions. It offers the capability of dealing with complex geometry and allows an intrinsic straightforward coupling with other lagrangian methods describing the disperse phase.

Monaghan proposed [5] and improved [7] a SPH model for treatment of interpenetrating fluids by using an improved drag technique and used the model to study dusty gas dynamics. The first implementation coupling SPH with DEM known to the authors was reported by Gao and Herbst [8], who modified the multiphase model proposed by Monaghan and added coupling terms to account for the interaction of fluid and solid phase.

In this work we improved the coupled SPH-DEM scheme to work on different spatial length scales by including additional force laws [9], [10]. For macroscopic simulation of pneumatic conveying of granular media we add forces to account for Saffman and Magnus effects [11]. On the microscopic scale we introduce intermolecular forces dominating the behaviour of dense suspensions. Details of our model together with a description of the added forces will be given in the next section. In the final part of the paper some applications of the model are presented

II. THE MODEL

A. *Fluid phase*

The motion of a fluid phase influenced by solid particles can be modelled by the following set of equations describing the conservation of mass (1) and momentum (2) [8]:

$$\frac{\partial \hat{\rho}}{\partial t} = -\hat{\rho}\nabla \cdot \vec{v} \tag{1}$$

$$\frac{\partial \vec{v}}{\partial t} = -\frac{\nabla P}{\rho} + \frac{\nabla \cdot T}{\hat{\rho}} + \vec{g} + \vec{f}_c, \tag{2}$$

Mesoscale fluid-particle interaction using two-way coupled SPH and the Discrete Element Method

Martin Robinson
Multiscale Mechanics
University of Twente
Enschede, Netherlands
m.j.robinson@ctw.utwente.nl

Marco Ramaioli
Nestle Research Center
Lausanne, Switzerland
Marco.Ramaioli@rdls.nestle.com

***Abstract*—We present a meshless simulation method for multiphase fluid-particle systems using Smoothed Particle Hydrodynamics (SPH) and the Discrete Element Method (DEM). Rather than fully resolving the interstitial fluid, which is often infeasible, we use a unresolved fluid model based on the locally averaged Navier Stokes equations. A variable-h SPH formulation is used to calculate the fluid phase, where the density of each particle is proportional to the local porosity. The SPH-DEM method is validated using simulations of single and multiple particle sedimentation in a 3D water column. The velocity and terminal velocity for the single particle compares well with the analytical solution, provided that the minimum resolution of the fluid phase is much larger than the solid particle diameter. For fluid resolutions close to the particle diameter there is no longer a clear separation between the resolved and unresolved fluid scales, leading to an artificial increase in the particles' terminal velocity. In the multiple particle sedimentation simulation, the two-way coupling between the phases permits the growth of a Rayleigh-Taylor instability. The growth rate of the instability in the linear regime is calculated and compares well with theory.**

I. Introduction

Fluid-particle systems are ubiquitous in nature and industry. Sediment transport and erosion are important in many environmental studies and the interaction between particles and interstitial fluid affects the rheology of avalanches, slurry flows and soils. In industry, the efficiency of a fluidised bed process (e.g. Fluidized Catalytic Cracking) is completely determined by the complex two-way interaction between the injected gas flow and the solid granular material. The dispersion of solid particles in a fluid is also of broad industrial relevance in the food, chemical and painting industries.

The length-scale of interest determines the method of simulation for fluid-particle systems. For very small scale processes it is feasible to fully resolve the interstitial fluid between the particles (see [1]–[3] for a few examples of particle or pore-scale SPH simulations). However, for many applications the dynamics of interest occur over length scales much greater than the particle diameter and it becomes necessary to use unresolved, or mesoscale, fluid simulations.

Fixed pore flow simulations using SPH for the (unresolved) fluid phase have been described by Li et al. [4] and Jiang et al. [5], but these do not allow for the motion and collision of solid grains. Cleary et al. [6] and Fernandez et al. [7] simulate slurry flow at the mesoscale using SPH and DEM in SAG mills and through industrial banana screens, but only perform a one-way coupling between the solid and fluid phases.

The SPH-DEM model presented in this paper can be used for both one and two-way coupling and is suitable for both dilute and dense particle systems. It is based on the locally averaged Navier-Stokes (AVNS) equations that were first derived by Anderson and Jackson in the sixties [8], and have been used with great success to model the complex fluid-particle interactions occurring in industrial fluidized beds [9]. Anderson and Jackson defined a smoothing operator identical to that used in SPH and used it to reformulate the NS equations in terms of smoothed variables and a local porosity field (porosity refers to the fraction of fluid in a given volume). Given its theoretical basis in kernel interpolation, it is natural to consider the use of the SPH method to solve the AVNS equations, coupled with a DEM model for the solid phase. This results in a purely particle-based solution method and therefore enjoys the flexibility that is inherent in these methods. In particular, the SPH-DEM model described in this paper is well suited for applications involving a free surface, including (but not limited to) debris flows, avalanches, landslides, sediment transport or erosion in rivers and beaches, slurry transport in industrial processes (e.g. SAG mills) and liquid-powder mixing in the food processing industry.

The next section describes the AVNS equations and the SPH and DEM models for the fluid and solid phases. The remainder of the paper then describes SPH-DEM simulations of single and multiple particle sedimentation in a 3D water column. These results are compared against analytical solutions in order to validate the proposed model.

II. The Locally Averaged Navier-Stokes Equations

Most models of the unresolved fluid phase are based on the locally averaged Navier-Stokes equations derived by Anderson and Jackson [8]. Anderson and Jackson defined a local averaging based on a smoothing function $g(r)$ with identical properties to the normal SPH kernel. The local average of any fluid variable $a'(x)$ can be obtained by convolution with the smoothing function

Three-dimensional validation of a SPH-FEM coupling method

G. Fourey, D. Le Touzé, and B. Alessandrini
Fluid Mechanics Lab., Ecole Centrale Nantes / CNRS
Nantes, France
guillaume.fourey@ec-nantes.fr

***Abstract*—We presented at the fifth SPHERIC Workshop a two-dimensional Fluid-Structure Interaction (FSI) coupling between the Smoothed Particle Hydrodynamics (SPH) method for the fluid, and the Finite Element Method (FEM) for the solid [1]. First validations were shown. With this coupling we take advantage of both methods, the SPH method capability to handle large deformations of fluid domains, and the established FEM to predict solid behaviors that undergo time-variable pressure loads. In addition, no specific contact algorithm is needed to avoid material interpenetration. This leads to a relatively easy implementation of the whole coupling. In the present paper we present further validation of the coupling and its extension to three-dimensional situations. In particular the global conservation of energy throughout the coupling is carefully monitored and analysed. To this purpose the different energies (kinetic, potential and internal) are expressed and monitored in time, both in the SPH method for the fluid, and in the FEM for the solid. The sum of these different energies shall keep constant in time in absence of dissipation. Three-dimensional validations of the SPH-FEM coupling developed are then presented, in comparison to experimental data. The validation strategy is first to compare to already performed 2D validations, as in the case of a dam-break through an elastic gate [3]. Then the 3D model is applied to realistic complex cases where 3D effects can not be neglected.**

I. Introduction

In [1] an explicit parallel SPH-FEM coupling method was presented and first validation was performed by studying the impact at high velocity of an elastic solid on the free surface. Despite the use of a weak coupling strategy, the results obtained were in good agreement with analytical solution provided in [2]. One advantage of the coupling developed here is that no specific treatment such as sub-iterations is done to verify physical conditions at the fluid-structure interface. Even if good agreements between simulations and experimental or analytical data are found, it remained necessary to assess the coupling consistency. One way to do so is to monitor time conservation of the total energy of the whole system. In the present paper the evolution of the different flow energies are first analysed on simple fluid simulations without the presence of a structure. This permits to discuss of energy preservation in the SPH scheme stabilized with a Riemann solver. Then this monitoring is applied to FSI situations. In particular, another two-dimensional test case is presented in detail with evaluation of fluid and solid energies. It concerns the escape of a water column through an elastic gate [3]. Finally, first 3D validation of the coupling is performed, and the FSI model applicability to realistic complex situations is performed on an industrial test case.

II. SPH solver

In this section, the system of equations and the numerical scheme used in the SPH-Flow solver are briefly described. We then focus on the evaluation of kinetic, potential and internal energies.

A. Governing Equations

Locally, we have the following conservation equations of mass and momentum.

$$\frac{\partial \rho}{\partial t} = -\vec{\nabla}.(\rho\vec{v}) \quad (1)$$

$$\frac{\partial \rho\vec{v}}{\partial t} = -\vec{\nabla}.(\rho\vec{v}\otimes\vec{v} + pI_d) + \rho\vec{g} \quad (2)$$

The Tait's equation of state relating pressure to density of the barotropic fluid closes the system above.

$$p = \frac{\rho_0 a_0^2}{7}\left[\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right] \quad (3)$$

B. Discrete scheme

In concrete terms, the field is described by a set of particles. The discrete scheme is written using the Lagrangian symmetrized form of equations, leading to, for the space discretization part:

$$\frac{d\vec{x_i}}{dt} = \vec{v_i} \quad (4)$$

$$\frac{d\omega_i}{dt} = -\omega_i \sum_j \omega_j \left(\vec{v_i} - \vec{v_j}\right).\vec{\nabla}W_{ij} \quad (5)$$

and for Euler equations:

$$\frac{d\omega_i\rho_i}{dt} = -\omega_i \sum_j \omega_j 2\rho_e(\vec{v_e} - \vec{v}(x_{ij})).\vec{\nabla}W_{ij} \quad (6)$$

$$\frac{d\omega_i\rho_i\vec{v_i}}{dt} = \omega_i\rho_i\vec{g} - \omega_i \sum_j \omega_j 2[\rho_e\vec{v_e}\otimes(\vec{v_e} - \vec{v}(x_{ij})) + p_e I_d].\vec{\nabla}W_{ij} \quad (7)$$

where ρ_e and $\vec{v_e}$ are the solutions of the Riemann problem solved for each interaction at the interface x_{ij}. Velocity of this interface is given by (8).

$$\vec{v}(x_{ij}) = \frac{1}{2}\left(\vec{v_i} + \vec{v_j}\right) \quad (8)$$

Validation of a conservative variable-resolution SPH scheme including ∇h terms

A. Marsh, G. Oger, D. Le Touzé
Fluid Mechanics Laboratory
Ecole Centrale Nantes / CNRS
Nantes, France

D. Guibert
Fluid Mechanics and Acoustics Laboratory
Ecole Centrale Lyon / CNRS
Ecully, France

Many of the hydrodynamic applications suited to SPH consist of large fluid volumes, requiring a high level of detail to be captured in specific local areas of interest. In order to efficiently achieve this, a variable resolution scheme should be used to minimise the total number of particles needed to represent the fluid volume. The SPH method allows for the simple implementation of such a scheme, although additional terms taking the gradient of smoothing length into consideration (∇h terms) must be included in the equations of motion to remain conservative.

The affect of the addition of the ∇h terms within an ALE formulism is analysed in this paper in two dimensions. This new formulation is then validated by simple hydrostatic analysis, and the benchmark test case of a rigid wedge impacting a free surface. ∇h terms are found to be corrective in nature, improving the schemes properties of conservation and stability.

I. Introduction

In order to take full advantage of the Lagrangian nature of SPH, the ability to accurately treat variations in smoothing length is needed. To achieve this, additional terms are required in the particle equations of motion to remain conservative, taking the gradient of h into consideration (∇h terms) [1-3]. It has been reported that these terms can be neglected in practice, providing steep density gradients are not present in the flow [4,5]. However, it will be shown in this work that issues can arise in maintaining the prescribed particle positions under simple hydrostatic conditions.

In this paper, a new method of introducing the ∇h terms into the equations of motion is presented. The objective is to treat a system of particles with prescribed smoothing lengths that remain unchanged over time $\left(\partial h/\partial t = 0\right)$, as opposed to smoothing lengths that evolve to maintain a constant number of neighbours within the local support [6,7].

The new formulation is subject to simple hydrostatic analysis, and then the benchmark test of a rigid wedge impacting a free surface. The ∇h terms are found to provide a conservative description, allowing the physical propagation of acoustic behaviour over variations in h, in a flow with steep pressure gradients.

II. SPH Scheme

The ALE formulation implemented in SPH-Flow is used in this work. The formulation is shown below:

$$\begin{cases} \dfrac{d\mathbf{x}_i}{dt} = \mathbf{v}_i \\ \dfrac{dw_i}{dt} = w_i \sum_j w_j \left(\mathbf{v}_j^0 - \mathbf{v}_i^0\right).\nabla \tilde{W} \\ \dfrac{dw_i\phi_i}{dt} + w_i \sum_j w_j \begin{pmatrix} F_E\left(\phi_i\right) - \mathbf{v}_i^0\phi_i \\ +F_E\left(\phi_j\right) - \mathbf{v}_j^0\phi_j \end{pmatrix}.\nabla \tilde{W} = S_i \end{cases} ; \quad (1)$$

where $\mathbf{x}_i$, $\mathbf{v}_i$, w_i are respectively, the position, velocity, and volume of the considered particle. v_i^0 is the transport field of the ALE approach. ϕ is the vector of conservative variables $\left(\rho_i, \rho_i v_i\right)^t$. $F_E(\phi_i)$ is the flux of conservative variables $\left(\rho_i v_i, \rho_i v_i \otimes v_i + p_i I_d\right)^t$, evaluated by the use of a Riemann solver. $\nabla \tilde{W}$ is the gradient of the smoothing function, having the gather formulation of $\nabla W\left(\mathbf{r}_{ij}, h_i\right)$ in cases when h does not vary.

In cases where h does vary, such a formulation of the smoothing function produces unsymmetrical forces between interacting particles [9]. In extreme cases, it is possible that particle j recognises particle i as a neighbour, but i does not recognise j. An example of this is shown in Figure 1.

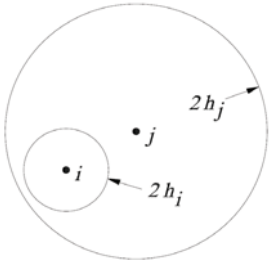

Figure 1. Example of non-reciprocal interaction.

A dynamic particle coalescing and splitting scheme for SPH

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 dynamic refinement algorithm for Smoothed Particle Hydrodynamics (SPH) is presented. The variable resolution in SPH is achieved using both the particle splitting and a novel conservative particle coalescing procedure. The particle splitting procedure is able to increase the resolution by increasing the number of particles in the domain, and conversely, the resolution is reduced by coalescing particles (to improve the efficiency). This is accomplished while conserving both mass and momentum, and calculating the smoothing length of the new particles by minimizing the density error. The general applicability of the new dynamic particle refinement procedure is assessed by applying it to shallow water equations (SWEs) and testing the numerical scheme against different reference solutions. In the real-life test case the particle coalescing procedure leads to a reduction of the computational time by a factor of 15.**

I. Introduction

In order to get robust and reliable results in computational fluid dynamics, it is important to simulate accurately relatively small-scale features such as recirculation zones, shock waves etc. In classical Eulerian models this is usually achieved using adaptive structured [5], [9], [15] or unstructured grids [8]. In meshfree numerical schemes there have been some early attempts to introduce variable resolution by either remeshing, particle splitting and particle insertion/removal techniques [3], [7], [17], or through variable smoothing lengths with dynamic particle splitting according to pre-defined criteria [4], [18]. However, particle splitting leads to an ever increasing number of particles in the numerical domain without the ability to coarsen resolution when desirable to improve efficiency during a simulation. In this work, we will present a 2-D particle refinement scheme that includes particle splitting and coalescing. We will be using the shallow water equations (SWEs) which are used to model different natural phenomena such as tidal flow, river flooding, tsunami, dam break etc. The technique presented herein is equally applicable to the Navier-Stokes equations.

This paper is organized as follows: in section II the main features of the SPH-SWEs numerical solver based on the variational approach are briefly recalled, In section III the particle splitting procedure for variable smoothing length schemes is outlined. In Section IV the novel methodology for particle coalescing is presented, and finally in Section V the particle refinement is tested against different reference solutions including a real life test case.

II. SPH Shallow Water Equations solver

The shallow water equations can be written in Lagrangian form as

$$\begin{array}{c} \frac{\partial d}{\partial t} = \nabla \cdot (d\mathbf{v}) \\ \frac{\partial (\mathbf{v}d)}{\partial t} + \nabla \cdot (d\mathbf{v}) = -gd\left(d+b\right) + gd\mathbf{S}_f \end{array} \tag{1}$$

where d is depth, $\mathbf{v}$ is depth-averaged velocity, t is time, b is the bottom elevation, g is the acceleration due to gravity and $\mathbf{S}_f$ is the bed friction source term. The SWEs are formally identical to the Euler equations if we re-define the density ρ as the mass of fluid per unit of area in a 2-D domain; with this definition of ρ we have $\rho = \rho_w d$, where ρ_w is the constant (conventional) density. The density ρ_i of a particle i can vary considerably during a simulation; therefore an SPH scheme with variable smoothing length h in time and space is used to keep the number of neighbour particles roughly constant during the processes of water inundation and retreat. Using these definitions the SWEs in Lagrangian form can be rewritten as:

$$\begin{array}{c} \frac{d\rho}{dt} = -\rho \nabla \cdot \mathbf{v} \\ \frac{d\mathbf{v}}{dt} = -\frac{\rho}{\rho_w} \nabla \rho + g\left(\nabla b + \mathbf{S}_f\right) \end{array} \tag{2}$$

The particle positions and velocities are integrated in time by means of a leap-frog scheme with a Lax-Friedrichs term keeping the solution stable even in the presence of shock waves where a MUSCL reconstruction is applied to increase the accuracy of the SPH interpolation [18]. The closed boundaries (walls) are simulated using the Modified Virtual Boundary Particle method [19], whereas the open boundaries are introduced using the same formulation described in [20].

SPH Modelling of Water/Soil-Flows using a Variable-Resolution Scheme

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

***Abstract*—The paper reports on SPH simulations of water/soil-flows using a variable resolution scheme. The research aims at harbour-flow problems. Emphasis is given to sediment scouring induced by an unberthing ship. The related multi-physics simulations usually involve complex geometries embedded in large computational domains. Important aspects are often concerned with the analysis of subtle details in a confined region along the dynamic fluid/soil-interface. In such cases, the use of traditional homogeneous particle-resolution techniques is afflicted with prohibitive computational costs. The paper aims to outline a viable route for a variable-resolution approach. The strategy is based on dynamic particle properties (mass, kernel length, initial spacing) within a fixed amount of particles framework. The variation of properties solely depends on the current particle position. Related non-solenoidal field effects and the influence of spatial variations of the kernel length on the kernel-function gradient are considered in the computational procedure. Reported results refer to illustrative hydrostatic tests and travelling waves, water-impact studies as well as transverse-thruster induced sediment-scouring simulations for unberthing ships. The study reveals that the strategy supports a significant reduction of particles without compromising the predictive accuracy and features a striking algorithmic simplicity. Moreover, results display the inadequacy of 2D simulation models and thus highlight the importance of variable-resolution techniques for 3D simulations.**

I. Introduction

Sediment scours represent undesirable phenomena in ocean and harbour engineering. They can be of several meters in depth and significantly weaken the structural support of e.g. subsea pipelines, platform legs or wharves. Cost-intensive counter measures have to be carried out to guarantee a safe operation of such facilities. These measures are usually based upon a rather weak background knowledge. Since laboratory experiments suffer from significant scale-effects, multi-physics SPH simulations can help to analyse the scour-formation process and to understand the interaction between water, soil, structures and scour protection. Such simulations need to be performed in large, three-dimensional domains, in which small areas of interest often demand fine resolution. Container ship lengths of about 300 m, docking in approx. 300 m wide basins, are very common in harbours like Hamburg or Rotterdam. Assuming a water depth of 16.5 m and a typical ship draught of approx. 13.5 m, the distance between ship bottom and harbour bed is rather small compared to the overall size of the domain.

Since scours can be induced by the ship's transverse thruster, propellers and hull shoulders, a simulation of the whole configuration is necessary. A typical diameter of the thruster for such container ships is around 3 m. Assuming that at least 10 particles per diameter are needed to discretize the thruster, the whole domain would easily come up to 100 Mio. particles with an uniform resolution. Such particle numbers are clearly unfeasible, even when massively-parallel procedures are employed.

The present research aims to advance the modelling capabilities of the massively-parallel SPH-code GADGET-$^{H}2^{O}$ [13] to compute engineering problems which involve the interaction of fluid, soil and suspension layers with nonuniform particle resolutions. The GADGET-$^{H}2^{O}$-procedure is a modification of Springel's [12] cosmological TreeSPH-Code GADGET-2. The approach offers a linear speed-up for several hundred CPU-cores when applied to hydrodynamic flow simulations [13] and has successfully been applied to complex maritime applications [14]–[16]. The suggested zonal resolution dynamically modifies the particle's mass and kernel length according to the current particle location and is thus labelled "Eulerian variable-resolution approach". Non-solenoidal effects are captured by supplementary source terms to ensure the momentum and density conservation. The strategy allows a significant reduction of particles towards the far field. Moreover, the local inhomogeneity of particle properties, often displayed by Lagrangian variable-resolution strategies, remains small and can be controlled.

The remainder of the paper is structured as follows: In section II, the employed governing equations and their respective finite approximations are described, followed by section III where the performance of the variable-resolution approach is validated. Section IV is devoted to a full-scale 3D application. 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 to the focal particle whereas the subscript

Particle initialization through a novel packing algorithm

B. Bouscasse, M. Antuono, S. Marrone
CNR-INSEAN (The Italian Ship Model Basin)
Rome, Italy

A. Colagrossi
CNR-INSEAN (The Italian Ship Model Basin), Rome, Italy
CESOS (Centre of Excellence for Ship and Ocean Structures)
NTNU, Trondheim, Norway

***Abstract*—A novel particle packing algorithm has been derived basing on a simplified standard SPH scheme and is used to initialize the particle distribution for generic SPH solvers. Using some intrinsic features of the SPH schemes, the proposed algorithm leads to a final particle distribution that is very stable and is not affected by a further resettlement during the early stages of the evolution. Moreover, the computational costs maintain very low since the final configuration is attained very quickly. Finally, a strong point of the packing algorithm is that it can be easily derived using whatever SPH scheme.**

I. Introduction

The matter of how initialize the particle positions in the SPH schemes plays a relevant role. Indeed, particles which are not initially set in "equilibrium" positions may resettle giving rise to spurious motions which can strongly affect the fluid evolution. Hereinafter we refer to an equilibrium configuration as the set of particle positions which, under static conditions, does not lead to particle resettlement.

Unfortunately, apart from few cases characterized by simple geometries, the equilibrium configuration is not known "a priori". This is a major issue since the generation of spurious currents/vorticity is particularly strong in presence of complex solid boundary profiles (i.e. corners, bended bodies, etc.).

In the SPH literature some ad-hoc procedures have been proposed to reduce particle resettlement. The simplest one consists in starting the SPH simulation using a high numerical viscosity and leaving a long enough time to make particle self-resettle in equilibrium positions (see, for example, Monaghan [1]). The actual SPH simulation (that is, the simulation with the correct viscosity and the desired initial conditions) starts after particles have reached an equilibrium configuration. Unfortunately, the attainment of a stable configuration can require a very long evolution, this leading to a large increase of computational costs. Moreover, the high viscosity used for particle initialization does not exclude that a further resettlement occurs when the actual simulation is started with smaller values of the viscosity.

In the SPH framework, the first attempt to define a proper algorithm for particle initialization is due to Oger et al. [2] who adapted the Bubble method described in Shimada [3] to SPH solvers. This algorithm is based on the use of Van der Waals-like forces to place particles throughout the fluid domain. This method proves to be quite fast, applies to general geometries and provides a regular particle distribution. One of the weak points is that the particle positions obtained through the Bubble algorithm may be not perfectly compatible with the SPH static solution leading to a further resettlement.

Then, the key point to build a robust packing algorithm relies on the capability of providing a regular particle distribution which is compatible with the SPH scheme, that is, that satisfies the static conditions when the SPH scheme is used. To this purpose a novel packing algorithm has been derived taking advantage of some intrinsic features of the SPH schemes. Thanks to this, the proposed method allows the attainment of a regular particle distribution compatible with the static solution. Further, it can be easily derived starting from whatever SPH solvers and applies to weakly-compressible or incompressible SPH schemes as well.

The paper is organized as follows: Section §II gives an insight of the constitutive features which are used to build the packing algorithm. Section §III describes the proposed algorithm and highlights some interesting aspects about its Lagrangian structure. Finally, Section IV provides a broad range of numerical test cases which prove the packing algorithm to be fast, robust and reliable also for complex geometrical configurations.

II. Some intrinsic features of the SPH

In the present paper we adopt the standard SPH scheme:

$$\begin{cases} \dfrac{D\rho_i}{Dt} = -\rho_i \displaystyle\sum_j (\boldsymbol{u}_j - \boldsymbol{u}_i)\cdot \nabla_i W_{ij}\, V_j \\ \dfrac{D\boldsymbol{u}_i}{Dt} = \boldsymbol{g} - \dfrac{1}{\rho_i}\displaystyle\sum_j (p_j + p_i)\nabla_i W_{ij}\, V_j + \boldsymbol{T}_i^{(v)}, \\ p_i = F(\rho_i), \\ \dfrac{D\boldsymbol{r}_i}{Dt} = \boldsymbol{u}_i, \end{cases} \tag{II.1}$$

As usual, the subscripts indicate the quantities associated with the i-th and j-th fluid particle. In the specific, V_i is the particle volume, $\rho_i = m_i/V_i$ and m_i is the particle mass. The term $\boldsymbol{T}_i^{(v)}$ indicates an artificial viscous force per unit of mass. This term is generally implemented in the SPH schemes for stability reasons (see, for example, [4]). In the present work, we use the artificial viscous term proposed by Monaghan and Gingold

Development of a new pre-processing tool for SPH models with complex geometries

J.M. Domínguez, A.J.C. Crespo, A. Barreiro and M. Gómez-Gesteira
EPHYSLAB Environmental Physics Laboratory
Universidad de Vigo
Ourense, Spain
jmdominguez@uvigo.es

A. Mayrhofer
School of Mechanical, Aerospace and Civil Engineering
University of Manchester
Manchester, U.K.

***Abstract*— In order to obtain realistic results with SPH it is pertinent that the initial geometry is as close to the real-engineering problem as possible. A new pre-processing tool has been developed to deal with complex geometries. The new code named GenCase is able to create particles starting from predefined geometrical shapes or by importing external files with geometries generated by dedicated design software. Thus, real-life engineering problems can be easily simulated with SPH independently of how complicated the initial configuration is. GenCase will allow researchers and engineers to carry out more viable simulations to study real-world problems.**

I. INTRODUCTION

SPH is too expensive in terms of computation time. Therefore, simulations that are carried out at reasonable runtimes may not involve large number of particles and consequently the geometry of the case is also limited.

During the last years, performing simulations with millions of particle at affordable computation time is possible thanks to development and progress of computational technology and the use of high performance computing, such as clusters of Central Processing Units (CPUs) or parallel programming in Graphics Processor Units (GPUs). Due to this capability there is a need to develop tools that generate experiments with geometries that represent real-world scenarios.

However, the process of generating the geometry of an experiment based on particles is not trivial and can give rise to a significant computational cost. Generating the initial configuration of particles for a SPH simulation requires filling volumes of irregular shapes using particles that must be spaced equidistant. Depending on the treatment of the boundary conditions, computation of the normal vectors of the boundary points might be required.

The SPHysics code (www.sphysics.org) uses a pre-processing code named SPHysicsGen that creates the initial configuration of the case. This software, implemented in Fortran, is based on a simple algorithm that allows users to create very simple predefined shapes such as parallelepipedons. However, complex geometries cannot be created using the available code capabilities of SPHysicsGen. To expand the possibilities of SPHysicsGen, an add-on was presented in [1] that centres around the open-source software Blender. Meshes that can either be created or imported in Blender are converted via a custom Python script. The output is then read by SPHysicsGen where specific properties of the simulation can be defined.

Working in the same direction, a code named GenCase was developed. GenCase is a tool implemented in C++ that works independently without the need for other design software. This code combines the simplicity of defining the case using basic geometrical shapes with the capacity of including 3D models. Thus, starting from the case description and the 3D external objects, the code is able to generate very complex geometries using millions of particles not only in an easy way but also almost instantaneously.

GenCase is a pre-processing tool that can be found included in the DualSPHysics project (can be downloaded for free from www.dual.sphysics.org). This new SPH solver was designed to study real engineering problems and can be executed either on CPUs or GPUs. Speedups of up to two orders of magnitude can be achieved by only one single GPU card compared to a single-core CPU. The reader can find more information in [2]. Since DualSPHysics has the capacity to simulate large number of particles at reasonable runtimes, new pre-processing and post-processing tools are needed to deal with the large data set. GenCase was developed specifically for this task of creating the initial configuration for simulations with DualSPHysics. However, the data can also be exported and used in any other SPH solver.

In this work, the main features of the GenCase code are described such as how particles are created starting from the nodes of a 3D mesh or how to import external objects created with any 3D design software. Additionally we show how it is possible to compute normal vectors for boundaries

Marching Correctors – Fast and Precise Polygonal Isosurfaces of SPH Data

Benjamin Schindler, Raphael Fuchs
Scientific Visualization Group
ETH Zürich, Switzerland
bschindler,raphael@inf.ethz.ch

Jürgen Waser
VRVis Vienna, Austria
jwaser@vrvis.at

Ronald Peikert
Scientific Visualization Group
ETH Zürich Switzerland
peikert@inf.ethz.ch

***Abstract*—This paper presents the first method for isosurface extraction from smoothed particle hydrodynamics (SPH) data that is exact with respect to the functional representation provided by SPH. The *Marching Correctors* algorithm is an extension of the *Marching Cubes* algorithm which is adapted to the SPH representation and avoids resampling to a full grid. The algorithm operates on a virtual grid of sufficiently high resolution to faithfully reconstruct the fields represented by the SPH data. The virtual grid is efficient in terms of both memory usage and computing time, because cells are only materialized and processed if they are either seed cells or intersected by the isosurface. Besides the virtual grid, a key idea of our algorithm is to add a correction step to the isosurface vertices. An evaluation of the algorithm in terms of accuracy and performance is given based on three SPH datasets. By comparing with [1] on similarly sized data a performance gain of almost two orders of magnitude was achieved. Moreover, it is demonstrated how the correction step effectively reduces the typical artifacts produced by the Marching Cubes method.**

I. Introduction

The extraction and rendering of isosurfaces is one of the basic visualization methods for scalar fields. There exists a huge body of literature on isosurface extraction with specializations for the different types of data discretization. Most of the existing methods deal with data organized in either hexahedral or tetrahedral cells, and only a small fraction addresses meshless data, that is data given on a set of points. In contrast to this, point-based representation on the output side has been treated more extensively, because many isosurface extraction methods generate points more immediately or more naturally than their connectivity. But even for meshless data, polygonal isosurfaces have a number of advantages, since for visualization purposes it is often required to compute additional properties such as volume, curvature or surface area of the extracted surfaces [2]. An example of four isosurfaces from SPH data can bee seen in Fig. 1.

SPH data sets are functionally represented scalar and vector fields which are given on a set of particles. Each particle is represented by the field values and a radially symmetric kernel. As opposed to other functional representations, SPH has kernels with relatively large support radii which means that in order to reconstruct the field at a given location, in the order of a hundred neighbors have to be evaluated.

While reconstruction of the field is comparably expensive, the SPH representation offers the advantage of high-quality gradients which can be computed at relatively little extra cost. We make use of these gradients for optimizing the intersection points generated by the standard Marching Cubes algorithm.

Our contributions are as follows:

- We present the first method for isosurface extraction from SPH datasets where all extracted vertices are guaranteed to be placed at positions where the field assumes the selected iso-value.
- We suggest a virtual grid to structure the processing and guarantee that the resulting mesh is water-tight without requiring resampling on all positions of a full grid. The virtual grid is never stored in memory in its entirety.
- We describe a trimming method at the free surface to get a consistent isosurface boundary.
- The selection of seed cells is the most expensive part of the Marching Correctors algorithm and it is computed on the GPU.
- Vertex normals are computed directly from the SPH representation instead of using a gradient estimation scheme.

In the next section we give some background on isosurface extraction with a focus on SPH data and point-sampled data. In Section III we give an overview of the Marching Correctors algorithm. An evaluation of performance and accuracy is given in Section IV followed by conclusion and future directions in Section V.

II. Related Work

a) Visualization of SPH data: There exist a few visualization packages for SPH data. SPLASH [3] is capable of producing 2D plots of data by projecting particles onto a plane, and 3D plots by integrating the kernel contributions of all particles intersecting a ray through the view pixel. In addition to these image-space methods, there are a few object-space methods available such as streamline plots. ParaView [4] supports SPH data through its meshless extension described in [5]. It has functionality for resampling SPH data onto planes, grids and arbitrary geometric meshes. This resampled data can then be used with grid-based visualization algorithms to generate isosurfaces, integral lines and surfaces, etc. However,

Improving Accuracy of Viscous Fluid Simulation Using Finite Particle Method

E. Jahanbakhsh, O. Pacot, F. Avellan
Laboratory for Machine Hydraulic (LMH)
École Polytechnique Fédéral de Lausanne (EPFL)
Lausanne, Switzerland
francois.avellan@epfl.ch

P. Maruzewski
Centre d'Ingénierie Hydraulique
EDF - DPIH
Le Bourget-du-Lac, France
pierre.maruzewski@edf.fr

***Abstract*—The present paper reports the development and application of Finite Particle Method for viscous fluid simulation. The main effort in present study is focused on increase of accuracy. To improve accuracy two approaches are studied. First of all, consistency of standard SPH restored using FPM. Secondly, automatic adaptivity and clustering of particles is remedied by applying shifting algorithm. Moreover new shifting appropriate for free surface flows introduced. Finally new algorithm validated and verified with 2D and 3D benchmarks.**

I. INTRODUTION

The Smoothed Particle Hydrodynamics (SPH) method is well suited for simulating complex fluid dynamics. Although, recent studies [l] [2] [3] [4] results in great improvement of viscous particle-based simulations, comparing to grid-based approaches, particle methods are still suffering from insufficient accuracy. Existing approaches mainly suggests two ways to increase the accuracy.

First of all, accuracy improvement is achieved by restoring consistency of standard SPH. To improve consistency of SPH substantial works has been done. For example symmetrisation formulations[12], corrective smoothed particle method (CSPM) [13], moving least square particle hydrodynamics (MLSPH) [14], the integration kernel correction [15], the reproducing kernel particle method (RKPM) [16] and finally finite particle method (FPM)[1] [17] [8].

Secondly, automatic adaptivity and clustering of particles is remedied by using re-meshing algorithms [5] or by shifting particles [4]. In re-meshing technique particles are reinitialised at regular intervals by interpolation onto a regular grid. For moving particle computations in this work we propose and demonstrate a formulation for multidimensional non-Lagrangian motion, henceforth referred to as particle motion correction

Shifting idea arose up from finite volume particle method (FVPM) where Schick [18] introduced non-Lagrangian particle motion to maintain adequate particle spacing for a 1-D problem. Furthermore Nestor et al. [3] and recently Xu et al. [4] used similar idea for more complicated problems.

In present study we try to get the benefit of both approaches simultaneously. To do this, FPM as a powerful model to restore consistency, combined with particle shifting approach.

In next section we briefly denote governing equation for fluid simulation. In Section III lack of consistency in particle approximation is addressed and FPM technique is explained by detail. Section IV mentioned a totally conservative SPH discretization adopted for FPM. We next speak about shifting technique in Section V and solution algorithm in section VI. In section VII some simulations are presented for verification and validation of proposed algorithm and .finally we have conclusion in last section.

II. GOVERNING EQUATION

To simulate viscous fluid behaviour, Navier-Stokes equations are often used as governing equations. Lagrangian description of these equations is:

$$\frac{D\rho}{Dt} = -\rho\vec{\nabla}\cdot\vec{C} \tag{1}$$

$$\rho\frac{D\vec{C}}{Dt} = \vec{\nabla}\cdot\overline{\overline{\tau}} + \vec{f} \tag{2}$$

Where (1) and (2) are mass and momentum equations respectively. Moreover, position of infinitesimal fluid elements is governed by:

$$\frac{D\vec{X}}{Dt} = \vec{C} \tag{3}$$

In (1), (2) and (3), D/Dt denotes substantial derivative; ρ is density; $\vec{C}$ is velocity vector; $\overline{\overline{\tau}}$ is stress tensor; $\vec{f}$ is external body force and $\vec{X}$ is position vector.

For weakly compressible Newtonian fluid, $\overline{\overline{\tau}}$ is defined as:

Finite volume particle method with non-circular particle supports

Libor Lobovský, Ruairi M. Nestor, Nathan J. Quinlan
Department of Mechanical and Biomedical Engineering
National University of Ireland, Galway
Galway, Ireland
libor.lobovsky@nuigalway.ie

***Abstract*—The finite volume particle method (FVPM) is a particle-based computational tool. The FVPM particles are traditionally represented by test functions that are defined on circular support domains. This study is focused on the accuracy and speed of the calculations involving particles with rectangular support domains. Error analysis as well as overall performance of such simulations is examined and a comparison to a previously published study on the method's convergence for circular supports is provided.**

I. Introduction

The finite volume particle method (FVPM) is a mesh-free particle method that inherits some of the properties of the smoothed particle hydrodynamics (SPH) as well as of the finite volume method (FVM). As in SPH, the FVPM computational domain is discretised with a finite number of computational points (particles) which are represented by smooth overlapping test functions [4]. As long as every arbitrary point in the computational domain is covered by at least one particle, FVPM maintains exact conservation.

The interaction between a pair of FVPM particles depends on the distribution of all neighbouring particles, rather than on the relative position of the two particles as in SPH. The FVPM particle interaction is realised through inter-particle fluxes which are weighted by particle interaction vectors reflecting an overlap of the particle pair and their neighbours. Computation of the particle interaction vectors requires numerical integration over complex-shaped domains and represents the most computationally demanding part of the FVPM simulation.

In order to reduce the computational costs, Quinlan and Nestor [10] suggested a fast procedure for analytic evaluation of particle interaction vectors using top-hat kernel functions. This method was proven to be an efficient and accurate alternative for two-dimensional simulations while a significant reduction of computational costs has been reported [7].

The particle test functions are traditionally defined on finite circular or spherical supports for two-dimensional, three-dimensional cases, respectively. While the top-hat kernel functions defined on circular domains are efficient for two-dimensional simulations, an application of the top-hat kernel functions for three-dimensional simulations requires an identification of complex-shaped spherical surfaces.

In order to simplify the algorithm for evaluation of particle interaction vectors in general, and especially in order to simplify the geometric complexity of this procedure when top-hat kernel functions are applied, test functions defined on rectangular supports are considered. Such an approach follows the work of Keck [5].

In the following, a comparison of the FVPM solution using particles defined on rectangular supports is compared to the results gained using traditional circular particles. Three test cases are designed in order to explore the FVPM performance for two-dimensional simualtions. Those address an FVPM capability of treating the shock wave propagation and a viscous flow in an unbounded and bounded domain. The particles with rectangular supports are tested for the real world application involving interaction of fluid with moving bodies.

II. Finite Volume Particle Method

A. Governing Equations

Within this section, a brief outline of the method is provided. The FVPM formulation for viscous fluid flow [4], [8] is derived from Navier-Stokes equations in a conservative form

$$\frac{\partial \mathbf{U}}{\partial t} + \nabla \left(\mathbf{F} - \mathbf{G} \right) = 0, \tag{1}$$

where $\mathbf{U}$ is the vector of conserved variables, $\mathbf{F}$ is the inviscid flux, $\mathbf{G}$ is the viscous flux and t is time. For a weakly compressible fluid, the system of Navier-Stokes equations is accompanied by an equation of state, in which the fluid pressure p is an explicit function of density ρ,

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

where ρ_0 is the reference density, c_0 is the reference value of speed of sound and $\gamma = 7$ [1].

The Navier-Stokes equations and the equation of state are discretised for a finite set of particles which fill in the computational domain. Each particle is associated with a compactly supported overlapping test function

$$\psi_i(\mathbf{x}, t) = \frac{W(\mathbf{x} - \mathbf{x}_i(t), h)}{\sum_{j=1}^{N} W(\mathbf{x} - \mathbf{x}_j(t), h)}, \tag{3}$$

where $\mathbf{x}$ is the position vector, N is the number of particles and $W_i = W(\mathbf{x} - \mathbf{x}_i(t), h)$ is a kernel function for a particle

A maximum entropy method for particle simulations

Daniel Duque
Dep. Ciencias Apl. Ing. Naval
ETS Ingenieros Navales
Universidad Politécnica de Madrid
28040 Madrid, Spain
daniel.duque@upm.es

Pep Español
Dep. Física Fundamental
Facultad de Ciencias
Universidad Nacional de Educación a Distancia
28040 Madrid, Spain

***Abstract*—Maximum entropy methods have recently been proposed to overcome known limitations of the basis functions used in Galerkin methods. The first one to appear was the local method, which focused on first-order accuracy and was presented as an extension of the standard finite element method. The next one was the second order method, which tried to impose second order accuracy. We discuss here the possibility of using these two methods for particle simulations, focusing on the difficult problem of evaluating space derivatives, specially second order ones, such as the Laplacian. In principle, this would produce a local method that first, or second order consistent. In fact, as we will see, the details of the implementation are not trivial.**

I. Introduction

A certain class of numerical methods, called maximum entropy (ME) methods, have recently been proposed to overcome known limitations of some basis functions used in Galerkin methods. The ideas behind these methods seem very interesting from computational fluid dynamics (CFD), since in principle they are "local" and comply with different degrees of consistency by construction. In particular, they hold the promise of providing good results for the Laplacian, which is needed for the viscosity term in the Navier-Stokes equations. As shown in Table I, an obvious candidate such as the linear finite elements on the Delaunay triangulation (just FEM for short) fails regarding second-order consistency (2C) in 2D, despite its brilliant performance in 1D, and despite it satisfying first-order consistency (1C) exactly!. In fact, it is not even L^2 convergent for quadratic functions, as we will discuss later here, and already mentioned in our previous contribution to this series [1].

Chronologically, the first method one to be presented was the local ME method, LME, which focused on first-order accuracy [2]. This will be the first to be discussed here in the Theories section. We will then move on to the more powerful second-order ME, or SME [3]. Both of these will be compared with FEM in the Results and Discussion section.

TABLE I
Consistencies of different methods; "y" means "yes" (compliance to machine precision), "n" means "no". *: consistency in the derivatives.

Method	Order of consistency		
	0C	1C	2C
FME 1D	y	y	y
LME 1D	y	y	n
SME 1D	y	y	y
FME 2D	y	y	n
LME 2D	y	y	n
SME 2D	y	y	y*

II. Maximum entropy theories

We present here a derivation of ME methods, from the simplest case to more complex ones.

A. A point-based procedure

We begin with a familiar problem in numerical analysis: given a set of $a = 1 \dots N$ nodes in space, evaluate a set of functions $s_a(x)$ (which we will call basis, or shape, functions) so that from field values defined at the nodes, f_a, we can compute faithful reconstructions everywhere, of the field $f(x) = \sum_a f_a s_a(x)$, and its derivatives $f'(x)$ and $f''(x)$.

In fact, our particular aim in CFD in an SPH-like mind frame is slightly different: given a set of $a = 1 \dots N$ nodes in space (aka "particles"), evaluate a set of functions $s_a(x)$ so that from field values defined at the nodes, f_a, (which will be hydrodynamic fields, such as the pressure) we can compute faithful reconstructions at other nodes, of the field $f_b = \sum_a f_a s_a(x_b)$, and its derivatives f'_b and f''_b.

ME methods differ from many others in a crucial feature: the properties of the functions are given *at each value* of the coordinate x. Thus, for a given x the procedure results in N values for the functions s_a evaluated at x. The theory begins with a prescription for the "entropy" of our N values; this formula is well known in information theory (Shannon's entropy):

$$S = -\sum_{a=1}^{N} s_a \log s_a.$$

The entropy is to be maximised if we are to make an unbiased guess. But we want to specify some additional requirements. A standard one in probability is to require $\sum s_a = 1$; this is precisely the condition that a constant function be reconstructed, or zeroth-order consistency (0C). Thus, we will want to extremize $S - C_0$, where the constraint

Simulation of gravity driven free-surface flow in fractured geological media

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 unsaturated free-surface flow in fractured geological media represents a challenge due to the highly heterogeneous flow field induced by extensive faults, joints and fissures. Free-surface flow in unsaturated media leads to highly intermittent flow regimes and flow velocities well above those assumed for the bulk volume. However, common modeling approaches relying on volume-averaged effective equations fail to capture this flow feature. In this work we present micro-scale flow simulations using a three-dimensional multiphase SPH code. Pairwise fluid-fluid and solid-fluid interaction forces are used to simulate a wide range of wetting conditions encountered on rock surfaces. It is shown that static contact angles for sessiles droplets are independent of the model discretization, i.e. the total amount of particles. Thus, computation times can be reduced without sacrificing qualitative or quantitative information. Furthermore we show that our model is in accordance with general scaling laws for droplet flow.**

I. Introduction

Facing a global climate change and a rapidly growing world population the management of limited water resources becomes increasingly difficult. Aquifers (i.e. porous and/or fractured rocks from which economically profitable quantities of water can be extracted) provide the main storage for fresh water within the hydrogeological cycle between atmosphere and surface waters. The subsurface can be divided into (1) an unsaturated and (2) a saturated zone. The saturated zone is delineated by the water table, i.e. all available voids are filled with water, whereas the unsaturated zone comprises the part of the subsurface between water table and surface where pores and fractures are only partially filled with water. The unsaturated zone therefor provides the main pathway for precipitation and surface waters to the saturated zone.

Compared to unconsolidated porous media tectonically stressed aquifers provide additional highly conductive fractures, joints and faults embedded in a low conductive matrix. Even though their total fraction of the aquifers porosity may be as low as $1\,\%$ [1] the importance of fractures for the transport of water and contaminants has been proven by a variety of authors using integral fieldtests as well as laboratory experiments ([2],[3],[4],[5],[6]) and analytical solutions ([7],[8],[9]). As about $50\,\%$ of the earth's surface [10] is covered by hard rocks integrated management of these resources largely depends on a thorough understanding of the aquifer's dynamical response to recharge and contamination.

Transport of water through the unsaturated zone is partially poorly unterstood due to the highly non-linear and intermittent flow processes involved ([11],[12],[13]). In unconsolidated media unsaturated flow is often simulated by volume-averaged effective modeling approaches such as the Richard's equation [14]. However, the behavior of water in unsaturated fractures is affected by a multitude of hydrodynamic effects that cannot be captured by classical continuum models and may give rise to highly nonlinear flow modes. Depending on the fracture properties (aperture, inclination, roughness) fluid flow within fracture elements is controlled by the interaction of body and surface forces. Thus gravity driven or capillary driven flow may prevail. Microscale modeling approaches should therefor be able to deal with the resulting highly dynamical interfaces and provide a flexible numerical tool for validation and prediction of transient flow. Furthermore transport relevant properties such as velocity and surface contact area are of interest for the characterization of matrix-fracture interaction and require adequate numerical techniques.

This paper demonstrates how Smoothed Particle Hydrodynamics (SPH) can be used to simulate small scale free-surface fluid flow in wide aperture fractures. Particular attention is paid to general approaches for calibration and verification of the SPH model for droplet flow.

II. Method

In the following we give a brief description of our model and the governing equations. More detailed derivations and approximations involved in the SPH method can be found for example in ([15],[16],[17]).

We use an SPH momentum conservation equation proposed in [15]:

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

SPH Simulation of Gas Explosion Inside an Underwater non-Cohesive Sediment Deposit

Sauro Manenti, Stefano Sibilla, Mario Gallati
Dept. of Hydraulic and environmental Engineering
University of Pavia
Pavia, Italy
sauro.manenti@unipv.it

Giordano Agate, Roberto Guandalini
Environment and Sustainable Development Dept.
RSE s.p.a.
Milan, Italy
giordano.agate@rse-web.it

***Abstract*— This paper shows an early application of an SPH-based numerical model toward the simulating of underwater explosion inside a non-cohesive sediment deposit. The analytical formulation of the phenomenon is briefly recalled and then the both discretized equations and numerical features are illustrated. Finally some results concerning relatively simple test cases are discussed in comparison with theoretical predictions and the main conclusions are given.**

I. INTRODUCTION

Several artificial reservoirs are affected by the siltation process owing to the sediment load of the inflowing river: this may cause a significant reduction of the initial storage capacity within a relatively short time compared with their life span. Such a phenomenon, besides limiting the operating capacity of a hydropower plant, contributes to wear out the turbines and decreases their efficiency. An economical and effective method for siltation control, referred to as flushing, consists of a periodical opening of the bottom outlet carried out for safety purposes: this causes the removal of part of the nearest bed sediments owing to the action of the bottom shear stress exerted by the rapid water outflow under the dam heel.

In order to improve the removal efficiency of this method a possible solution consists in adopting explosive charges, of suitable size, power and allocation, which are embedded below the sediment surface, whose goal is the re-suspension of the sediment in the water column and increase of the bottom roughness. Since the use of explosive charges near the dam is rather dangerous owing to its possible damage or failure, it deserves particular care and should be carefully investigated.

This work reports the preliminary phase of a wider research program aiming at the evaluation of the feasibility of the proposed method through experimental and numerical investigations.

II. ANALYTICAL AND NUNERICAL ASPECTS

A. Physics and Governing Equations of Explosion

The explosion process of high explosive (HE) materials is characterized by a violent oxidation, involving a chemical compound and an oxidizer, releasing a great amount of heat (say reaction heat) since the internal energy of the products is lower then the one of the reactants [1], [2].

Even if such a phenomenon develops at a very high speed of reaction, in the early phase it is characterized by two distinct inhomogeneous zones [3]: a detonation-produced explosive gas and a non-oxidized explosive; they are separated by a very thin layer which represents the front of a reacting shock wave (detonation wave) advancing with a characteristic velocity U.

The detonation wave triggers the chemical reaction in the non-oxidized explosive and the release of the chemical energy, in turn, sustains such a shock wave.

This early phase is called detonation and is very fast if compared to the subsequent one since U is supersonic. After the detonation, a gaseous mass at a very high pressure, density and temperature expands in the space: this is referred to as expansion phase and its timescale is lower than that one of the detonation phase; this is the reason why in the following will be assumed that the HE charge is fully detonated.

The discretized governing equations describing the expansion of the detonated gas can be written by considering an inviscid fluid [4] and assuming the equation of state for an ideal (polytrophic) gas undergoing an adiabatic process [5]:

$$\begin{aligned}
\left\langle \frac{D\rho}{Dt} \right\rangle_i &= \sum_{j=1}^{N} m_j \left(\mathbf{v}_i - \mathbf{v}_j\right)\cdot\nabla W_{ij} \\
\left\langle \frac{D\mathbf{v}}{Dt} \right\rangle_i &= -\sum_{j=1}^{N} m_j \left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2} + \Pi_{ij} \right)\cdot\nabla W_{ij} \\
\left\langle \frac{De}{Dt} \right\rangle_i &= \frac{1}{2}\sum_{j=1}^{N} m_j \left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2} + \Pi_{ij} \right)\left(\mathbf{v}_i - \mathbf{v}_j\right)\cdot\nabla W_{ij} \\
p_i &= \left(\gamma - 1\right)\rho_i\, e_i \qquad c_s^2 = \gamma\,(\gamma-1)\, e = \frac{\varepsilon}{\rho}
\end{aligned} \tag{1}$$

In (1) ρ, p, e, $\mathbf{v}$, are respectively the density, pressure, specific internal energy and velocity of the fluid; γ and ε denote the constant in the equation of state and the fluid compressibility modulus. The kernel function W_{ij} is a cubic spline; the standard formulation of the Monaghan's artificial viscosity Π_{ij} is adopted, with β_M=0.

A SPH model for seepage flow through deformable porous media

Ha H. Bui, C.T. Nguyen
Department of Civil Engineering
School of Science & Engineering, Ritsumeikan University
1-1-1 Noji-Higashi, Kusatsu, Shiga, 525-0055, Japan
Email: hhbui@fc.ritsumei.ac.jp

K. Sako, R. Fukawaga
Department of Civil Engineering
School of Science & Engineering, Ritsumeikan University
1-1-1 Noji-Higashi, Kusatsu, Shiga, 525-0055, Japan
Email: fukagawa@se.ritsumei.ac.jp

***Abstract*—This paper presents a new numerical framework based on the SPH method for modelling seepage flow through deformable porous media, and its application to investigate seepage induced soil erosion and failure. The two-phase flow model developed herein was derived from the general Biot's two-phase mixture theory wherein soil was modelled using an elasto-plastic constitutive model and pore-water was modelled as an incompressible fluid. The incompressible smoothed particle hydrodynamics (ISPH) method was adapted to account for the interaction between soil and water inside the porous media. The coefficient of porosity was considered to vary with large deformation of soil, and the coefficient of permeability was assumed as a function of the void ratio. The developed model was first validated with the finite element solution for the seepage flow through an embankment. Then, it was applied to simulate the experiment of seepage induced embankment failure. Comparisons with finite element solution and experiment showed that the developed model is a promising numerical approach to investigate seepage flow through a porous media and seepage induced soil erosion/failure.**

I. INTRODUTION

Flow through deformable porous media can be found in many industrial and environmental applications, for example geomechanics, oil production, filtration process, wastewater treatment, etc. In our research, we are mainly interested in applications in geomechanics, which is characterized by seepage flow through saturated/unsaturated porous soil media. The seepage flow may induce soil erosion or reduction in soil strength, which may lead to failure of a soil structure. Therefore, study of seepage flow through porous soil media is of an important issue in geotechnical engineering. In the past, numerical studies of seepage flow induced soil failure and erosion are often performed using the finite element method (FEM), in which both seepage flow and soil deformation are resolved through the grid system. However, dealing with large deformation and post-failure of soil, FEM often leads to numerical difficulty due to the well-known mesh-tangling problem even when the updated Lagrangian method is adopted. To overcome this limitation, the smoothed particle hydrodynamics (SPH) method has been recently extended by the authors, and showed to be a powerful alternative for simulations of large deformation and post-failure of geomaterials (Bui et al. 2007a, b, 2008, 2011). In this paper, taking the above advantages of SPH into account, a numerical framework for seepage flow through deformable porous media is developed. Our two-phase flow model was based on the general Biot's two-phase mixture theory (Biot, 1956). Herein, both soil and water were resolved separately using their own system of governing equations. The incompressible smoothed particle hydrodynamics (ISPH) (Shao et al. 2003) was adapted to model the development of the pore-water pressure through the earth structure. The deformable porous media was modelled using the elasto-plastic SPH framework (Bui et al., 2008). To simplify our model, unsaturated soil behaviour was not considered. Furthermore, spatial variation of the porosity was also not considered when discretizing the two-phase flow equations. The developed model was then applied to investigate the seepage flow behaviour through a riverbank, and to simulate the seepage induced progressive failure of the riverbank. Physical model test of riverbank failure was also conducted to investigate mechanism of riverbank failure, and to verify performance of the proposed model.

II. LABORATORY TEST OF RIVER EMBANKMENT FAILURE

A. Physical model test

The geometry and boundary conditions of the embankment model test is shown in Figure 1. The embankment was made of Masa soil, which is weathered granite, typically found in Kansai area in Japan. The initial water content of slope was kept at approximate 5% and the porosity is about $n = 0.4$.

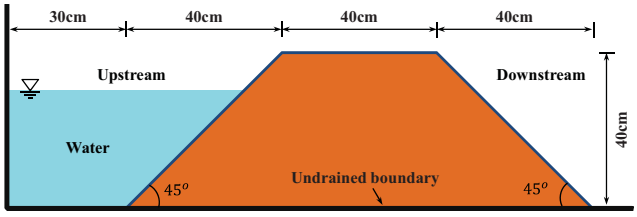

Figure 1. Geometry and boudary condition of embankmen model

A simple and efficient SPH algorithm for multi-fluids

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

***Abstract*—In this paper an SPH algorithm for multi-fluid flow is described. The algorithm is efficient, simple and robust. The inviscid equations of motion are derived from a Lagrangian together with the constraint provided by the continuity equation. The viscous flow equations then follow by adding a viscous term. Rigid boundaries are simulated using boundary force particles. Each fluid is approximated as weakly compressible. When the SPH force interaction is between two particles of different fluids we increase the pressure terms in a similar way to Grenier et al. (2009). This simple procedure stabilizes the interface between the fluids. The equations of motion were integrated using a time stepping rule based on a second order Symplectic integrator. When linear and angular momentum should be conserved exactly they are conserved to within roundoff errors. The algorithm was tested by applying it to three problems. The first is a free surface problem involving fluids with a density ratio in the range 1 to 1000 The simulations converge and the results are in good agreement with the exact inviscid, incompressible theory. The second is a Rayleigh Taylor instability where convergence and agreement with level-set calculations is established. The third is a gravity current with density ratios in the range 2 to 30. The results of these simulations are in good agreement with those of other authors, and in satisfactory agreement with experimental results.**

I. Introduction

Many fluid dynamical problems involve more than one fluid. In some cases, for example most liquids at normal temperature and pressure, the density ratio of the fluids is < 1.3, but in the case of gases, the density ratios can be much larger. For example the density ratio of air and helium is ~ 7, and in the experiments of Gröbelbauer et al. (1993) the ratio was as high as 20.6. Two fluids with a very high density ratio which are often present in fluid dynamical problems are air and water. Although the density ratio in this case is ~ 800 the air cannot be neglected because its thermodynamic pressure has a significant effect on the dynamics.

In problems involving more than one fluid there is a discontinuity in the density and other material properties at the interface between any two fluids. Colagrossi and Landrini (2003) showed that a standard form of weakly compressible SPH gave very poor results when the density ratios were large. They found that for stable SPH weakly compressible simulations of two fluids with large density ratios, it was necessary to introduce the following changes to their standard SPH algorithm: density renormalization, a large surface tension in the low density phase, a smoothing of the velocity field, and a very large difference in the speed of sound in the two fluids. The first of these increases the computing time, though whether or not it is significant depended on the number of time steps between renormalization. The second is unphysical because surface tension in a liquid is larger than in a gas. The velocity smoothing has a stablising effect, but it also increases the computing time. The ratio of the speed of sound in the low density fluid (subscript g) to that in the high density liquid (subscript ℓ), was approximately $\sqrt{\rho_\ell \gamma_g/(\rho_g \gamma_\ell)}$. For a density ratio 1000:1, and taking the values of γ used by Colagrossi (2005), this factor is ~ 14, so the speed of sound in the gas is much greater than in the liquid. Although this is unphysical, because the speed of sound in water is greater than that in air at room temperature and pressure, in these quasi-incompressible calculations the speed of sound of the liquid is artificial, and chosen to ensure the density fluctuations are sufficiently low. The speed of sound required for this is, for typical simulations, a factor ~ 50 less than the true speed of sound in the liquid. In any case, the higher speed of sound in the gas was found by Colagrossi and Landrini (2003) to be necessary to stablise their algorithm. A consequence of this high speed of sound is that the CFL time step condition requires very small time steps.

Because of the discontinuity in the density Hu and Adams (2006) re-wrote the SPH equations in terms of particle number density which can easily be made continuous across a density discontinuity. They applied the resulting algorithm to problems involving surface tension. They obtained good results but their method was not applicable to free surface problems. In later papers Hu and Adams (2007, 2009) applied a projection method to handle a system with one or more incompressible fluids. This method was successful for incompressible fluids without free surfaces, but it is not applicable to problems such as an expanding bouyant bubble interacting with a free surface. In a recent paper (Grenier et al. 2009) proposed a new SPH based method for multi-phase problems. In this method, the density is calculated from a standard SPH summation using a Shepard kernel which in turn requires a particle volume distribution which is calculated from a version of the SPH continuity equation. In addition they added a repulsion term

ISPH Modelling of Rayleigh-Taylor Instability

Mostafa Safdari Shadloo
Faculty of Engineering and Natural Sciences,
Advanced Composites and Polymer Processing Laboratory
Sabanci University,
Istanbul, Turkey
mostafa@sabanciuniv.edu

Mehmet Yildiz
Faculty of Engineering and Natural Sciences,
Advanced Composites and Polymer Processing Laboratory
Sabanci University,
Istanbul, Turkey
meyildiz@sabanciuniv.edu

***Abstract*- This paper presents a *Smoothed Particle Hydrodynamics* (SPH) solution to a *Rayleigh-Taylor Instability* (RTI) problem in an incompressible viscous two-phase immiscible fluid with an interfacial tension. The evolution of the fluid-fluid interface is numerically investigated for four different density ratios. The simulation outcomes are compared with existing results in literature. Three stages of instability, namely the exponential growth rate, the formation of circular form at the crest of spike and the appearance of the final shape of instability, are discussed for different density ratios. It is shown that the numerical algorithm used in this work is capable of capturing the complete physics behind the RTI, such as interface evolution, growth rate and secondary instability accurately, and successfully.**

I. INTRODUTION

Instability at the interface between two horizontal parallel fluids of different viscosities and densities with the heavier fluid at the top and the lighter at the bottom is known as the Rayleigh–Taylor Instability (RTI) to honour the pioneering works of Lord Rayleigh [1] and G. I. Taylor [2]. This phenomenon can be observed in a wide range of natural and astrophysical events. The instability initiates when a multiphase fluid system with different densities experiences gravitational force. As a result, an unstable disturbance tends to grow in the direction of the gravitational field, thereby releasing the potential energy of the system and reducing the combined potential energy of the fluids. Due to being an important phenomenon in many fields of engineering and sciences, the RTI has been widely investigated by using experimental [3, 4], analytical [5, 6] as well as numerical approaches [7, 8].

The Smoothed Particle Hydrodynamics (SPH) is a relatively new numerical approach that has attracted significant attention in the last 15 years. Compared with the conventional mesh-dependent computational fluid dynamics (CFD), the SPH method exhibits unique advantages in modelling fluid flows and associated transport phenomena due to its capabilities of handling complex material surface behaviour as well as modelling complicated physics in a relatively simple manner.

There are a few works which have used the SPH method to model the RTI problem [12-15]. Cummins and Rudman [9] solved the RTI phenomenon using a projection method-based Incompressible SPH (ISPH) approach. Tartakovsky et al. [10] modelled the Rayleigh-Taylor instability in a multiphase and multi-component mixture with the Weakly Compressible SPH (WCSPH) method through solving momentum balance and species mass balance equations concurrently. Hu and Adams [11] solved the RTI problem as a benchmarking problem through combining projection methods used in [9] and [12]. More recently Grenier et al. [13] presented a WCSPH formulation for simulating interface flows, and model the RTI to validate their numerical scheme. It should be emphasized here that none of these cited works has included the effect of the surface tension in their simulations. These works handled the RTI as a validation test case for their algorithm and did not focus on the physics of the problem in detail.

The aim of this work is to simulate the RTI by using the ISPH method, thereby showing the ability of the SPH technique to capture this hydrodynamic instability and relevant physics for a wide range of density ratios. The current presentation differs from earlier works in the following aspects: Even though multiphase grid-based methods considered the RTI problem in detail, it has been barely considered within the context of the SPH method, and if so, mainly for the density ratio of $\rho_2 / \rho_1 = 1.8$.

II. SMOOTHED PARTICLE HYDRODYNAMICS

A. Introduction

Initially developed to solve the astrophysics problems in 1977 by Gingold and Monaghan, and Lucy in separate works [14, 15], and later extended to solve a wide variety of fluid dynamics problems [16-18], SPH is a member of Lagrangian methods. The SPH method is based on the smoothing of the hydrodynamics properties of fluid elements, which are represented by movable points (also referred to as particles), over the solution domain using a weighting function, $W\left(r_{ij},h\right)$, or in short W_{ij}. The weighting function W_{ij} (also known as the kernel function in the SPH literature) is an arbitrary function (e.g. exponential, spline, and etc.) with some special properties as listed [19]. Here, r_{ij} is the magnitude of the distance vector ($\vec{\mathbf{r}}_{ij} = \vec{\mathbf{r}}_i - \vec{\mathbf{r}}_j$) between the particle of interest $\mathbf{i}$ and its

A fully coupled 3D transport model in SPH for multi-species reaction-diffusion systems.

S. Adami, X.Y. Hu and N.A. Adams
Institute of Aerodynamics and Fluid Mechanics
Technische Universität München
85748 Garching, Germany
Contact: stefan.adami@tum.de

E.M. Ryan and A.M. Tartakovsky
Computational Mathematics Technical Group
Pacific Northwest National Laboratory
Richland, WA 99352, USA

***Abstract*—In this paper we present a fully generalized transport model for multiple species in complex two and three-dimensional geometries. Based on previous work [1] we have extended our interfacial reaction-diffusion model to handle arbitrary numbers of species allowing for coupled reaction models. Each species is tracked independently and we consider different physics of a species with respect to the bulk phases in contact. We use our SPH model to simulate the reaction-diffusion problem on a pore-scale level of a solid oxide fuel cell (SOFC) with special emphasize on the effect of surface diffusion.**

I. Introduction

In the context of global warming a lot of effort has been focused on developing renewable and alternative energy sources that reduce the production of greenhouse gases. A promising alternative energy technology, which converts chemical energy to electrical energy is the cold combustion in fuel cells. Amongst various realizations of this technology, solid oxide fuel cells (SOFC) are of special interest as their working conditions at high temperatures enable the usage of a wide variety of fuels [2]. Degradation is a central issue in SOFCs, such as chromium poisoning, which arises from the chromium contained in the stainless steel which is typically used for the current collectors. This chromium reacts with air to form volatile chromium species that [3] migrate into the porous cathode and react with its surface. This so-called *chromium poisoning* has been shown to decrease the efficiency of the fuel cell dramatically and has to be controlled [4].

Ryan et al. [5] have developed a pore-scaled SPH model of a SOFC to investigate the reactive transport of chromium species in the cathode. Based on a multi-scale approach including a cell level model of the cathode, air channel and the current collector they determined the boundary conditions for the pore scale simulations. In their two-dimensional work they varied the reaction rates of oxygen and chromium and the working conditions of the fuel cell to study the deposition of chromium. They could reproduce qualitatively the species distributions in the cathode as compared to experimental findings [4] and show that their nonlinear competitive adsorption-desorption model is adequate to study the complex chromium poisoning.

In our current work we want to use the competitive adsorption-desorption model to simulate the chromium deposition in a realistic three-dimensional cathode. Different from the two-dimensional study of Ryan et al., here we cannot neglect surface diffusion in the porous material as this structure allows an interfacial connection throughout the entire domain and diffusion along the interface can alter the species dynamics strongly.

To simulate a multi-component reactive transport problem we have extended our SPH method for surfactant dynamics [1] to account for multiple species and coupled transport models. We have validated this method with analytical solutions for coupled transport-diffusion systems with different boundary conditions (Neumann, Dirichlet and Robin) and demonstrate the significance of surface diffusion for the species transport in a real porous cathode structure.

II. Governing equations

Briefly we recall the governing equations of the fluid and species dynamics in a porous structure in a very general form. From mass conservation we can formulate the continuity equation in the form

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

where ρ, $\vec{\mathbf{u}}$ and t denote the density, the velocity vector and the time, respectively. The momentum equation in Lagrangian form with the pressure p, a bodyforce $\vec{\mathbf{g}}$ and the dynamic viscosity η is given by

$$\rho\frac{d\vec{\mathbf{u}}}{dt} = -\nabla p + \rho\vec{\mathbf{g}} + \nabla\eta\nabla\ .\vec{\mathbf{u}} \tag{2}$$

An advection-diffusion equation is used to describe the dynamics of a species α in a bulk phase l according to

$$\frac{dm_C^\alpha}{dt} = \int \nabla\mathbf{D}_\infty^\alpha\nabla C^\alpha \ dV - \int \dot{S_\Sigma}^\alpha\delta_\Sigma \ dV \ . \tag{3}$$

Here, m_C^α and C^α denote the mass and mass concentration of the species α in the bulk. Assuming isotropic bulk diffusion the diffusion tensor $\mathbf{D}_\infty{}^\alpha$ reduces to the scalar diffusion coefficient D_∞^α. Note that the bulk diffusion can vary for each species in different bulk phases, i.e. the diffusion coefficient D_∞^α is a function of the species type α and the bulk phase where it is dissolved. The last term in (3) represents the transport of species α from/to the bulk phase to/from an

ISFAA, prospects for an implicit SPH

Escartín, J.A.
Dept. de Física i Enginyeria Nuclear. Universitat Politècnica de Catalunya, Compte d'Urgell 187, 08036 Barcelona (Spain)
jose.antonio.escartin@upc.edu

García-Senz, D.
Dept. de Física i Enginyeria Nuclear. Universitat Politècnica de Catalunya, Compte d'Urgell 187, 08036 Barcelona (Spain)
and Institut d'Estudis Espacials de Catalunya. Gran Capità 2-4, 08034 Barcelona (Spain)
domingo.garcia@upc.edu

Cabezón, R.M.
Dept. Physik Universität Basel. Klingelbergstrasse 82, 4056 Basel (Switzerland)
and Dept. de Física i Enginyeria Nuclear. Universitat Politècnica de Catalunya, Compte d'Urgell 187, 08036 Barcelona (Spain)
ruben.cabezon@unibas.ch

***Abstract*—The explicit nature of the smoothed particle hydrodynamics technique restricts its applications to dynamical phenomenae such as collisions or explosions. Nevertheless a large field of applications in astrophysics are related to quasi-hydrostatic evolution as for instance the pre-explosive stage leading to novae or Type Ia supernovae or advanced stellar evolutionary phases. If we want to apply the SPH technique to these systems an implicit scheme has to be built. Nevertheless devising an implicit SPH presents a number of numerical difficulties which have prevented its development until recent times, Knapp [4]. In this paper we explain the main features of an parallelized implicit SPH called ISFAA (Implicit SPH for Astrophysical Applications) which extends the work of Knapp by including a more powerful numerical scheme, and incorporates artificial viscosity, gravity, conductive transport and nuclear reactions. We have checked the scheme through several tests such as simulating a the wall heating shock test, Sedov like explosion or stability of a massive white dwarf. These tests were calculated using a low number of particles ($\sim 10,000$). Our conclusion is that although the scheme is promising it would be necessary to make use of supercomputers to carry out realistic calculations with ISFAA. Another important improvement would be to enhance the stability of the numerical scheme in order to increase the time step to overpass the Courant time-step in a larger factor which until now is around 50.**

I. Introduction

Implicit quasi-hydrostatic schemes written in spherical symmetry were one of the pillars of computational astrophysics during the second part of the last century. The generalized use of the so called Henyey-like codes, Henyey et al. [3], were crucial to understand stellar evolution during the long quiescent stages associated to hydrogen and helium burning. On the other hand explicit hydrocodes were used to reproduce very dynamical events such as novae or supernovae explosions. Nowadays the computer performance is large enough to allow detailed hydrodynamic simulations in 3D. In this respect a widely used hydrodynamic code is the Smoothed Particle Hydrodynamics (SPH) which is a gridless method of Lagrangian nature with negligible numerical diffusion. Until now SPH has been applied to treat dynamical phenomena and therefore explicit integration schemes were preferred as a framework to implement the method. However it should be recognized the potential relevance of an implicit scheme of integration to simulate multi-D astrophysical problems which requires the use of long time-steps (as compared to the Courant time-step). For example very few is known concerning the long quasi-hydrostatic stage leading to the explosion of a white dwarf, a phenomenon known as a Type Ia supernovae explosion Hillebrandt & Niemeyer [2]. Such stage may last hundred or thousands years making explicit hydrodynamics useless because typical Courant time is lesser than a second. Therefore implicit hydrodynamics could be the solution to handle these kind of problems. Nevertheless, because of its inherent algebraic complexity and demanding computer resources very few effort, Knapp [4], has been invested until know to build an implicit SPH which can be used to study quasi-hydrostatic multi-D phenomena.

In this paper we explain the main features and the prospects for the development of an implicit SPH code characterized by: 1) It uses the modern conservative SPH formalism based in the variational principle, 2) the use of efficient PARDISO parallelized routines to handle sparse matrix, 3) it incorporates the necessary physics to handle common astrophysical problems in three dimensions, i.e. artificial viscosity, gravity, conductive transport, nuclear reactions and complex equation of state (EOS). Preliminary tests calculations using the developed scheme and N=$(1-8)\ 10^4$ particles running in a small 16 core desktop computer are promising. In Section II we explain the main technical features of the developed scheme. Section IV is devoted to describe and discuss the results of three tests: wall heating shock problems and the Sedov test and the stability of a $1.15 M_{\odot}$ white dwarf. Finally we summarize the conclusions of our work in the last section V.

ISPH-FEM coupling simulator for the FSI problems

Mitsuteru ASAI, Abdelraheem M. Aly, Yoshimi SONODA
Department of Civil Engineering
Kyushu University
Fukuoka, Japan
Email: asai@doc.kyushu-u.ac.jp

***Abstract*— Fluid-Structure Interaction (FSI) simulator has been developed by using a stabilized Incompressible SPH method and the largely deformed FEM. A small modification of ISPH is given in the source term of pressure Poisson equation to evaluate smooth pressure distribution. In addition, large eddy simulation (LES) with the Smagorinsky sub-grid scale model is introduced to include the eddy viscosity effect. After these modifications in the ISPH, the moving boundary ghost particles have been introduced to communicate ISPH and FEM results. During the FSI simulation, the FEM structural boundary has been changing especially in the large deformed analysis. The moving boundary particles are located at the same position of the nodes of FEM. Therefore, the movement of boundary particles is automatically captured from the result of FEM. Then, the position and velocity of the ghost particle can be easily evaluated from the information at the boundary particle as the nodal result of FEM.**

I. INTRODUTION

For the FSI simulation, accurate pressure representation and robust boundary treatment are strongly desired.

The source term in pressure Poisson equation (PPE) for ISPH is not unique, it have several formulations in the literature as Lee et al. [1] and Khayyer et al. [2-3]. The source term is derived as a function of density variation and velocity divergence condition. The former formulation with density variation can keep an uniform particle distribution, although evaluated pressure include high unrealistic fluctuation. On the other hand, the formulation of the divergence-free condition evaluates much smoother pressure distribution, but density errors may occur due to particle clustering. Then, modified schemes have been proposed to satisfy the above two conditions; density invariant and divergence free condition and the relaxation coefficient is multiply in term of density invariant for smoothing the resultant pressure. Recently, in the framework of MPS, there is a trend to introduce a higher order source term in the PPE. Kondo and Koshizuka [4] proposed a new formulation with a source term composed by three parts; one is main part and another two terms related to error-compensating parts. Tanaka and Masunaga [5] introduced a similar high order source term with two components incorporated with quasi-compressibility. Note that the number of PPEs per time step in their higher order source term formulations is just one and its numerical cost is almost same as the original scheme. In this study, we re-formulate a source term of the PPE which contains both contributions from velocity-divergence free and density invariance conditions. The suitable relaxation coefficient will be obtained from the hydrostatic pressure calculations as pre-analysis calculations.

In order to communicate boundary information between ISPH and structural FEM, the ghost boundary technique is utilized in this paper. Precious pressure value should be sent from the ISPH and FEM, and reasonable displacement and velocity filed of structure surface should be referred in the ISPH boundary. The conventional ghost boundary works well for the above purpose. The proposed FSI simulator was demonstrated with a simple example.

II. STABILIZED ISPH

In this section, a stabilized ISPH is developed by modifying the source term in the pressure Poisson equation. In addition, a conventional turbulence model 'Smagorinsky model' is introduced into ISPH.

A. Governing equations

The mass and momentum equations of the flows are given as

$$\frac{D\rho}{Dt} + \rho \nabla . \overline{u} = 0 \tag{1}$$

$$\frac{D\overline{u}}{Dt} = -\frac{1}{\rho}\nabla \overline{p} + \upsilon \nabla^2 \overline{u} + \frac{1}{\rho}\nabla . \overline{\tau} + \overline{F} \tag{2}$$

where ρ and υ are density and kinematic viscosity of fluid, $\overline{u}$ and $\overline{p}$ are the velocity and pressure vectors of fluid respectively, $\overline{F}$ is external force, and t indicates time. In the most general incompressible flow approach, the density is assumed by a constant value with its initial value ρ^0.

B. Modification in the source terom of pressure Poisson equation

The main concept in an incompressible SPH method is solving a descritized pressure Poisson equation at every time step to get the pressure value.

In the sense of physical observation, physical density should keep its initial value for incompressible flow. However, during numerical simulation, the 'particle' density may change slightly from the initial value because the particle density is strongly dependent on particle

Testing accuracy and convergence of GPUSPH for free-surface flows

Erin A. Rooney, Wan-Yi Wei,
Jennifer L. Irish, Robert Weiss
Dept of Civil Eng; Dept of Geology and Geophysics
Texas A&M University
College Station, Texas, USA
jirish@civil.tamu.edu

Robert A. Dalrymple
Dept of Civil Engineering
Johns Hopkins University
Baltimore, Maryland, USA

Alexis Hérault
Instituto Nazionale di Geofisica e Vulcanologia
Catania, Italy
Département d'Ingénierie Mathématique, CNAM
Paris, France

Giuseppe Bilotta
Dipartimento di Matematica e Informatica
Università di Catania
Instituto Nazionale di Geofisica e Vulcanologia
Catania, Italy

***Abstract*—SPH can be a valuable tool for modelling the impact of vegetation on coastal flooding due to extreme events. This project uses GPUSPH, SPH implemented on graphical processing units (GPU), to begin studying the impact of vegetation by validating the model for flow around one square cylinder obstruction. The project also determines the importance of choosing a particle spacing to provide the most accurate results. For this case using single precision hardware, a particle spacing of 0.007 m to 0.010 m is most accurate when compared to the experimental data.**

I. INTRODUCTION

Understanding the effect of vegetation on flooding and water wave dissipation during extreme events such as hurricanes and tsunamis is important in protecting coastal communities and environments. While this effect can be complicated to measure, numerical models assist scientists and engineers in understanding the flow structure caused by the coastal vegetation. To fully understand this flow phenomenon, researchers must first study the flow dynamics around one vegetative stem using numerical models and then apply this knowledge when studying flow through the vegetative field.

A technique to simulate one vegetative stem is to model a flow encountering a single obstruction. Similar to this technique, Yeh and Petroff [1] completed an experimental study of a single bore impacting and flowing around a square cylinder obstruction. While the obstruction in this test is square instead of the usual circular cylinder model for vegetation, it provides a basic setup to model and test the Smoothed Particle Hydrodynamics (SPH) code. Gómez-Gestiera and Dalrymple [2] used the study to validate the SPH method using SPHysics and showed that SPH can produce accurate results for this type of flow.

Since these numerical tests were performed, GPUSPH, SPH implemented on graphical processing units (GPU), has been created. Hérault, Bilotta, and Dalrymple [3] discuss the implementation of SPH on GPU and notes that they were able to use more than 100 times more particles with GPU than the approximately 35,000 particles in Gómez-Gestiera and Dalrymple [2] with the traditional central processing unit (CPU). The authors also discuss the computing logistics in more detail showing the many advantages of using the GPU [3].

In this paper, the Yeh and Petroff experiment was used to validate GPUSPH for this flow and to also determine the effects of particle spacing on the results. One of the parameters specified in any SPH model is the initial spacing between the moving grid points known as particles. It is expected that smaller particle spacings would provide more accurate results because there would be more grid points to capture the fluid flow structure. This statement is not necessarily true as will be seen in the results, but the initial particle spacing significantly impacts the accuracy of the numerical results.

II. SETUP

To test the GPUSPH model and study the effect of the particle spacing, the dam break experiment by Yeh and Petroff [1] was used. The numerical tank was initiated as an open-top cube with dimensions 1.6 m long by 0.67 m wide by 0.4 m tall, representing the x, y, and z directions, respectively. Water filled the tank from 0.0 m to 0.4 m in the x and z directions, and across the whole width as indicated by the gray box in Fig. 1. In addition to the initial block of water on the side, the remainder of the tank was filled with one to two layers of particles, depending on the particle spacing, to simulate the residual water from the experiment. A quadratic weighting kernel was used for the fluid calculations.

Semi-analytic Approach for Treating Fixed and Mobile Boundaries: Validation Cases

Roberto Guandalini, Giordano Agate,
Andrea Amicarelli
Environment and Sustainable Development Dept.
RSE S.p.A.
Milan, Italy
roberto.guandalini@rse-web.it

Sauro Manenti, Mario Gallati, Stefano Sibilla
Dept. of Hydraulic and environmental Engineering
University of Pavia
Pavia, Italy
sauro.manenti@unipv.it

***Abstract*— This paper shows some validation cases of the improved SPHERA code for treatment of 3D irregular boundaries, both fixed and mobile, by means of the semi analytic approach. In the first case the jet originated by the collapse of a water column impacts on a rigid fixed box at the bottom of a tank. In the second case the simulation of a wave generator in a 3D experimental tank has been carried out. Finally, in the third case a transient wave has been simulated in the condition of the benchmark "Tsunami Wave Attack on Beach" based on the experiment realized at the Oregon State University. The semi analytic technique has been successfully applied in all the cases, and the results show a satisfactory degree of accuracy. The simulation of the effects of moveable structure allows to extend the field of application of the SPHERA code to several hydraulics problems having strong environmental impact.**

I. INTRODUCTION

When the SPH method was applied to confined fluid-dynamics [1], and in particular to free-surface hydrodynamics [2], several approaches for treating boundary conditions were introduced., as the ghost particles, the boundary particles, the boundary forces and the boundary integrals. Each of these approaches, however, introduces limitations and drawbacks, in particular for 3D problems with very irregular boundaries. In the SPHERA code, developed in the frame of a project funded by the Research for the Italian Electric System Fund [3], a general semi-analytical technique which overcomes the above mentioned limits has been included [4].

In this approach, the portions of the solid boundary which contribute to the mass and momentum equations of the generic particle can be replaced, just as in the ghost particle technique, by a fluid region which extends beyond the solid boundary: this virtual fluid region is treated as a material continuum (i.e. composed of an infinite number of particles) with a suitable distribution of velocity, density and pressure. The boundary contributions in the balance equations of each particle result therefore in integral terms extended to the portion of the sphere of influence of the particle laying beyond the boundary surface. These integral terms can be computed analytically and their value depends only on the geometry of the problem.

The consideration of the semi-analytical approach for the boundary treatment, however, plays an important role in the simulation of a number of different problems where the solid boundaries affect the evolution of the phenomena. In addition, the last code release includes also a number of improvements, both in terms of physical models and of computational algorithms [5]. One of the most important is the model of the mobile solid boundary, specially designed for the simulation of very fast movements. An other important new feature of the code is the consideration of the Shields shear stress model in addition to the classic Mohr-Coulomb criteria [5].

Therefore, a number of validation cases has been considered in order to quantify the performance and accuracy gain with respect the previous releases and to verify the correctness of the innovative models. More precisely, three different problems have been investigated:

1) *a classic dam break wave problem;*
2) *the simulation of a wave generator;*
3) *the simulation of a tsunami wave attack on beach.*

The results obtained in the SPHERA simulations are discussed in comparison with experimental and reference data, with special attention to the possible extensions to engineering application fields other then those for which the code has been initially designed.

II. THE DAM BREAK CASE

A comparison between numerical and experimental data for the dam-break wave problem, developed by Kleefsman et al. [6] and defined by the SPHERIC ERCOFTAC Special Interest Group (Fig. 1), have been already considered in order to prove the validity of the semi-analytical approach in [4], but it has been proposed again with the main goal of quantify the performance gain of the last code release. Mainly, the same case has been simulated considering the Shield stress model and with 85400 and 676500 fluid particles respectively. The comparison between numerical and experimental data are shown in Fig. 2 for different experimental frames, showing a good agreement. The improvement of accuracy obtained using a large number of fluid particles results from the analysis of the Fig. 3, where comparisons for the gauges P1 and P5 are shown. At gauge P1 the comparison shows that in both analyses a slight

Modelling armour block sea defences with SPH on GPUs

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

B.D. Rogers[1], A.J.C.Crespo[2], P.K. Stansby[1]
[1]School of Mechanical, Aerospace and Civil Engineering
University of Manchester, Manchester, UK
[2]EPHYSLAB
Universidad de Vigo, Vigo, Spain

***Abstract*—Smoothed particle hydrodynamics (SPH) is used to simulate flows over armoured breakwaters to determine the numerical value of the equivalent surface roughness in the absence of overtopping. Results are presented for simulations of regular waves for the Zeebrugge geometry from the European CLASH database. The new GPU-based code DualSPHysics is used to perform a preliminary analysis in a 3-D domain. A previous study has been performed in 2-D, using the open source SPHysics code, with good results [9]. The numerical results are compared with experimental data for the ratio of the runups of the armoured breakwater and a smooth face breakwater for two different armour units: (i) cubes and (ii) antifer blocks. With the current SPH formulation in DualSPHysics, the 3-D surface equivalent roughness is overpredicted. The particle size and the boundary conditions are key parameters affecting the results.**

I. INTRODUTION

Wave run-up is one of the most important factors in the design of coastal structures exposed to wave attack. It is a largely complex and nonlinear wave phenomenon, depending on geometrical, structural and hydraulic parameters. In breakwater design, if the maximum wave run-up height is greater than the structure freeboard overtopping of the structure occurs, affecting the security and functionality. Therefore the wave run-up height, as well as the mean overtopping rate, is a key parameter in the design of the coastal structures.

Slope equivalent roughness is one of the principal design parameters affecting run-up and overtopping flows over sea defences such as breakwaters. This expresses the armour units layout, shape and size, and hence the porosity of the seaward layers. An appropriate value of slope equivalent roughness based on detailed experimental and numerical validation could lead to a significant reduction of run-up height and overtopping flows without unnecessary expensive designs, therefore efficiently preventing risks to people, vehicles, ships and structures.

Simulation of coastal protection structures such as breakwaters using numerical schemes is very challenging task due to the complex geometries and violent hydrodynamics. Numerous studies have been undertaken examining the use of numerical schemes to model accurately overtopping rates, but modelling armoured breakwaters where the armour units may move and fail is still to be modelled comprehensively. The Smoothed Particle Hydrodynamics (SPH) method can address these issues due to its meshless Lagrangian approach.

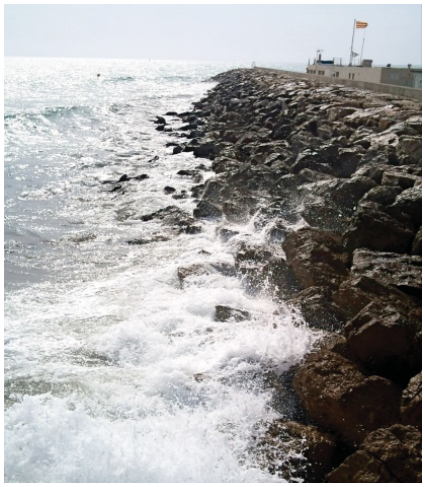

Figure 1. Example of a real sea defence, Sitges Coast (Spain).

First, however, before the combined effects of overtopping can be modelled, the ability of a numerical method to predict accurately the slope equivalent roughness needs to be assessed. This is the scope of the present work where the 3-D GPU-based DualSPHysics code [1] simulates flows over an armoured and impermeable core breakwater. The analysis is based on a previous study which has been performed in 2-D [2], using the open-source parallelSPHysics Fortran code [9]. By matching the porosity of armour layer, the application of SPH code to assess overtopping and run-up and stability for armoured structures could became a very promising tool to use as support to empirical formulae and Neural Network tool in the design of jetties or coastal defences.

This paper is organised as follows. In Section 2, we describe briefly the DualSPHysics code. In Section 3, we then present the different three-dimensional geometries, which match both the shape and porosity of the prototype structures where the breakwater roughness of the outer layer is modelled using objects that have the capability to move representing concrete cubes. In Section 4, results for the breakwater surface roughness are compared with data from Zeebrugge prototype tests which form part of the CLASH database [3] and with 2-D numerical SPH previous results. Finally, in Section 5, we draw conclusions and present some recommendations for future research.

Prediction of Ship Motions for a Wigley Hull Using SPH

Mark Pearce, Dr. Kenji Takeda, Dr. Dominic Hudson
School of Engineering Sciences
University of Southampton
Southampton, United Kingdom
mgp104@soton.ac.uk, ktakeda@soton.ac.uk, dominic@soton.ac.uk

***Abstract*—This paper outlines a framework that will allow SPH to be used to simulate the motion of a Wigley hull. A number of modifications and novel additions to the open source SPH code SPHysics are proposed to enable this simulation to be carried out. These are a modified wavemaker, a novel non-reflecting open boundary condition and the inclusion of the Wigley hull geometry. The results from two validation test cases are used to verify the capability of these new methods. Firstly a plunger dropped into a wave tank to test the non-reflecting open boundary. Secondly a Wigley hull in regular waves to demonstrate the combination of new techniques applicable to ship motion simulation.**

I. Introduction

Historically ship motion computer simulations have been carried using mature methodologies such as the Strip Theory and Reynolds Averaged Navier-Stokes (RANS) methods. However, the ever increasing power of new computers has made it possible to investigate using alternative numerical approaches to model ship simulations. In this paper we develop the SPH method to extend its capability to model ship motion in open water. For a methodology to be accepted it must be validated against a set of empirical or experimental data. A generic ship hull is often used in these validation studies to allow fair comparisons to be made with other methodologies.

The experiments used in this paper to validate the method are for the well-known Wigley Hull wave tank tests as described in [1,2]. The hull is subjected to regular head waves at different Froude numbers (*Fn*) and wave frequencies. A number of experimental situations are carried out including modelling the ship motion (in pitch and heave only) and also calculating the forces on a restrained hull. These experimental results are valuable for validating computational simulations, such as RANS solvers [3].

To conduct these validation simulations the SPHysics open source SPH code has been used ([4]). However, some modifications were required to enable the simulations to be conducted accurately. In addition 2D simulations were carried out to test the new features, requiring a further validation case. The case selected was that generated by a plunger dropped into a wave tank, described in [5]. This paper compares sets of RANS and SPH simulations to experimental data of the resulting waves. This paper has the additional benefit of allowing the comparison of SPH simulations to RANS simulations.

II. Formulations And Methodology

To simulate a wave tank and Wigley hull a number of changes and additions have been made to the original SPHysics version 2. To enable the required wave height, wave frequency and flow speed to be controlled a new wavemaker method has been created. The rigid floating body scheme, which in SPHysics version 2 is designed for simple geometries, has been modified to allow the Wigley hull form to be correctly modelled. Finally a new non-reflecting boundary condition based on the formulations proposed in [6] has been added.

A. SPHysics

A full account of the features available in SPHysics version 2 can be found in [7], in this section only the original formulations that have been altered for this project will be discussed.

There are two solid wall boundary conditions that are available in SPHysics, dynamic and repulsive, a modified version of the repulsive boundary condition is used in this paper. The repulsive boundary condition uses a sheet of particles to define the boundaries of a solid object, these particles impart a repulsive force when a fluid particle approaches the boundary. The force is defined by a delta function. This force is further modified by a fluid depth and a particle velocity correction.

The two types of wavemaker in SPHysics both involve a moving wall which is defined by a line of boundary particles. The first type is a piston that translates laterally to describe an oscillatory motion, the second is a paddle that is fixed at one end and oscillates. The motion of the wavemaker is defined by setting the period and amplitude. These variables allow the frequency and height of the generated waves to be prescribed. However, in order to simulate the test cases in [1] and [2] the flow speed also needs to be defined; this means that a new wavemaker is required that can include a non-zero free stream velocity.

B. Altered and Novel Methodologies

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

The first alteration was to the repulsive boundary condition. It was found, by moving a single particle towards a solid boundary (as in [7]), that with sufficient closing speed a fluid particle could penetrate the boundary.

Simulating ship wave patterns through a 3D parallel SPH code

S. Marrone, B. Bouscasse, A. Colagrossi
CNR-INSEAN
Rome, Italy
s.marrone@insean.it, b.bouscasse@insean.it, a.colagrossi@insean.it

***Abstract*— An analysis of the 3D wave pattern generated by a ship in stationary forward motion is performed with a particular focus on the bow breaking wave. For this purpose a new 3D parallel SPH solver has been designed. An assessment of the capabilities of the 3D solver to perform wave breaking simulations is preliminarily performed on a test case specifically conceived. The numerical effort to simulate the complete evolution of the flow around the ship with sufficient resolution is considerable in 3D. Consequently, an ad hoc hybrid MPI-OpenMP parallelization has been developed to achieve simulations of hundred million of particles running on a computer cluster. The outcomes are compared with experimental measurements in terms of wave height and features of the bow wave.**

I. INTRODUCTION

The paper deals with the problem of a ship in steady-state forward motion with particular reference to the possibility of predicting the extent of breaking bow wave. From the physical point of view, it has been crucial for a long time to understand how bow waves are generated, and how they modify the flow field. The purpose was to improve the evaluation of the resistance. The progress given by the numerical and experimental study dedicated to the breaking phenomena have made possible to apprehend its effect on ship visibility.

At present, some numerical models for free-surface flows, in particular RANSE with level-set algorithm, are able to catch 3D breaking phenomena in bow waves (see for example Carrica et al. [1] and Di Mascio et al. [2]). SPH simulations have proven to be reliable over 2D breaking scenario. On the specific topic of bow waves, Colagrossi & al [3] proposed a 2D+t SPH model, and the computed flows qualitatively matched the breaking scenario. In Colagrossi [4], the prescriptions about the minimum discretization required to catch breaking inception were deeply investigated. In the present work, a 3D SPH solver is developed and tested in order to carry on the 3D wave pattern computation and the physical characterization of the bow wave.

The activity is complex, in fact simulations with varying spatial discretization require a *h-variable* formulation, see [5] and [6].

An assessment of the capabilities of the 3D solver to perform wave breaking simulations is performed on a test case specifically conceived. From the analysis of such test the minimum discretization necessary to describe the plunging jet is obtained and compared to previous 2D results from 2D+t analysis. The numerical effort to reach such a resolution in 3D for the ship bow wave is considerable because the fluid domain has to be large enough to simulate deep water behaviour and avoid wave reflections at the boundaries. The number of particles needed for these simulations is of the order of 10^8, therefore, an hybrid MPI-OpenMP parallelization has been developed. The SPH results obtained are then compared with experimental measurements and with simulations obtained by a RANSE solver coupled with a Level Set algorithm.

II. NUMERICAL SCHEME

A. SPH model adopted

The standard weakly-compressible SPH scheme is generally affected by some drawbacks. The main problems are the correct evaluation of the pressure values for strong impacts and the flickering of pressure profiles that may occur also in non-particularly violent flows. This is related to the fact that, in general, the SPH predicts kinematics very well but, conversely, large random pressure oscillations are present due to numerical high frequency acoustic signals.

Antuono et al. [7] added a numerical diffusive term in the density equation which smoothes out the numerical noise inside the pressure field. Further the model was proved to be consistent and convergent all over the fluid domain. Such a model is hereinafter referred as δ-SPH.

The discrete δ-SPH scheme proposed by Antuono et al. [7] reads:

$$\begin{cases} \dfrac{D\rho_i}{Dt} = -\rho_i \sum_j \left(\boldsymbol{u}_j - \boldsymbol{u}_i\right)\cdot\nabla_i W\left(\boldsymbol{r}_j\right)V_j + \delta h c_0 \sum_j \boldsymbol{\psi}_{ij}\cdot\nabla_i W\left(\boldsymbol{r}_j\right)V_j \\ \rho_i \dfrac{D\boldsymbol{u}_i}{Dt} = -\sum_j \left(p_j + p_i\right)\nabla_i W\left(\boldsymbol{r}_j\right)V_j + \rho_i \boldsymbol{f}_i + \alpha h c_0 \rho_0 \sum_j \pi_{ij}\nabla_i W\left(\boldsymbol{r}_j\right)V_j \\ p_i = c_0^2\left(\rho_i - \rho_0\right) \\ \dfrac{D\boldsymbol{r}_i}{Dt} = \boldsymbol{u}_i \end{cases}$$

Coupling SPH with a 1-D Boussinesq-type wave model

Christophe Kassiotis*, Martin Ferrand†, Damien Violeau*, Benedict D. Rogers‡,
Peter K. Stansby‡ and Michel Benoit*

*Laboratoire d'Hydraulique Saint-Venant,
Université Paris-Est (Joint Research Unit EDF R&D / CETMEF / École des Ponts ParisTech),
Email: christophe.kassiotis@enpc.fr
†MFEE, EDF R&D, Chatou, France
‡School of MACE, University of Manchester, Manchester,UK

***Abstract*—The high computational cost of SPH remains problematic in dealing with wave propagation, especially when the domains considered are large. In order to overcome this difficulty, we propose to couple 2-D SPH with a 1-D Finite Difference Boussinesq-type model. The latter deals with wave propagations for most of the spatial domain, whereas SPH computations focus on the shoreline or close to off-shore structures, where a complex description of the free-surface is required.**

The re-use of existing codes is achieved using a generic implementation based on Component Technology. The communication between software is ensured by the middleware Component Template Library (CTL) [1], [2]. In order to deal with open domains, open-boundaries have to be implemented for SPH, with water height and velocity varying in space and time. These velocity and water height values are then driven by the Boussinesq-type model.

As an illustration of the one way coupling, we present herein two simple examples of water waves, the first one with a flat bottom, the other one representing a schematic coastal protection.

I. Introduction

The recent events in Japan have dramatically underlined the necessity of accurate predictions for coastal protections. For this kind of civil engineering coastal devices, it is required to compute the flow near the shoreline, where the waves break. But even if the recent trends in the development of GPU-based SPH software seem promising for the computing of large 3D domains, to compute wave propagation on the scale of oceans with SPH is still beyond current computational capabilities. Furthermore, it is often useless, as simplified models are able to represent accurately the wave propagation on most of the domain.

Indeed, the complexity of flows, and especially those at large scales such as wave propagation across ocean make the introduction of simplified models a natural development [3]–[5]. Since the XIXth century, and through the XXth, models such as Saint-Venant [6], Boussinesq [7], [8] and fully Non-Linear Shallow Water equations (NLSW) [9], [10] provide satisfying results in their respective ranges of application (from deep to shallow water).

They are however, by definition, unable to represent accurately the complexity of the flow near the coast, when waves are breaking. The violent hydrodynamics of the wave is often handled by energy dissipation models in the nearshore region (e.g. roller models or sponge layers), often with coefficients that need to be tuned for specific cases. One such example is a study that compares the results from analytical,NLSWE software and two-phase slightly compressible flows solved by VOF strategy for the classical dam break problem [11], and shows the necessity of advanced models for this kind of application.

The development and implementation of appropriate models however is to represent the complex free-surface, evolving in time, with a possible multi-connected domain. Among all the options now available, Smoothed Particle Hydrodynamics (SPH) is offers one of the most attractive approaches. However, 3-D SPH models still often suffer from damping with the waves being dissipated before reaching the coast if no proper treatment is applied.

For these reasons the coupling between any of the wave propagation models and complex free-surface flow strategies seems appropriate to tackle this problem [12], [13]. One of the motivations for this work is to re-use existing codes in order to avoid the long development and validation phase.

The outline of this paper is as follows: in the next section, we present the chosen formulation of the SPH numerical model. In particular, a semi-implicit wall boundary condition is used, as presented in this conference in [14]. In Section III, we detail the Finite Difference Boussinesq-type model used hereafter. The coupling algorithm, the open-boundary conditions required for SPH and the communication between software are detailled in Section IV. In Section V, we present the results of preliminary computations and in Section VI, we finish with some conclusions.

II. SPH Numerical model

A. Continuous equations

We consider a turbulent weakly compressible free-surface flow. The velocity vector, pressure, turbulent kinetic energy

3D long-wave runup with GPUSPH, nee GPU-SPHysics

Robert Weiss & Andrew Munoz
Department of Geology and Geophysics
Texas A&M University
College Station, USA
weiszr@tamu.edu

Robert A. Dalrymple
Department of Civil Engineering
Johns Hokins University
Baltimore, USA

Alexis Hérault
Mathematical Engineering Department
Conservatoire National des Arts et Metiers
Paris, France

Eugenio Rustico & Giuseppe Bilotta
Dipartimento di Matematica e Informatica
Universita di Catania
Catania, Italy

[1] ***Abstract*—Tsunamis need to be studied more carefully and quantitatively to fully understand their destructive impact on coastal areas. Numerical modeling provides an accurate and useful method to model tsunami inundations on a coastline. However, models must undergo a detailed verification and validation process to be used as an accurate hazard assessment tool. Using standards and procedures given by NOAA, a new code in hydrodynamic modeling called GPUSPH can be verified and validated for use as a tsunami inundation model. GPUSPH is a meshless, Lagrangian code that utilizes the computing power of the Graphics Processing Unit (GPU) to calculate high resolution hydrodynamic simulations using the equations given by Smooth Particle Hydrodynamics (SPH). GPUSPH has proven to be an accurate tool in modeling complex tsunami inundations, such as the inundation on a conical island, when tested against extensive laboratory data.**

I. Background

A. Motivation

Since the 2004 Sumatra event, tsunamis have received increased attention at all levels of government. Not surprisingly, assessments of the tsunami hazard for coastal communities have been initiated and tsunami-warning systems have been extended or newly developed. For USA coastlines, Dunbar and Weaver in [1] compiled a report on tsunami hazard assessment. From a scientific point of view, this report highlights two major items. First, that numerical modeling of tsunami generation, propagation, and inundation needs to be included in hazard assessments to predict the environmental and social impact of future events more robustly. Second, that geological information needs to be included for a reliable assessment because large earthquakes may have recurrence periods that are larger than the recorded history in different regions, given the heterogeneous settling history of the North American continent. In most areas, tsunamis leave sedimentary deposits that serve as forensic evidence of events; meaning that such sedimentary layers serve as records of past events. The relevance of tsunami hazard assessments, especially in a probabilistic sense, is described in [2]. However, the sedimentary layers not only record the incidence of a tsunami event, they also contain information on the causative tsunami. As not all tsunami deposits are alike, the complex interactions of the tsunami wave with the movable bed and geometry of the bathymetry cause a generation of specific features of deposits that are a function of the number and characteristics of the incident waves as well as their withdrawal. If deciphered, the information on the tsunami characteristics can be inferred from deposits if –most importantly– the mechanics of the transport, erosion and deposition processes are understood. Our motivation is to tackle the complex physics of the sediment transport processes due to tsunamis and contribute in this way to a more meaningful numerical modeling of past events that will make tsunami hazard assessments more robust and more reliable.

B. Classical Tsunami Modeling

As every problem in fluid mechanics, the dynamics of tsunamis obeys Newtons Second Law of Motion, and can hence be described with the Navier-Stokes equations. Depending on the processes and the scale these processes operate, the Navier-Stokes and Euler equations can be simplified. For example, for tsunami wave propagation without nonlinear effects, such as dispersion and breaking, the Euler equations can be integrated over depth to arrive at nonlinear and linear versions of the Shallow Water equations. With these equations, non-breaking or weakly breaking waves can be modeled, as demonstrated by Titov and Synolakis in [3] and [4], with the MOST code. The Weather Service of the National Oceanic and Atmospheric Administration (NOAA) uses the MOST code for the early tsunami warning system for Pacific Ocean and US coastlines. However, if processes that operate on a smaller physical scale are considered, such as dispersion or breaking,

[1]This manuscript broadly is a reproduction of Weiss et al. *Three-Dimensional Modeling of Long-Wave Runup: Simulation of Tsunami Inundation with GPUSPH*, Proceedings Of The International Conference On Coastal Engineering, No. 32(2010), Shanghai, China. Paper 1432: currents.8, retrieved from http://journals.tdl.org/ICCE/ .

Rigid bodies moving in a stratified fluid

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

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

***Abstract*—We present SPH simulations of fluids with density stratification interacting with rigid bodies. The stratification in these simulations is in a stable configuration, with the less dense fluid above the denser fluid. The lower fluid is typically 5-20% more dense than the upper fluid, which is similar to the density ratio of fresh and salty water. The SPH algorithm is simple, accurate and robust. It easily handles different boundaries: free surfaces, periodic cells, and rigid walls (with arbitrary shape). These small density ratios are modelled with a straightforward and novel approach, but a technique for modelling higher ratios is also discussed. The rigid boundaries are modelled using boundary force particles [1]. We present a series of applications: a) surface waves in a tank of fluid; b) waves generated at the interface by a moving cylinder, and c) the effect of stratification on the motion of a fish-like swimmer [2], [3] near the interface.**

I. Introduction

This study continues the series of work on modelling fish-like swimmers using SPH [2]–[4]. We present the preliminary results for a swimmer moving in a stratified fluid. The aim of this work is to determine whether there are any speed or energy advantages when moving at or near a stratified layer. The typical stratification fish encounter in nature is bodies of water where a fresh water layer sits upon a saline layer. The density variation can be of the order of 5-20%.

The mechanism that can have a possible effect on swimming motion is the so-called 'dead-water' phenomenon. The effect is occasionally experienced as an abnormal resistance on the motion of ships near the Arctic circle. It was carefully studied and described over one hundred years ago by Ekman [5]. The phenomenon manifests typically when a fresh water layer sits on top of a saline layer. It is understood that as ship moves along the surface, a wave is generated at the interface between the two layers, which then creates a drag force. The effect was shown experimentally by Vasseur et al. [6].

In order to test the accuracy of the SPH algorithm, we ran preliminary tests for waves and stratified fluids. In the first test, a standing wave on a free surface is considered. The results are compared to the theory of Lamb [7]. In the second test, a cylinder is moved through the upper layer of a two-layer stratified fluid. The lengths of the waves generated at the interface are compared to the experimental study of Carpenter and Keulegan [8]. Finally, we model a fish-like swimmer moving at a range of distances from the interface of a two-layer fluid. Our aim is to determine whether the waves generated at the interface by the swimmer impact on the swimming speed or energy expenditure.

II. Governing equations

The continuum equations we solve are the Navier-Stokes equations, with boundaries formed by parts of rigid bodies and sections of skin. Apart from the introduction of the skin, the continuum equations are the same as those given by Kajtar and Monaghan [2].

A. SPH equations

In the following, the labels a and b are used to denote SPH fluid particles, label j is used for the boundary force particles on the rigid bodies, and label s is used for the skin particles. The equation of motion for fluid particle a is given by

$$\frac{d\mathbf{v}_a}{dt} = \mathbf{F}_a(\mathit{fluid}) + \mathbf{F}_a(\mathit{body}) + \mathbf{F}_a(\mathit{skin}) + \mathbf{g}_a, \qquad (1)$$

where

$$\mathbf{F}_a(\mathit{fluid}) = -\sum_b m_b\ \sigma_{ab}\nabla_a W_{ab}, \qquad (2)$$

$$\mathbf{F}_a(\mathit{body}) = \sum_j m_j \left(\mathbf{r}_{aj} f(r_{aj}) - \sigma_{aj}\nabla_a W_{aj}\right), \qquad (3)$$

$$\mathbf{F}_a(\mathit{skin}) = \sum_s m_s \left(\mathbf{r}_{as} f(r_{as}) - \sigma_{as}\nabla_a W_{as}\right), \qquad (4)$$

and

$$\sigma_{ab} = \frac{P_a}{\rho_a^2} + \frac{P_b}{\rho_b^2} + \Pi_{ab}. \qquad (5)$$

In these equations the mass, position, velocity, density and pressure of particle a are denoted m_a, $\mathbf{r}_a$, $\mathbf{v}_a$, ρ_a, and P_a respectively. The gradient ∇_a is taken with respect to the coordinates of particle a. $\mathbf{F}_a(\mathit{fluid})$ is the pressure and viscous force per unit mass due to the other fluid particles. $\mathbf{F}_a(\mathit{body})$ is the force per unit mass due to the rigid bodies, which consists of two parts. The first term, involving $f(r_{aj})$, is a repulsive boundary force. The second is a direct pressure interaction which is a result of deriving the equations of motion from a variational principle using the continuity equation as a constraint. The force per unit mass due to the skin particles $\mathbf{F}_a(\mathit{skin})$ is identical to $\mathbf{F}_a(\mathit{body})$, except the summations are over skin particles. The final term on the right hand side of (1) denotes the gravitational acceleration.

SPH Simulations of Pedestrian Crowds

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

***Abstract*—The simulation of crowds in emergency situations, e.g. evacuation of passenger ships or underground stations, has recently gained increasing scientific interest. Such simulations are used to analyse and optimise the design of escape-route elements in order to ensure a successful evacuation process. The paper reports the development of an SPH-based crowd-simulation model and its application to normal and evacuation situations. The employed rationale refers to a generalised behavioural-force model to describe the interactive pedestrian dynamics. The distinct features of the approach refer to the inherent three-dimensionality in combination with the microscopic computational model. The latter supports a detailed design analysis and can also be coupled to fluid-dynamic properties (water, stack-gas) obtained from the same simulation tool. Validations and applications refer to phenomena which are difficult to capture with non-microscopic or planar crowd-simulation techniques. Benchmark examples involve lane-pattern formation (aka. self-organization) studies at variable pedestrian density, escape from a single room, as well as simulations of the attainable walking speed on stairs which are a critical path in multi-floor facilities. Finally, a more complex application to an escape-route section is presented. Examples included reveal that the model is able to simulate evacuation scenarios of practical interest with reasonable accuracy.**

I. Introduction

The behaviour of individuals inside pedestrian crowds is of interest for their safety in emergency situations. Shopping-malls, sport arenas, office-buildings, cruise-liners etc. are designed for an ever increasing amount of people at high pedestrian density. The dynamics of large groups at high density is thus present in many peoples normal cruise of life. Moreover, the public safety-awareness with respect to evacuations has continuously been reinforced by recent tragedies, e.g. the capsizing Baltic Sea ferry MV ESTONIA (1994, 852 casualties) or the Hillsbrorough Stadium disaster in Sheffield (1989, 96 casualties).

Fortunately, first-hand disaster experience is rare and evacuation simulations are perhaps the only option to analyse and secure the compliant evacuation performance of a new, non-prescriptive assembly space with respect to its capacity and design/layout. Simulations are nowadays used to scrutinise the access of rescue zones and optimise the design of escape routes.

The dynamics of pedestrian crowds and its simulation is nontrivial. It involves social & physical interaction between humans as well as their response to external factors (e.g. fire, visibility) and layout aspects. Similarities to fluid dynamics exist, e.g. between contact avoidance and pressure-driven fluid displacement, but some fundamental aspects of Newtonian mechanics, e.g. the adherence to the reaction principle, are also missed. In general, the pedestrian dynamic distinguishes between normal and panic situations. Different modelling strategies exist to simulate the behaviour of individuals (aka. agents) in normal or panic situations. They involve less elaborate empirical & observation-based methods [15], more detailed macroscopic gas-kinetic or fluid-dynamic approaches [9] and complex many-particle continuous-space [7] or cellular-automata discrete-space techniques [11].

The present paper is concerned with particle-based simulations of pedestrian crowds. The computational approach is based upon detailed microscopic models for the behaviour of each agent in continuous space. It utilises social forces to determine the agent movement in a particle framework along a route outlined by Helbing et al. [6], [7]. As opposed to macroscopic (fluid-type) models, the microscopic model inheres non-conservative interaction terms, e.g. in conjunction with avoidance/territorial effects, and simulates the individual pedestrian motion using a momentum equation. Transition between normal and panic situations is managed using a nervousness parameter, which primarily feeds back to the desired speed [8].

Distinct from other many-particle simulations, the present approach aims to set-up an SPH-model for the agent interaction. The kernel-based SPH-representation reveals benefits with respect to the computational efficiency. Due to the inherent smoothing procedure, it features an enhanced macroscopic nature. The model is thus less prone to subtle parameter details and related tendencies towards an unrealistic behaviour prognosis for isolated agents displayed by many-particle simulations. In view of an integrated evacuation simulation, the approach can easily be coupled to fluid-dynamic particle simulations (e.g. for the evolution of stack gas). The employed SPH-model uses a dynamic kernel width and inheres features of a modified social-force formulation recently suggested by Lakoba et al. [14].

The implementation is based upon the GADGET-$^{H}2^{O}$ procedure [4], [5] which is a modification of the massively-parallel astrophysics GADGET-2 SPH code of Springel [12] dedicated to engineering multi-phase flows. Due to the large kernel widths associated to the interaction of agents, a specific SPH wall-model outlined by Lehnart et al. [1] has been utilised. This wall model avoids interaction across walls, while maintaining the software's existing wall-particle representation.

SPH Modelling of Fragmentation of Electromagnetically Driven Rings

T. De Vuyst, J. Campbell, R. Vignjevic
Crashworthiness, Impact and Structural Mechanics Group, School of Engineering
Cranfield University
Cranfield, MK43 0AL, UK
t.devuyst@cranfield.ac.uk, j.campbell@cranfield.ac.uk, v.rade@cranfield.ac.uk

***Abstract*—Meshless methods, such as Smoothed Particle Hydrodynamics (SPH), are of particular interest for the accurate prediction of fragmentation and fracture at high strain rate in metals. This paper describes research on the prediction of necking and failure of electromagnetically driven rings using the SPH method. The experimental data by Zhang and Ravi-Chandar [1] is used to evaluate the accuracy of the model predictions. The model developed in this paper evaluates the Eulerian and Total Lagrangian SPH formulation. A Johnson-Cook plasticity model is used in combination with a Lemaitre damage model to describe the plastic deformation and fracture of the rings. The effect of Joule heating due to the current induced in the ring is taken into account in the constitutive model. The acceleration due to the ring currents has been implemented in Cranfield's SPH code as a body force.**

I. Introduction

Meshless methods, such as Smoothed Particle Hydrodynamics (SPH), are of particular interest for the accurate prediction of fragmentation and fracture at high strain rate in metals. This paper describes research on the prediction of necking and failure of electromagnetically driven rings using the SPH method.

In an electromagnetically driven ring experiment a metal ring is radially accelerated by electromagnetic force acting on the ring. These electromagnetic forces are generated by inducing a current in the ring by discharging a capacitor into a coil which sits inside the ring. A schematic overview of this setup is shown in Figure 1.

Several publications are available which present experimental results as well as numerical models of expanding ring experiments. In the following paragraphs a brief overview will provided of some of the relevant papers.

A. Electromagnetically Driven Ring Experiments

A widely cited paper on electromatically driven expanding ring experiments is by Grady [13]. This paper describes an electromagnetical expanding ring test method in detail and presents results for copper and aluminium rings. They show the influence of ring velocity on fracture strain and the dependence of fragment number on kinetic energy. The data in this paper is frequently used to validate numericla models of ring fragmentation. Satapathy and Landen [9] describe an electromagnetically driven expanding ring experiment. They study the effect of adiabatic heating, due to the current induced in the rings, on the mechanical properties during expansion, and compare this with equivalent isothermal properties. The paper does not discuss fragmentation of the rings. In a second paper [14] they extend their experiments to discuss the effect of (rapid) preheating on the mechanical properties and their effect on expansion of the rings. A good review of fragmentation models and experiments can be found in the paper by Zhang and Ravi-Chandar [11]. It is concluded in this paper that for fragmentation statistics to be captured appropriately models must account both for localization and for statistical variations in this localization. A series of experiments performed by these authors will be described in more detail in Section I-C since the simulation results presented in this paper are based the experiments performed by these authors.

B. Electromagnetically Driven Ring Models

Several models of the electromagnetically driven ring experiment have been published. Most models use Lagrangian FE code, although sometimes Eulerian hydrocodes or finite difference models are also used. The models range from simple 1D models to 3D models. The fragmentation stage is not always modelled with models restricting themselves to predicting the expansion speed and strain rate. Johnson J.N. [1] proposed a 1D model for expanding ring fragmentation. The fragmentation model is based on void growth and uses a intial random distribution of porosity. This 1D model is solved for a known strain rate history using the finite difference method. Hu and Daehn [8] also present a 1D finite difference model for dynamic tensile and expanding ring tests. They use a Holomon-type material law with power law starin-rate sensitivity. They analyse the effect of velocity, and parameters of the constitutive law (strain hardening exponent and strain rate sensitivity exponent) on ductility in tensile and expanding ring tests. The results compare well with experiment data from the literature. The analyses enable them to identify the Von Karman velocity as well as other critical velocities which determine the velocity at which inertial effects significantly affect ductility. Becker [10] presents 3D finite element modelling results of electromagnetically driven expanding ring experiments. The radial velocity of the ring is imposed during the acceleration phase. The model uses the Gurson constitutive

Consistent wall boundary treatment for laminar and turbulent flows in SPH

Martin FERRAND, Damien VIOLEAU
Electricité de France R&D
LNHE and MFEE departements
Chatou, France
damien.violeau@edf.fr

Benedict D. ROGERS, Dominique LAURENCE
School of Mechanical, Aerospace and Civil Engineering
University of Manchester
Manchester, UK

Abstract — A consistent wall boundary treatment for SPH has been developed in [1] based on a renormalizing factor for writing boundary pressure forces. This factor depends on the local shape of a wall and on the position of a particle relative to the wall, which is described by segments (in 2-D), instead of the cumbersome fictitious or ghost particles used previously. By solving a dynamic equation for the renormalizing factor, the authors have significantly improved traditional wall pressure treatment in SPH. The present paper aims to extend this method to wall friction and turbulent variables' boundary conditions, on the basis of the standard k–ε model. By using Gauss' theorem in a continuous SPH form of the fluid equations, all diffusive terms are re-written with boundary contributions. The latter are then discretized using particles (for the fluid) and segments (for the wall), leading to corrections for the strain rate and flux conditions of the dissipation of turbulent kinetic energy. This method yields consistent Von Neumann wall conditions for momentum, turbulent energy and energy dissipation. Two validations are presented: (*i*) a steady laminar flow in a closed channel and (*ii*) a turbulent steady flow in a periodic fish-pass, with comparison to a validated open-source Finite-Volume code. Both give very satisfactory results.

I. Introduction

The issue of boundary treatment in SPH has been widely addressed in the literature during the past decade, especially regarding wall treatment. The work presented in [2] provides an interesting framework to ensure solid wall impermeability from a variational principle. More recently, [3] and [4] proposed simpler approaches based on similar concepts, *i.e.* wall renormalization. There is, however, no clear model capable to handle arbitrary boundary conditions for all differential operators in SPH.

We extend here the concept of renormalization to derive a consistent boundary treatment for gradient, divergence and Laplacian discrete operators, based on our previous work [1]. We then apply this method to solve the Reynolds-Averaged Navier–Stokes (RANS) equations together with a standard k–ε model for turbulence closure. After stating the mathematical basics of our model, we present the discretized RANS and k–ε equations, before showing two applications including validation against theory and comparison to a Finite-Volume technique.

II. Mathematical model

A. Consistent wall boundary conditions in SPH

We first consider the following renormalized SPH interpolant:

$$[A]^\gamma(\mathbf{r},t)=\frac{1}{\gamma(\mathbf{r})}\int_\Omega A(\mathbf{r}',t)w(|\mathbf{r}-\mathbf{r}'|)d^n\mathbf{r} \qquad (1)$$

where A is an arbitrary field, n the spatial dimension, w the kernel and γ the integral of the latter:

$$\gamma(\mathbf{r})=\int_\Omega w(|\mathbf{r}-\mathbf{r}'|)d^n\mathbf{r} \qquad (2)$$

This integral extends over the entire physical domain Ω. Due to the compactness of the kernel support, however, it is restricted to the coloured area on Fig. 1.

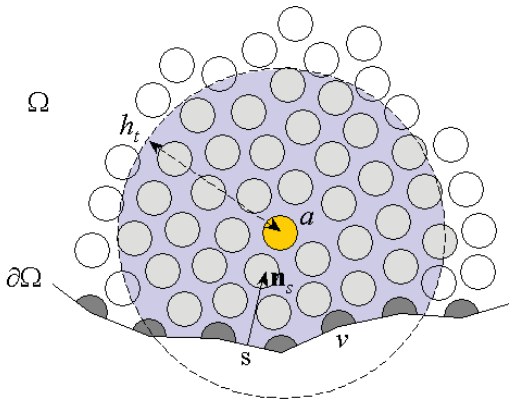

Figure 1. The discretization used in this work is based on usual particles (a or b), vertex particles (v) and wall segments (s) (in two dimensions). The truncation of the kernel is considered through the integral γ (coloured arear).

The gradient of the normalizing function is

$$\nabla\gamma(\mathbf{r})=\int_{\partial\Omega} w(|\mathbf{r}-\mathbf{r}'|)\mathbf{n}'d^{n-1}\Gamma' \qquad (3)$$

On the non-slip boundary condition enforcement in SPH methods.

Leo M. González
School of Naval Architecture ETSIN
Universidad Politécnica de Madrid
Madrid, Spain
leo.gonzalez@upm.es

A. Souto-Iglesias
School of Naval Architecture ETSIN
Universidad Politécnica de Madrid
Madrid, Spain
antonio.souto@upm.es

F. Macià
School of Naval Architecture ETSIN
Universidad Politécnica de Madrid
Madrid, Spain
fabricio.macia@upm.es

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

M. Antuono
The Italian Ship Model Basin
CNR-INSEAN
Roma, Italy
matteoantuono@gmail.com

***Abstract*—The implementation of boundary conditions is one of the points where the SPH methodology still has some work to do. The aim of the present work is to provide an in-depth analysis of the most representative mirroring techniques used in SPH to enforce boundary conditions (BC) along solid profiles. We specifically refer to dummy particles, ghost particles, and Takeda *et al.* [1] boundary integrals. A Pouseuille flow has been used as a example to gradually evaluate the accuracy of the different implementations. Our goal is to test the behavior of the second-order differential operator with the proposed boundary extensions when the smoothing length h and other dicretization parameters as dx/h tend simultaneously to zero. First, using a smoothed continuous approximation of the unidirectional Pouseuille problem, the evolution of the velocity profile has been studied focusing on the values of the velocity and the viscous shear at the boundaries, where the exact solution should be approximated as h decreases. Second, to evaluate the impact of the discretization of the problem, an Eulerian SPH discrete version of the former problem has been implemented and similar results have been monitored. Finally, for the sake of completeness, a 2D Lagrangian SPH implementation of the problem has been also studied to compare the consequences of the particle movement.**

I. Introduction

The SPH simulations in Engineering involve usually solid boundary conditions (BC) for the velocity field and Dirichlet and Neumann type BC for other fields as, for instance, the temperature. In the SPH framework, these conditions are tackled in a number of ways: by using boundary forces-type models [2], [3] ; by modifying the structure of the kernel in the neighborhood of the boundaries [4]; by creating virtual particles inside the solid boundary domain through mirroring techniques. This latter approach is the main focus of the present work. An interesting study for the linear Couette and Pouseuille flows have been already performed in [5], but unfortunately the evolution of the kinetic energy was the only variable monitored in time.

In our case a well know problem as the Pouseuille flow will be used as a benchmark. The evolution of the velocity profile and the forces involved in the dynamics of the flow will be carefully studied.

II. Theoretical setup.

Before proceeding to the analysis, we briefly recall the principal results about the consistency of the continuous SPH formulation **without boundaries**. The fluid domain is $\Omega = \mathbb{R}^d$ and, therefore, its boundary is $\partial\Omega = \emptyset$.

Let $W(\mathbf{x}; h)$ be a function depending on $h > 0$ defined by

$$W(\mathbf{x}; h) := \frac{1}{h^d}\tilde{W}\left(\left|\frac{\mathbf{x}}{h}\right|\right), \tag{1}$$

We also define the function $F(r)$ as

$$F(r) := -\frac{1}{r}\tilde{W}'(r), \tag{2}$$

In the following we denote by $u(\mathbf{x})$ a smooth scalar field on $\mathbb{R}^d$.

For the approximation for the Laplacian of a function, the following formula due to Morris *et al.* [6] and Español *et al.* [7] is used:

$$\langle \Delta u \rangle_{\mathrm{M}}(\mathbf{x}) = 2\int_{\mathbb{R}^d} \frac{(\mathbf{x}' - \mathbf{x}) \cdot \nabla_{\mathbf{x}} W(\mathbf{x}' - \mathbf{x}; h)}{|\mathbf{x}' - \mathbf{x}|^2} \left[u(\mathbf{x}') - u(\mathbf{x})\right] d\mathbf{x}'. \tag{3}$$

As proved in [7], it follows:

$$\langle \Delta u \rangle_{\mathrm{M}}(\mathbf{x}) = \Delta u(\mathbf{x}) + \mathcal{O}(h^2). \tag{4}$$

A modified no-slip condition in weakly-compressible SPH

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

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

Abstract — **Modelling no-slip conditions by the ghost particle method in weakly-compressible SPH (WCSPH) is prone to instabilities, even at low Reynolds numbers (around 100). These instabilities are characterized by a breakdown of the particle distribution, leading to clumping. Such instabilities cause, at least, a drastic change in the flow description, and in the worst case, failure of the simulation.**

In this paper, an investigation of these instabilities is performed. A method and its justification are then proposed to overcome the difficulties involved. Validation is provided by studying the academic test case of Couette flow. More complex test cases are then presented, highlighting the benefit of the proposed change in boundary conditions.

I. INTRODUCTION

Modelling viscous flow in WCSPH requires using a precise and robust method to impose no-slip condition; otherwise the boundary layer simulated does not well represent the physical one, and can present numerical instabilities. The ghost particle method provides very good results to represent free-slip condition in the case of flat walls, but surprisingly presents significant difficulties in the case of no-slip condition. Indeed, already for moderate Reynolds (~100) this method generates numerical instabilities. These instabilities are characterized by a breakdown of the particle distribution, leading to clumping. Such instabilities cause a drastic change in the description of the flow, and can lead to the failure of the simulation. The difficulties involved are mentioned in the literature by e.g. Watkins et al. [7], and Basa et al. [8], and at previous SPHERIC workshops by e.g. Federico et al. [1], and Ramos-Becerra et al. [2].

In the present paper, a first study of the widely-studied Poiseuille flow permits to highlight these instabilities. An energy study further shows that the classical no-slip condition introduces energy in the hyperbolic part of the Navier-Stokes equations, which does not respect the physics of the problem. To recover a physical modelling a specific treatment is proposed for the hyperbolic part of the system. Its effectiveness is then verified by studying the value of the velocity divergence, especially at the wall.

Validation is then provided by studying the academic test case of Couette flow. A lid-driven cavity test case and the flow past a cylinder are then presented, highlighting the benefit of the proposed change in boundary condition.

II. GOVERNING EQUATIONS

The discretization of the Euler equations considered in this paper is the one proposed by Monaghan [3] :

$$\begin{cases} \dfrac{d\overline{x_i}}{dt} = \overline{v_i} \\ \dfrac{d\rho_i}{dt} = \displaystyle\sum_{j\in P(\Omega)} m_j \left(\overline{v_i} - \overline{v_j}\right).\overline{\nabla}_i W_{ij} \\ \dfrac{d\overline{v_i}}{dt} = -\displaystyle\sum_{j\in P(\Omega)} m_j \left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2}\right)\overline{\nabla}_i W_{ij} - \overline{g} \end{cases} \quad (1)$$

with $\overline{x_i}$, $\overline{v_i}$, m_i, ρ_i, p_i respectively the position, velocity, mass, density, and pressure of the particle i. $\overline{g}$ is the gravity vector. The gradient of the kernel is denoted:

$$\overline{\nabla}_i W_{ij} = \overline{\nabla}_{\overline{x_i}} W\left(\overline{x_i} - \overline{x_j}, h\right) \quad (2)$$

with h the smoothing length which is constant throughout the domain. The kernel used is either the cubic spline or Wendland's.

For then discretizing of the Navier-Stokes equations, a viscous term is added to the previous system. We consider the classically-used methods of Monaghan or Morris. For the Monaghan formulation, the momentum equation becomes:

$$\frac{dv_i}{dt} = -\sum_{j\in P(\Omega)} m_j \left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2} + \Pi_{ij}\right)\overline{\nabla}_i W_{ij} \quad (3)$$

with:

$$\Pi_{ij} = -C\frac{\mu}{\rho_i \rho_j}\frac{\overline{v_{ij}}.\overline{x_{ij}}}{\left\|\overline{x_{ij}}\right\|^2} \quad (4)$$

with μ the dynamic viscosity of the fluid, $\overline{v_{ij}} = \overline{v_i} - \overline{v_j}$, and $\overline{x_{ij}} = \overline{x_i} - \overline{x_j}$. Constant C is equal to 8 in two dimensions and 10 in three dimensions.

For the formulation of Morris, the momentum equation becomes:

SPH concepts for continuous wall and pressure boundaries

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

***Abstract*—The most commonly used SPH concept for modelling walls is boundary particles. The concept is used to prevent wall penetration and impose no-slip conditions. However, it is shown that it suffers from some shortcomings, in particular for the modeling of inclined and curved walls.**

Attention is paid to the wall boundary conditions, which must be imposed on the normal and shear stresses, to allow for an exact modelling of walls. It is also shown that the conditions needed for wall boundaries conflict with those needed for density estimates.

Based on the wall boundary conditions, a novel wall concept is presented, introduced as the continuous wall. In this concept wall repulsion is modelled as a reaction force to fluid compression. Wall shear is based on the wall-function concept, as used within conventional CFD, which allows the modelling of hydraulically smooth and rough walls. The continuous wall concept makes no use of boundary particles, which makes simulations less expensive. It allows for inclined and curved walls.

Further a novel concept for pressure boundaries is introduced. Although a pressure boundary is relatively simple to model within CFD, it is not straightforward in SPH. To the authors' knowledge a pressure boundary does not exist within SPH.

The concepts for the continuous wall and pressure boundary are explored in some case studies.

I. Introduction

In this paper some results are presented from a feasibility study into the use of SPH for some typical aero engine applications. The challenge is to model complex geometries, associated with two-phase flows in gear boxes and bearing chambers. Of particular interest is the modelling of thin films. For such applications the modelling of smooth and rough, inclined and curved, internal and external walls is essential. From these requirements the idea was born to model walls without boundary particles and to develop a pressure boundary.

Walls are commonly modelled in SPH using boundary particles [e.g. 1, 2]. A distinction can be made between frozen ghost particles behind the wall [1], frozen wall particles on the wall [3, 4, 5], and non-frozen mirror particles behind the wall, as introduced in [6] and used in [6, 7, 8]. In these concepts the wall is modelled by inter-particle forces, which should prevent wall penetration (via pressure forces) and impose no-slip wall conditions (via viscous forces). However, the boundary particle concept suffers from some shortcomings:

- The pressure and shear forces exerted by ghost and wall particles on a fluid particle are not exactly normal and parallel to the wall.
- With inter-particle forces the location of the (soft) wall is not precisely defined.
- The no-penetration and no-slip conditions are not fully guaranteed, since the pressure and viscous forces are smoothed, due to the use of kernels.
- Wall roughness on a scale smaller than inter-particle spacing cannot be taken into account.
- The contours of curved walls are not well described by ghost and mirror particles. Mirroring particles along a curved wall affects the density field (Fig. 1).
- The stepwise structure of a regular lattice may result in crystallization (i.e. freezing of fluid particles between ghost particles), as illustrated in Fig. 2.

For the reasons mentioned above, a new wall boundary concept is introduced in section IV, after describing the governing equations in section II and wall boundary conditions in section III. A treatment of the pressure boundary concept is given in section V. In the two case studies in the sections VI and VII, the new concepts are illustrated and explored.

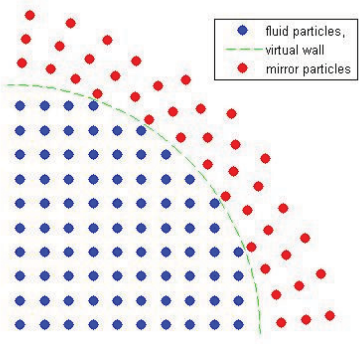

Figure 1. Mirroring particles along a curved wall affects the density field.

A unified view of SPH wall boundary conditions and particle motion correction methods

Paul H.L. Goenenboom
ESI Group Netherlands
Delft, Netherlands
Email: pgr@esi-group.com

Abstract—**In this contribution a unified representation of various approaches to SPH wall boundary conditions will be given and relations with approaches to maintain a more regular particle distribution will be made. A frequently used method to model a wall boundary for SPH is by definition of special boundary particles which provide repulsive forces similar to the Lennard-Jones force or according to the smoothing kernel [1]. Such an approach has also been extended to model fluid-structure interaction by defining finite element nodes as 'boundary' particles. A comparison between these approaches mutually and with the penalty contact method used in PAM-CRASH [2] will be made and some consequences will be discussed. The resemblance of one of the boundary particle approaches with the tensile instability correction will also be discussed.**

To improve the particle distribution for SPH fluid flow simulations, the XSPH method is well-known. More recently an alternative approach, for which the correction is based on the mutual distance between pairs of particles, rather than the difference in velocity, has been suggested [3]. The relation of this approach to the boundary algorithms will be discussed. For a test case of two boxes floating in water the particle distribution remains quite uniform using this new particle motion correction. It should be noted that the mutual distance of all particles along the boundaries remains highly regular, whereas with the original XSPH coalescence of particles was difficult to avoid. This approach is proposed as an alternative to XSPH.

I. INTRODUTION

There exist various methods to model wall boundary conditions required to contain SPH particles. Some of the methods allow for fluid-structure interaction in which the fluid is usually modeled by SPH, whereas others are only available for fixed boundaries. It is, however, not clear what the relation is between these methods, what value should be assigned to various strength parameters and which method would be best for which type of simulation study. In the following section a comparison is made between selected SPH boundary algorithms with the penalty contact used in PAM-CRASH for contact with SPH particles from which a few relations and parameter estimates are made. It will be argued that for cases in which the particles are not much larger than the elements, the penalty algorithm should provide the most accurate direction in which the contact force acts.

One of the well-known drawbacks of the standard SPH method is that the distribution of particles in space may become irregular over time which may lead to particle clustering. This problem may be counteracted by the XSPH method. Nevertheless, particle clustering may still occur i.e. along 'wetted' surfaces. In some cases this was found to influence the results. Hence, it is desirable to develop a correction in SPH that is better in preventing particles from getting too close to each other. In section 3 we have modified and extended a method based on the repulsive force of the Lennard-Jones (L-J) type [3] to overcome particle clustering. A relation will be defined between these repulsive effects with the penalty forces from the PAM-CRASH contact definitions that will enable to relate the parameters of both algorithms. An example will be discussed to demonstrate that this option is superior to the XPSH for maintaining a regular particle distribution in a fluid.

II. SPH WALL BOUNDARY CONDITIONS

A. Overview

In this section we will compare various approaches to model SPH wall boundary conditions by means of boundary particles located at the surface, with methods modeling contact between a finite element (FE) mesh with the (interior) fluid particles. For the first type of contact no direct account is taken of the surface normal direction. The interaction is based on particle-to-particle forces acting in the direction of the segment connecting the particle centers. In case there are sufficient boundary particles contributing to the interaction with a selected fluid particle and the magnitude decreases sufficiently fast with the distance, the direction in which the interaction acts may be expected to converge to the surface normal.

B. Penalty Algorithm of PAM-CRASH

For the SPH/FE coupling in PAM-CRASH the well-established penalty contact algorithm is used. This algorithm determines the nearest (triangular) surface segment for a given slave node representing a fluid particle. Any particle approaching the surface (slave) within a user-defined contact thickness will be repelled by a force in the direction of the surface normal. The same force in opposite direction is distributed to the nodes of the element with which the slave particle interacts. The strength of this penalty force is chosen such that the relative velocity in the normal direction reduces to zero within a limited number of

Generalized Ghost Particle method for handling reflecting boundaries

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

***Abstract*—This work represents an attempt to develop a generalized ghost particle method which can handle reflecting boundaries of arbitrary shapes, both convex and concave. The method generalizes the idea of ghost particles by splitting a connected boundary into corners and lines (corners, edges, and planes in 3D) where each corner is connected to 2 lines. Ghost particles are generated either as simple mirror images across a line or as generalized mirror images across a corner. In the latter case, both the number of ghost particles generated from a given interior particle and the masses assigned to each of these ghost particles can vary. The generalized ghost particle method will be described in some detail. Test results on relevant weakly-compressible applications will be presented, and where this is possible, we compare with results from the literature made with other boundary techniques. The paper concentrates on 2D applications but a discussion on the possible extension to 3D will be given.**

I. Introduction

Achieving accurate descriptions of reflecting boundaries has long been an important topic in the development of improved SPH formulations. Numerous techniques have been proposed as solutions to the problem, ranging from classic ghost particle techniques [8], normalizing conditions [6], and boundary force techniques [10]. Many of the proposed methods work well in a range of situations, but all seem to have drawbacks when it comes to simplicity, accuracy, or flexibility. A fully satisfactory approach to handling reflecting boundaries in SPH is therefore yet to be found.

In this work, we revisit the first of the above mentioned approaches, the ghost particle method. Ghost particles are generated by making mirror images of the interior particles across the boundary. It is easily implemented, robust and accurate when the boundary in question is a plane. It has also been generalized to problems where right-angle boundaries are present [1], [3]. However, it is not clear how the method can be generalized to arbitrarily-shaped boundaries in 2 and 3 dimensions. The general opinion is therefore that the ghost particle technique can only handle the simplest of geometries. This work represents an attempt to develop a generalized ghost particle methodology which can handle boundaries of arbitrary shapes, both convex and concave. It is to be considered work in progress and the methods described have yet to be generalized to three dimensions. To the authors knowledge, however, this is the first publication to formulate a comprehensive generalization of the ghost particle method. The long-term aim is to arrive at a ghost-particle method which can handle reflecting boundaries of arbitrary shapes in three dimensions with a reasonable accuracy.

This paper is organized as follows: Section II outlines the overall principles behind the current generalization. These principles mark the starting point for any algorithm, whether it is in 2D or in 3D, which constitutes a part of the generalized ghost particle method. Section III describes in some detail how ghost particles are generated when faced with reflecting boundaries which form: (a) A Convex corner. (b) A concave corner. (c) A curved boundary. It will be shown how the generalized ghost particle method, hereafter referred to as the **GGP** method, reduces to the standard ghost particle method in the trivial cases of a plane and a convex, right-angle boundary. This section also discusses how the ghost particle method can be expanded to also include a short-range boundary force. In section IV we report on results obtained with the GGP method on 4 different tests involving reflecting boundaries. A brief description is given on the approach chosen for arriving at relaxed particle distributions. The tests themselves are two-dimensional and describe a weakly-compressible fluid placed inside containers of varying shapes. Finally, section V discusses to what extent the two-dimensional methods in section III can be generalized to three dimensions.

II. Main principles of the generalized ghost particle (GGP) method

In the trivial case of a plane boundary, the ghost particle method represents a very simple approach to achieving reflecting boundary conditions. Ghost particles are generated as mirror images of interior particles found within interaction range of the boundary. An interior particle can only result in maximum one ghost particle, and the mass of the ghost particle should equal that of its parent particle. (If external forces such as gravity is present, then, the ghost particle attributes must be adjusted to fit with the local equilibrium. Since the effect of any external forces can be added as a separate step, we will in the following assume that no external forces are present.) The generalization of this concept to handle corners as well as curved boundaries is based on 3 main principles: (1) In the continuous limit, the mass density outside a boundary should be a transformed image of the mass density in the interior. (2) Although different algorithms will be used for different types

3D SPH Simulation of non-Cohesive Sediment Flushing

Sauro Manenti, Stefano Sibilla, Mario Gallati
Dept. of Hydraulic and environmental Engineering
University of Pavia
Pavia, Italy
sauro.manenti@unipv.it

Giordano Agate, Roberto Guandalini
Environment and Sustainable Development Dept.
RSE s.p.a.
Milan, Italy
giordano.agate@rse-web.it

***Abstract*— This paper shows an application of the improved SPHERA code for the numerical simulation of non-cohesive sediment flushing induced by a rapidly varied water flow discharged from the bottom outlet of a tank. After showing the improvements in tracking fluid-sediment moving interface and in computing the bottom shear stress, some numerical results are shown concerning the simulation of a laboratory flushing test carried out with both 2D and 3D geometries. The main conclusions are finally given.**

I. INTRODUCTION

One of the main concerns in the management of artificial reservoirs is the siltation process which is caused by the sediment load of the inflowing river; beside contributing to wear out the turbines and decreasing their efficiency, siltation may cause a significant reduction of the initial storage capacity within a relative short time compared with the reservoir's life span.

An economical and effective method for siltation control, referred to as flushing [1], consists of a periodical opening of the bottom outlet carried out for safety purposes: this causes the removal of part of the nearest bed sediments through the bottom shear stress exerted by the rapid water outflow from the dam heel.

There are several factors influencing the effectiveness of the above mentioned method [2] (e.g. basin morphology, sediment characteristics, river hydraulic regime, features of the bottom outlet etc.) therefore, reliable quantitative assessment of the volume of flushed sediment and of its distribution on a specific artificial basin usually requires the adoption of sophisticated and expensive physical models.

The attempt to simulate the sediment flushing through numerical models poses some critical issues connected with the adopted approach: for example the assumption of rigid-lid for water surface in three-dimensional finite volume models [3] or the absence of 3D effects (e.g. secondary currents) in a depth-averaged two-dimensional approach [4].

The SPH method allows overcoming some of the above difficulties and seems to be promising for simulating rapid sediment scouring [5], even in complex 3D problems [6].

The aim of this work is to set up an SPH-based 3D numerical model supporting the strategies for siltation control in the management of the artificial reservoirs through proper simulation of the relevant physical aspects influencing water-sediment coupled dynamics.

In the following are illustrated first the features of an enhanced version of the Shields erosion criterion described in [5] for evaluating the trigger of sediment motion; then a technique for detecting the local water-sediment interface is shown; finally some numerical results are provided for testing the effectiveness of the numerical improvements in 2D and 3D flushing problems.

II. IMPROVEMENTS OF THE NUMERICAL MODEL

A. *Shields Failure Criterion*

The typical situation which characterizes the status of the bed sediment in a flushing problem induced by a rapid water flow is schematized in Fig. 1.

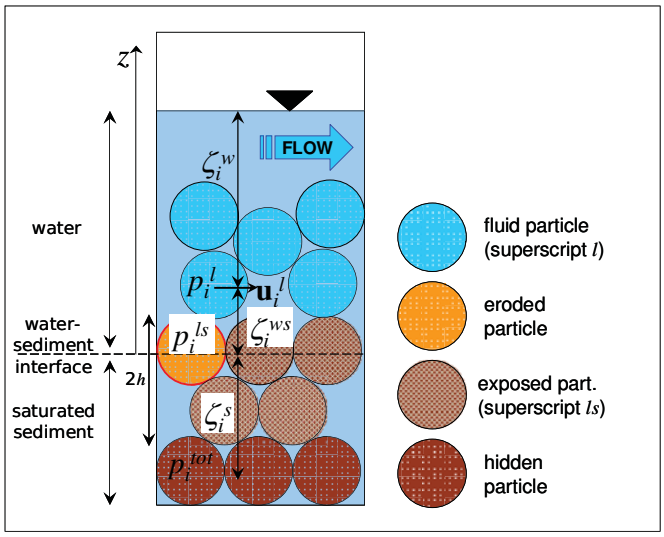

Figure 1. General treatment of bottom solid particles.

SPH simulations of buoyant and non-buoyant jets into open-channel flows

I. Federico[1], S. Marrone[2,3], F. Aristodemo[1], A. Colagrossi[2], P. Veltri[1]
[1]Dipartimento di Difesa del Suolo, Università della Calabria, Italy (federico@dds.unical.it)
[2] CNR-INSEAN, The Italian Ship Model Basin, Italy
[3] Dipartimento di Meccanica e Aeronautica, Università di Roma "Sapienza", Italy

***Abstract*—Flow phenomena induced by jet intake in a water body occur in natural environments such as pollutant discharge in rivers and engineering applications such as marine waster water outfalls. A study of buoyant and non-buoyant jets propagating into open-channel flows has been performed through a 2D SPH modelling.**

Two-phase flows in open channels are treated through an appropriate algorithm to model inlet/outlet boundary conditions. SPH equations of fluid mechanics are coupled with a SPH form of advective diffusion equation to treat different ranges of pollutant-water density ratios. Attention is paid to determine the induced flow and concentration field due to the interaction between the jet and the ambient flow.

SPH simulations of coflow and crossflow jets in buoyant and non-buoyant conditions have been carried. The model has been validated near the jet nozzle and far from it, comparing the numerical jet trajectories, velocities and concentration fields with the analytical ones.

I. Introduction

Jets have been investigated intensively for many years by the fluid mechanics community (see *e.g.* [1], [2]). The interest comes from the importance of these phenomena in several environmental and industrial flows. Jet discharges from industrial and domestic sources often enter rivers and marine areas. This flow configuration is of theoretical significance in environmental hydraulics and fluid mechanics due to the complex interaction between a jet and an ambient flow. This interaction leads to large-scale vortical structures which play a fundamental role in the entrainment of the ambient fluid into the discharge jet. In addition the transport process gives rise to mixing and dilution processes of the jet [3]. The present analyses are addressed to investigate buoyant and non-buoyant jets in shallow water streams.

SPH modelling of continuous jet discharges in open-channel flows needs appropriate inlet/outlet boundary conditions. As is common knowledge, the enforcement of these conditions is not trivial for Lagrangian particle models. Some researches have developed SPH models to treat upstream/downstream conditions in order to simulate uniform flows [4], [5]. Federico et al. [6] have proposed a suitable SPH-based algorithm to model these boundary conditions in handling different flow regimes in water streams. In this work, initial velocities, pressures and water depths both upstream and downstream in the computational domain are defined. Here, the basis of the computational method given by Federico et al. [6] is extended to model continuous two-phase fluids through the intake of tracer with the same or a different density as the surrounding open-channel flow. In order to determine the main flow phenomena induced by jet-water interaction, appropriate inflow-jet particles are introduced at different locations to simulate crossflow and coflow jets. SPH equations of fluid mechanics [7] are coupled with an advective diffusion SPH model [8] to simulate the flow field and the consequent mass transport for different range of jet-water density ratios. SPH simulations are performed in near field and far one.

In the following section the adopted SPH governing equations are recalled. Afterwards the algorithm to model jets in a finite open-channel flow through appropriate boundary conditions is illustrated. Comparisons of the proposed SPH model with analytical solutions are reported, showing the evolution of jet trajectories, velocities and concentrations given by the two-phase fluid interaction.

II. Numerical scheme

A. Governing equations

The reference equations for the flow evolution assuming a weakly-compressible fluid are:

$$\begin{cases} \dfrac{D\mathbf{v}}{Dt} = -\nabla p + \rho\,\mathbf{f} + \nabla\cdot\mathbf{V} + \mathbf{F}_s \\ \dfrac{D\rho}{Dt} = -\rho\nabla\cdot\mathbf{v} \\ p = c_0^2\,(\rho-\rho_0) \end{cases} \tag{1}$$

where $\mathbf{v}$, p and ρ are, respectively, the velocity, pressure and density of a generic material point, $\mathbf{f}$ represents the mass force acting on the fluid, ρ_0 the initial density at the free surface, c_0 the initial sound speed, $\mathbf{V}$ the viscous stress tensor and $\mathbf{F}_s$ the surface tension forces. The continuum equations proposed by Grenier et al [7] are here used to model two-phase flows. The SPH scheme is:

Effect of wall boundary treatment in SPH for modelling turbulent flows with inlet/outlet

Omar MAHMOOD, Christophe KASSIOTIS, Damien VIOLEAU
Laboratoire d'Hydraulique Saint-Venant
Joint research unit EDF / CETMEF / ENPC
Chatou, France
damien.violeau@edf.fr

Martin FERRAND, Christophe DENIS
Electricité de France R&D
MFEE and LNHE departements
Chatou, France

***Abstract*—The effect of bed friction and turbulent wall boundary conditions on the behaviour of a free-surface turbulent flow can be important in many cases. We investigate here a steady hydraulic jump and a steady flow over a Creager weir in 2-D, after developing inlet/outlet conditions. A comparison is made between two formulations of SPH for wall boundary conditions: the traditional method based on fictitious (or ghost) particles and the new approach developed in [1]. It is proved that the later provides much better results in both cases, due to a better prediction of bed friction. The discharge coefficient of the Creager weir is plotted against the upstream head, with excellent results.**

I. Introduction

The computation of flows over river waterworks is important in the framework of river basin management. It addresses the questions of people safety, energy production and many others. For this purpose, existing commercial codes based on the Volume of Fluid technique are relatively efficient, but suffer from problems when considering complex flows with moving free-surfaces, for example.

SPH is obviously one of the most relevant alternatives for such numerical studies, but still the question of wall boundary treatment remain. In the case of a flow over a weir, for example, the motion of the thin turbulent water layer running along the weir needs to be accurately predicted to calculate the distribution of pressure, which finally determines the force experienced by the structure.

We compare here two different SPH approaches for wall treatment in this context: the first one is the traditional SPH method with fictitious particles behind walls; the second one is based on our work [1], considering dynamic renormalization and wall segments, thus allowing the prescription of consistent wall boundary conditions. Then, we present inlet/outlet steady boundary conditions, developed in both models for our purposes.

Two cases are first tested in 2-D and steady regime: a hydraulic jump and a flow over a Creager-type weir, in order to compare the two approaches. Finally, a first 3-D attempt is proposed.

II. Numerical model

A. Continuous equations

We consider a turbulent weakly compressible free-surface flow. The velocity vector, pressure, turbulent kinetic energy and energy dissipation rate are denoted by $\mathbf{u}$, p, k and ϵ, respectively. Velocities and pressures are Reynolds-averaged, and the effects of turbulent fluctuations are modelled through the concept of eddy viscosity (see [2]), estimated from the $k-\epsilon$ model [3].

The Lagrangian forms of the Reynolds-averaged Navier-Stokes (RANS) and $k-\epsilon$ equations read

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

where $\mathbf{g}$ is the gravity acceleration and ρ the fluid density. The modified pressure $\tilde{p}$ and production of turbulent energy P are given by

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

with c_0 the speed of sound, $\xi = 7$, and $\mathbf{S}$ the rate-of-strain tensor field. Lastly, the dynamic viscosities are given by

Developing massively parallel SPH simulations on multi-GPU clusters

Daniel Valdez-Balderas[a,*], José M. Domínguez[b], A.J.C. Crespo[b] and Benedict D. Rogers[a]
[a]School of Mech., Aero. & Civil Engng., University of Manchester, Manchester, M13 9PL, UK
[b]Environmental Physics Laboratory, Universidad de Vigo, Ourense, Spain
*Daniel.ValdezBalderas@manchester.ac.uk

***Abstract*—We report progress towards the development of a three-dimensional, massively parallel SPH scheme using multiple Graphics Processing Units (GPUs). Starting from the single-GPU DualSPHysics code, which has proven to be a powerful, stable and accurate SPH software, we develop a multi-GPU program based on a volume domain decomposition technique, whereby different portions of a physical system are assigned to different GPUs. Inter-GPU communications and data preparation are achieved with the use of MPI directives and the efficient sorting algorithm radix sort. With the resulting software we can carry out even faster simulations than could also be done on a single GPU, but we can also perform accelerated simulations of large systems that, because of their size, would be impossible to carry out on a single GPU. We present a study of both weak and strong scaling behaviour, speedups and efficiency of our program as the number of GPUs and the size of the system under study are varied, and attempt to elucidate the computational bottlenecks. Last, we explore various possibilities for reduction of the effects of overhead on computational efficiency in future versions of our program.**

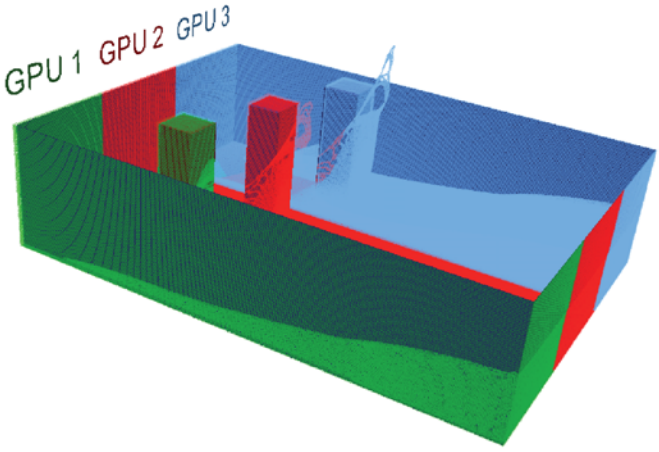

Fig. 1. Snapshot of a multi-GPU SPH simulation of a dam break with three obstacles. A system of 7 million particles was simulated for a total of 1.5 seconds of physical time in two hours, when running on three GPUs, each residing on a different host.

I. Introduction

The applicability of particle-based simulations is typically limited by two different but related computational constraints: simulation time and system size. That is, to obtain physically meaningful information from a simulation, one must be able to simulate a large-enough system for long-enough times. In the particular case of the Smoothed Particle Hydrodynamics (SPH) method, certain types of applications, for example the study of coastal processes and flooding hydrodynamics, have been limited until now by the maximum number of particles in order to perform simulations within reasonable times.

To overcome these limitations, various types of computational parallelization efforts have been made, which can be grouped in two main categories based on the type of hardware used: On the one hand there are the traditional High Performance Computing (HPC) techniques which involve the use of hundreds or thousands of computing nodes, each hosting one or more Central Processing Units (CPUs). Those nodes are interconnected via a computer networking technology (e.g., Ethernet, Infiniband, etc.), and programmed with the help of protocols like the Message Passing Interface (MPI). Examples of this type of approach include, for example, the work of Maruzewski *et al.* [1], who carried out SPH simulations with up to 124 million particles on as many as 1024 cores on the IBM Blue Gene/L supercomputer. Another recent example in this field is that of Ferrari *et al.* [2], who reported calculations using up to 2 million particles on a few hundred CPUs. The drawback of this type of approach comes from the fact that, for SPH, an enormous number of cores is needed, which require a considerable investment derived from the purchase, maintenance, and power supply requirements of this type of equipment.

The second category of parallelization approaches for the acceleration of SPH simulations involves the use of a type of hardware different from the CPUs: the Graphics Processing Unit (GPU). The development of GPU technology is driven by the computer games industry but has recently been exploited for non-graphical calculations as well. GPU programming is a parallel approach because even a single GPU contain hundreds of computing cores, and multiple threads of execution are launched simultaneously. The use of GPUs for scientific computations has come to represent an exciting alternative for the acceleration of scientific computing software. The release of the Compute Unified Device Architecture (CUDA) and its development kit (SDK) by Nvidia in 2007 has facilitated the popularization of the use of these devices for general purposes, but efforts in this direction existed even prior to

DualSPHysics, new GPU computing on SPH models

A.J.C. Crespo, J.M. Domínguez, A. Barreiro and M. Gómez-Gesteira
EPHYSLAB Environmental Physics Laboratory
Universidad de Vigo
Ourense, Spain
alexbexe@uvigo.es, jmdominguez@uvigo.es,
anxo.barreiro@uvigo.es, mggesteira@uvigo.es

B.D. Rogers
School of Mechanical, Aerospace and Civil Engineering
University of Manchester,
Manchester, U.K.
benedict.rogers@manchester.ac.uk

***Abstract*—The capacity of a single processor is insufficient to run huge 3D simulations with Smoothed Particle Hydrodynamics (SPH) methods. Here we present the development of a new code named DualSPHysics that can be executed either on either Central Processing Units (CPU) or on Graphics Processor Units (GPUs). Thus, the parallelisation approach using the Compute Unified Device Architecture (CUDA) of nVidia is described. Simulations with more than one million particles on a single GPU card exhibit speedups of up to two orders of magnitude over using a single-core CPU. The DualSPHysics code is validated with the experimental data corresponding to a dam break flow impacting on an obstacle and the agreement between the numerical and experimental results is analysed. This new technology makes possible the study of real-life engineering problems at a reasonable computational cost such as fluid-structure interaction.**

I. INTRODUCTION

Despite being able to model complex nonlinear flows, Smoothed Particle Hydrodynamics (SPH) possesses numerous drawbacks. One of its main problems is the expensive computational cost in comparison with other mesh-based methods for CFD problems. Using the weakly compressible approach for SPH, the time steps involved in the simulation are too small giving rise to large computational times. On the other hand, the number of neighbours per particle is large which implies a correspondingly large number of interactions per particle at each time step. Thus, the computation of real cases on a single central processing unit (CPU) is simply prohibitive since the computational cost is unaffordable when the resolution is increased and the physical information is too coarse when the resolution decreases. The further development of SPH and its application by industry to real problems has therefore been limited. In order to make SPH simulations computationally feasible and useful for industrial purposes, the need to perform simulations with millions of particles is imperative. Therefore, hardware acceleration is essential.

Nowadays, two main techniques can be employed to accelerate SPH simulations. The first one is using supercomputers with thousands of CPU cores (such as the IBM supercomputer Blue Gene/L at Ecole Polytechnique Fédérale in Switzerland) and the second option is using the novel architectures of the Graphics Processing Units (GPUs).

This new technology is imported from the computer games industry. Thus, due to the growth and progress in the video game market and multimedia, the performance of GPUs has increased much faster than that of CPUs during the last decade. GPUs are designed to treat large data flows and to render pixels at several tens of frames per second. From a computational point of view they are highly efficient thanks to their multi-threading capability. For example, with a GTX480 card a maximum of 23,040 threads could be executed simultaneously (15 multiprocessors and 1,536 threads per multiprocessor as a maximum). GPUs are thus an accessible and cheap option to accelerate SPH models. The graphics cards can be used as the execution devices taking advantage of their parallel programming power. GPUs are not only cheaper but also easier to maintain than large cluster multi-core machines.

The first attempt to perform an entire implementation of an SPH scheme on a GPU was developed by Harada *et al.* [1]. Previously, schemes only implemented selected parts of the code, but in this work the entire SPH computation was executed on the GPU using a simple and classical SPH formulation. In [1], SPH was accelerated with initially satisfactory results obtaining speedups of over 28 times using a GPU (GeForce 8800GTX) compared to a CPU with tests of 260,000 particles. The proposed method was implemented before the appearance of the Compute Unified Device Architecture (CUDA), which is both a programming environment and a language for parallel computing specifically for nVidia GPUs. Thus, Harada's work was a significant advance even when most of its limitations could now be fixed using the advanced GPU programming features introduced by CUDA. For SPH, CUDA-enabled GPU technology was first introduced during the SPHERIC workshop in Nantes (2009) by [2]. The same authors published a work [3] describing the GPU model and applying SPH to study free-surface flows where they

GPU implementation of a SPH-ALE fluid dynamics solver

Jean-Christophe Marongiu, Julien Leduc
Hydraulic R&D department
ANDRITZ Hydro
Vevey, Switzerland
jean-christophe.marongiu/julien.leduc@andritz.com

Matthieu Schaller
Ecole Polytechnique Federale de Lausanne
Lausanne, Switzerland
Matthieu.schaller@epfl.ch

Abstract— **Application of SPH methods in an industrial context is closely linked to the capacity of exploiting the very last and best computational resources available. Indeed SPH involves a computational load that can be significant compared to others and well established numerical methods. Two years ago appeared in the SPHERIC community the first full GPU implementation of a SPH flow solver [1], based on the high level API CUDA from Nvidia. This pioneering work revealed the high potential of the GPU technology and further works [2] confirmed that hybrid implementations can challenge the classical MPI- or OpenMP-based approaches. However the SPH-ALE method [3] exhibits important peculiarities that require a dedicated approach. The paper thus reports the implementation strategy that has been applied for the different parts of the algorithm, leading to the full GPU implementation of a second order SPH-ALE flow solver, including the management of all types of boundary conditions (solid walls, periodic, symmetry). The impact of numerical models on performance is analyzed. The respective assigned roles to CPU and GPU are also discussed, together with the choice of floating point accuracy. A detailed validation of the GPU implementation versus its CPU counterpart is also shown. The resulting flow solver can achieve speed-ups up to 40 on industrial applications.**

I. INTRODUTION

Among the challenges that can prevent dissemination of the SPH method in industrial processes, the question of its competitiveness versus well established numerical methods has to be addressed. Considering a whole simulation process, from creation of a 3D simulation domain to the post-processing, SPH presents the main advantage of not requiring a 3D computational mesh. The preparation of the simulation in itself is thus greatly simplified and accelerated. The associated costs can thus be reduced, but one should also keep in mind that industrial processes often involve automatic meshing that hides the complexity of this phase in the daily work. The computational efficiency of the method consequently remains at the heart of its global competitiveness.

The ability of SPH to model free surface flows without modeling the gaseous phase is undoubtly a major advantage in most of hydrodynamic applications, as it can lead to a drastic reduction in the number of calculation points compared to mesh-based techniques. However some basic features of the SPH method are also disadvantaging: its meshless numerical stencil involves a much higher number of neighbouring points (typically around 100 in 3D, to be compared to around 10 for mesh-based techniques), fully explicit time integration and weakly compressible formalism lead to very small time steps, and finally the relatively low accuracy of the method requires a finer spatial discretization. Parallel computing is consequently a widely adopted implementation paradigm in order to accelerate the execution of numerical simulations. Classical approaches – multithreading with OpenMP, domain decomposition with MPI – have been investigated in the SPH community ([4], [5]). Another parallel model based on GPGPU (General Purpose Graphic Processor Units) appeared some years ago and is becoming more and more popular. In particular the development framework CUDA supported by the GPU provider NVIDIA has been used in the pioneering work of Herault [1] and later by Crespo [2] to implement SPH-based flow solvers in massively parallel approaches. These works allowed to accelerate simulations significantly (Crespo obtained a speed-up of 30), at the expense of a major coding effort. Indeed the GPGPU approach requires a complete rethinking of the implementation choices, from the data structures to the succession of operations.

The present work aims at implementing a SPH-ALE (SPH-Arbitrary Lagrange Euler) flow solver on GPU architecture using the CUDA framework. Even if SPH-ALE is a close parent of the standard SPH method, its increased complexity has to be carefully taken into account along the implementation and optimization process. Indeed some of the conclusions drawn from the work of Crespo could not be extended to SPH-ALE and different options had to be chosen. An analysis of the SPH-ALE algorithm will first highlight some specificity, and then the step-by-step implementation of the main parts of the algorithm will be reported. Implementation choices will be discussed and validated. Applications of the resulting GPU flow solver on free surface flows will then be presented and will demonstrate the great benefit for industrial applications.

II. ANALYSIS OF SPH-ALE ALGORITHM

In this article only a brief overview of the SPH-ALE method is reported, a thorough description can be found in [3].

Simulating rheology and microrheology with Smoothed Dissipative Particle Dynamics

Adolfo Vázquez-Quesada
Lehrstuhl für Aerodynamik
und Strömungsmechanik.
Technische Universität München.
Munich, Germany.
adolfo.vazquez@aer.mw.tum.de

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

Pep Español
Departamento de Física Fundamental.
Universidad Nacional de Educación a Distancia.
Madrid, Spain.
pep@fisfun.uned.es

***Abstract*—The Smoothed Dissipative Particle Dynamics model for a viscoelastic fluid is used to simulate a Small Amplitude Oscillatory Shear flow. The same model is used to simulate a colloidal suspension in a viscoelastic solvent. Microrheology measurements are performed where the thermal movement of the embedded colloidal particles allows one to obtain the rheological properties of the solvent. The rheological properties obtained from these simulations are successfully compared with analytical solutions.**

I. Introduction

Rheology studies the mechanical response and flow properties of complex fluids/materials. Traditionally, one uses rheometers in which a macroscopic amount of liquid is confined between moving walls and the deformation and stress response are measured. Recently, a new technique called microrheology has been devised that shows several advantages with respect to traditional rheology: it allows to use minuscule quantities of fluids, the measurements are local, and a much broader frequency range can be investigated. The main ingredient in this technique is the probe: a colloidal particle, embedded in the fluid which acts like a microscopic rheometer stressing and deforming the surrounding medium. From the diffusion motion of the probe the viscoelastic properties of the suspending medium fluid can be inferred through a generalized Stokes-Einstein relation [1], [2].

Despite its current widespread use, there are still open questions about the general applicability of microrheology. For example, the generalized Stokes-Einstein equation is advocated as a useful empiricism but its range of validity should be fully explored []. Also, the influence of other colloidal particles or confining walls on the diffusive motion of a single test particle deserves some attention. A simulation model able to address this type of problems should be very useful for the fundamental understanding of microrheology.

A computational technique suitable for this task needs several ingredients like modelling viscoelastic fluids, including thermal fluctuations, and interacting with embedded rigid inclusions. The model used here [3] is a variation of the Smoothed Dissipative Particle Dynamics (SDPD) model [4], where a conformation tensor is used to reproduce the elastic behaviour of the fluid. As in the original SDPD model, the viscoelastic model resorts to the GENERIC formalism [5] to obtain thermodynamic consistency. The resulting equations conserve linear momentum and energy and fulfill the Second Law of Thermodynamics. The discrete equations thus obtained can be understood as a discretization of the equations of the Oldroyd-B model with thermal noise consistently incorporated.

The walls and the rigid colloidal particles moving through the viscoelastic matrix are modeled using frozen boundary SDPD particles filling the solid domain [6], [7]. In order to assess the accuracy of the model in describing complex viscoelastic fluids under standard rheometric conditions, a simple small amplitude oscillatory flow (SAOS) is simulated and the results are compared with theoretical predictions. Furthermore, rheological properties are evaluated under simulated *microrheological conditions* and the results discussed.

II. The model

We present in this section the model for a colloidal suspension in a viscoelastic solvent. A brief explanation of the SDPD viscoelastic model is done in the subsection II-A. For further details we refer to [3]. The way how the solid inclusions are introduced in the solvent is described in subsection II-B.

A. Viscoelastic solvent

The SDPD model for a viscoelastic fluids [3] is obtained within the GENERIC framework [5], [8]. The dynamic equations conserve linear momentum and energy and fulfill the Second Law of Thermodynamics. Moreover, thermal fluctuations are introduced in a natural way through the Fluctuation-Dissipation theorem and scale automatically with the resolution of the simulation [9]. In this way, the SDPD model for viscoelastic fluids is a Smoothed Particle Hydrodynamics (SPH) model for viscoelastic fluids with thermal fluctuations.

The fluid is represented by a set of N_F SPH particles of mass m. For the Newtonian case [4], the state of the system is given by the position $\mathbf{r}_i$, velocity $\mathbf{v}_i$ and internal energy E_i of each fluid particle i. The elastic behavior of the viscoelastic fluid is obtained by supposing that every fluid particle i has N^p

SPH simulations of a viscoelastic flow around a periodic array of cylinders confined in a channel

Marco Ellero and Adolfo Vázquez Quesada

Institute of Aerodynamics and Fluid Mechanics
Technische Universität München
Munich, Germany
marco.ellero@aer.mw.tum.de

***Abstract*—In this talk a numerical study on the performance of SPH in the case of a flow of a viscoelastic liquid around a linear array of cylinders confined in a channel will be presented. Numerical convergence in the case of a low Reynolds number Newtonian flow was demonstrated in [9]. In this talk the viscoelastic effects are incorporated in the SPH scheme according to the Oldroyd-B model presented in [7]. Good agreement of the dimensionless drag force acting on the cylinder for a wide range of Weissenberg numbers is observed. The case of closely spaced cylinders is also investigated and the impact of the Weissenberg number** We **on the solution will be discussed. It turns out that in the Newtonian case a stable secondary flow is observed. For increasing** We **these counter-rotating vortices become eventually unstable, breaking the plane symmetry and producing a quasi-periodic flow of mass in and out of the wall region in the expansion zone. Purely elastic instability arising at zero Reynolds number have been recently observed in several flow problems and should be caused by the presence of non-zero normal stresses along curved streamlines which can drive the flow out of equilibrium generating complex structures. The elastic instability found here is in remarkable good agreement with experimental observations of dilute polymer solutions under similar flow conditions reported in [14].**

I. Introduction

The study of viscoelastic flows in complex geometries is of crucial importance for the proper modelling and accurate prediction of complex liquid behaviours. Flow of polymeric suspensions through porous media, contraction/expansion geometries are employed in a variety of engineering applications including composite manufacturing processes, paper coating and recently also in microfluidics devices [1]. Generally, the rheological behaviour of complex fluids can be very well characterized under simple viscometric conditions, that is, under a pure shear or extensional flow. However, modelling the flow in geometries where the two components are simultaneously present represents a big challenge both, from the mathematical and numerical point of view. Recently, several complex flows have been studied which include, for example, the flow through channels with variable cross-section, flow around cylinder arrays both unbounded and confined by plane walls. In particular, the latter test case has been investigated experimentally [12]–[14] in relation to the occurrence of purely elastic instabilities, namely flow instabilities arising in absence of inertia. The study of elastic instabilities in complex fluids has been reviewed in [2], [3] and is receiving increasing attention also due to the strict connection with the phenomenon of elastic turbulence [4]. An empirical explanation has been attempted by McKinley et al. in terms of the possible destabilizing mechanism linking curved flow streamlines to the presence of normal elastic stresses [16], however a detailed numerical analysis is still lacking. Elastic instabilities show up generally together with an alteration of the global flow behaviour: as a critical Weissenberg number is achieved, global quantities (e.g mean flow, drag coefficients, applied pressure) start to exhibit fluctuations which increase in magnitude as the effect of liquid elasticity becomes dominant. Moreover, this phenomenon is accompanied by an abrupt increase in the flow resistance which is now believed to be related to a non-linear transition from a steady (time-independent) towards a more dissipative flow structure (either three-dimensional or unsteady) [13], [14]. Although many experimental evidencies of this process can be found in the literature, few numerical computations have been able to reproduce the elastic transitions mentioned above. For example, the elastic transition of an Upper-Convected-Maxwell fluid (UCM) towards a steady secondary asymmetric flow field have been simulated in cross-channel geometries [17] and found in agreement well with previously reported experimental observations [18]. In the context of a cylinder array structure, however, no numerical simulations have been able to reproduce, at least qualitatively, the time-dependent flow behaviour and the relative abrupt increase in flow resistance observed in the experiments. As suggested by [15], the main reason might be that, if a Hopf bifurcation in the mathematical solution of the given viscoelastic model exists, standard numerical methods can follow only its steady branch which is physical unrealizable and which can correspond to a lower resistance.

In this work we consider the viscoelastic flow of an Oldroyd-B liquid around a linear array of cylinders placed at different distances and confined in a channel. This problem allows to combine a relatively simple domain geometry together with a flow characterized by a complex mixing of both shear and extensional behaviour. Furthermore, the problem has been extensively studied numerically [11] and experimentally [14].

GPU-LAVA: SPH lava flow simulation on CUDA

Alexis Hérault, Annamaria Vicari, Ciro Del Negro
Sezione di Catania
Istituto Nazionale di Geofisica e Vulcanologia
Catania, Italy

Giuseppe Bilotta, Eugenio Rustico
Dipartimento di Matematica e Informatica
Università di Catania
Catania, Italy

Alexis Hérault
Département Ingénierie Mathématique
Conservatoire des Arts et Métiers
Paris, France

***Abstract*—A Smoothed Particle Hydrodynamics (SPH) method for lava flow modeling has been implemented on a graphical processing unit (GPU) using the Compute Unified Device Architecture (CUDA) developed by Nvidia, resulting in up to two order of magnitude speed-ups. The 3-dimensional model is able to simulate a lava flow on on a real topography with free-surface, non-Newtonian fluids and with phase change. The entire SPH code, with its three main components: neighbor list construction, force computation, and integration of the equation of motion, is computed on the GPU, fully exploiting its computational power. The simulation speed achieved is one to two orders of magnitude faster than the equivalent CPU code. GPU implementation of SPH permits high resolution SPH modeling in hours and days rather than weeks and months on inexpensive and readily available hardware.**

I. Introduction

Developing physical-mathematical models that can describe the spatial and temporal evolution of natural phenomena is a key point in many scientific areas. Predicting the potentially affected areas of high-risk volcanic phenomena such as lava flows is essential to support risk mitigation and land planning, in combination with laboratory and field observations.

Physical and numerical models developed so far for the simulation of lava flow work under simplified assumptions and have provied sufficiently accurate results to be employable for civil protection. These models are however inadequate for the description of more sophisticated phenomena like crust and lava tube formation and ephemeral vent opening, which can strongly increase the hazard associated with lava flows.

A complete modelling of lava flows is challenging from a modellistic, numeric and computational point of view: the natural topography irregularities, the dynamic free boundaries and phenomena such as solidification or friction, presence of floating solid bodies or other obstacles and their eventual fragmentation, make the problem difficult to solve using traditional numerical methods such us finite volumes or finite elements

We offer a solution to lava flow modeling based on the Smoothed Particles Hydrodynamics (SPH) approach. To overcome the high computational requirements of the method, we exploit the intrisic high degree of parallelism of the SPH method and rely on low-cost, energy-effective parallel computing capabilities offered by the new generations of Graphic Processing Units (GPUs), resulting in one to two orders of magnitude in speed-up over standard CPU execution.

The work we present has been developed at the Istituto Nazionale di Geofisica e Vulcanologia (INGV), Section of Catania, within the course of LAVA Project. The three-dimensional model can describe the flow of a fluid with thermal-dependent non-Newtonian behavior, including phase transition, on a natural topography and with a free surface, allowing for accurate forecasting of the possible paths of a lava flow during an eruption.

II. SPH modeling of a lava flow

A. Equation of the problem

Lava is considered incompressible, and the liquid phase is modeled by Navier-Stokes equations (continuity equation, forces balance, incompressibility condition):

$$\frac{D\rho}{Dt} = -\rho\frac{\partial u_i}{\partial x_i} \tag{1}$$

$$\frac{Du_i}{Dt} = g_i + \frac{1}{\rho}\frac{\partial \sigma_{ij}}{\partial x_j} \tag{2}$$

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{3}$$

coupled with the heat equation:

$$\rho c_p \frac{DT}{Dt} = \frac{\partial}{\partial x_i}\left(\kappa\frac{\partial T}{\partial x_i}\right) + \frac{\partial}{\partial x_i}\left(\sigma_{ij}u_j\right) \tag{4}$$

where $\frac{D}{Dt} = \frac{\partial}{\partial t} + u_i\frac{\partial}{\partial x_i}$ is the total derivative operator, $\sigma_{ij} = -P\delta_{ij} + \tau_{ij}$ is the stress tensor and τ_{ij} the viscous stress tensor.

We consider that the Etna lava flows are laminar with a low Reynolds number (viscosity $\approx 10^4 Pa \cdot s$ and maximum velocity $\approx 1m \cdot s^{-1}$).

Rheological Model — constitutive equation: The constitutive equation is a relation between the viscous stress tensor τ_{ij} and the strain tensor $\epsilon_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$. In the case of a

Numerical investigation of wave-induced mean flows in the surf zone using the SPH method

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*— Smoothed Particle Hydrodynamics is used to model temporal and spatial variations of wave-induced nearshore circulation. An idealized rip current system consisting of a single bar and a rip channel is analysed and the interaction of rip current and waves is investigated. Numerical results include time-averaged and depth-averaged velocities as well as three-dimensional structure of flow in rip channel, over the bar and in nearshore region. Satisfactory agreement is achieved between the numerical data and laboratory measurements of Drønen et al. (2002) including mean water surface and cross-shore and long-shore velocity components.**

I. INTRODUTION

In surf zones, rip currents, often flowing seaward in gaps between sand bars, are important phenomena because of their environmental and life hazards concerns. The driving mechanism for rip currents is the momentum flux of the waves. As the waves break more on the bars and less in the rip channel, a set-up (of the mean water level) gradient occurs that drives a current towards the sea.

Haller et al. (2002) set up a series of experiments in a 17 m by 18 m wave basin. Rip current and nearshore circulation were measured over two bars and a rip channel between them [1]. Their measurements were mostly in two-dimensions, illustrating depth-integrated velocities and mean water surface elevations. The experimental results were later modelled numerically by Chen et al. (1999) [2]. They used a model based on Boussinesq equations to study nearshore circulations and compared their results with the Haller experiments. Drønen et al. (2002) performed a laboratory study of flow over a single bar and a rip channel in a 4 m wide and 30 m long wave tank. Their experiments were mostly in three-dimensional space, measuring the 3D structure of rip channel flow. Particle trajectories tests were also performed to track the particles in a rip current system [3].

This paper presents a three-dimensional numerical modelling of rip current and the associated circulations in a bar/rip channel system. The GPUSPH model [4] is extended to be capable of finding fluid parameters in fixed positions as well as from moving particles. Depth and wave-period averaging is used to determine wave-induced flows. The 3D flow pattern is studied to show the depth variation of rip current intensity. Changes of mean velocity circulations and mean water surface elevations are presented. At the end the effect of particle spacing and numerical resolution On the accuracy of results is investigated. Numerical results are compared to the experiments performed by Drønen et al. (2000).

II. GPUSPH MODEL

Implementation of the SPH method with fine resolution (large number of particles) can lead to large computational time. However, some complicated cases need a large number of particles to capture flow fields accurately. This issue leads to the use of parallel computing that has mostly been done on multiple central processing units (CPUs) of computers. Recently graphical processing units (GPUs) have been recognized as capable of parallel computing with high computational power and low expenses. GPU, because of its multi-core processors, are strongly parallel in nature and the data-parallel nature of SPH method makes it perform well on multithreaded GPU. GPU programming can be traced back to General-Purpose computation on GPU (GPGPU). This method can also use graphic programming languages such as OpenGL to provide the real-time graphic output.

Recently a new GPU parallel architecture is introduced as Compute Unified Device Architecture (CUDA), which uses high-level languages such as C, C++ and Fortran to introduce the problem to graphic cards so transfer of the problem data to GPU has become easier.

In this study, the GPUSPH package developed by Hérault et al. (2010) is implemented. It is an open-source code available from www.ce.jhu/dalrymple/GPUSPH. This package is written using CUDA programming language and is consists of two types of objects: those that are run on the CPU and the ones that are run on GPU. Data transfer from CPU to GPU and vice versa, is done in different stages of program but the main calculations are done on GPU. In

Validation of standard and Riemann-based SPH solvers for shallow water flows including bed discontinuities

Jie Zhao[1], Matthieu de Leffe[1,2], David Le Touzé[1], Lionel Gentaz[1], Pierre Ferrant[1]
[1] Fluid Mechanics Laboratory, Ecole Centrale Nantes / CNRS, Nantes, France
[2] HydrOcean, Nantes, France
jie.zhao@ec-nantes.fr

In the present paper, SPH numerical methods for modelling shallow water coastal flows are proposed and compared, with the aim of performing free-surface wave propagations over complex topographies, especially including discontinuities of the bed. A shallow-water SPH solver based on an artificial viscosity is first implemented, following previous work by De Leffe et al [1]. To obtain a numerical model which can reliably and accurately solve the shallow water equations over a step-like bottom, a Riemann-based SPH solver is then proposed, inspired by the widely-adopted Roe solver in finite volume methods. These solvers are then validated and compared on different shallow water flows. The results obtained are in good agreement with the existing data in the literature.

I. INTRODUCTION

In coastal engineering, free surface flows such as tsunamis, ocean tides, dam breaches, and river floods may inundate low-lying regions, causing significant loss of life and property. The shallow water equations provide an efficient means of accurately modeling the long wave forms produced by such flows, and therefore remain of interest to the research community. The mesh-free lagrangian nature of SPH gives the method flexibility to adaptively track the development of the wave-front. Such a property is not shared by classical grid-based methods, highlighting the advantage of SPH when modeling events with large scales, severe deformations, and dry-wet topography.

In the past, significant work in modeling the shallow water equations within an SPH framework has been completed. A variable smoothing length model was developed by Rodriguez-Paz and Bonnet [2]. Almost simultaneously, the new technique based on Riemann solver was introduced to improve the stability of the model by Ata and Soulaimani [3]. With the aim of setting up a Tsunami early warning system, Panizzo et al. [4] developed a scheme to model nonlinear shallow water equations in SPH (SPH-NLSWE). A method was then implemented to follow the expansion of the fluid domain throughout flooding simulations by De Leffe et al. [1]. Vacondio et al. [5] introduced a particle splitting procedure to overcome the poor performance at small depths caused by the widely adopted law that the smoothing length is inversely proportional to water depth. However, the SPH modeling of shallow water coastal flows has not yet been applied and tested to catching the shoreline wave information and diverse wave propagation over discontinuous bed.

In the present work, the model of SPH-NLSWE has first been extended to the application to dry-wet shoreline flows on the basis of work developed in the authors' laboratory of Ecole Centrale Nantes. Two benchmark tests for long waves have been reproduced, namely the solitary wave propagation on a canonical beach, and shoreline wave runup-rundown motions with time variation. The objective was then to develop a numerical model that can reliably solve a variety of shallow water flows over variable bathymetry in diverse conditions. To achieve this, the Godunov-type Roe solver has been introduced to SPH-NLSWE to deal with intractable flows with standard methods, involving discontinuous free surface waves and step-like beds. The obtained formulation is robust and provides reliable results, showing good agreement with reference results.

II. PHYSICAL AND STANDARD SPH MODEL

The NLSWE are derived from the depth-integration of the Navier-Stokes equations especially for the common situations in fluid dynamics where the horizontal length scale is much greater than the vertical length scale. For instance, tsunami is a typical long wave motion, even the offshore zone may be considered to be shallow as its depth is much smaller compared with tsunami wave length. For an incompressible flow, the shallow water equations take the following conservative form:

$$\frac{\partial U}{\partial t} + \nabla \cdot F = S \tag{1}$$

and

Shallow water sloshing modeled through the δ-SPH scheme

Matteo Antuono, Benjamin Bouscasse
CNR-INSEAN (the Italian Ship Model Basin)
Rome, Italy

Andrea Colagrossi, Claudio Lugni
CNR-INSEAN (the Italian Ship Model Basin), Rome, Italy
CESOS (Centre of Excellence for Ship and Ocean Structures)
NTNU, Trondheim, Norway

***Abstract*—The δ-SPH scheme proposed in Antuono et al. [1] has been applied to the analysis of sloshing phenomena (i.e. violent fluid motions inside a tank) in shallow water regimes. The study of such motions is of fundamental importance for the stability and structural safety of Liquid Natural Gas carriers. In this context, sloshing phenomena in shallow water represent a very demanding problem for both numerical schemes and theoretical models because of the major role played by nonlinearities. To assess the accuracy of the δ-SPH scheme, comparisons have been made with experimental measurements and with an analytical model. In all the considered cases, the δ-SPH scheme, the experiments and the analytical model display a good agreement. This further confirms the ability of the δ-SPH scheme in the modeling of violent free surface motions.**

I. Introduction

The sloshing phenomenon indicates the fluid motion inside a tank induced by external forces. It is particularly important for Liquid Natural Gas carriers and, in general, for ships which transport fluids. In fact, during the largest part of the ship travel, tanks used for the fluid/gas storage are only partially filled and the external sea motion can induce violent fluid motions inside them. Specifically, when the frequency spectrum of the ship motion is peaked in the region close to the lowest natural tank mode, violent free-surface flows may appear, inducing large local loads (see Faltinsen et al. [2]) and increasing the risk for the integrity of the structure.

For a complete description of the sloshing phenomenon it is, therefore, necessary a numerical scheme which can describe complex free surface flows (including fragmentation and wave breaking) and which can accurately predict the pressure loads against the tank walls. In this context, the SPH scheme has proved to be a reliable choice (see, for example, [3]–[6]) thanks to its Lagrangian structure and to the absence of any computational grid.

The aim of the present work is to add a further contribution to the modeling of sloshing phenomena through SPH schemes. Specifically, we adopt the δ-SPH scheme proposed in Antuono et al. [1] and further inspected in Antuono et al. [7] for gravity wave propagation. This scheme has proved to be very accurate in both the description of the free surface evolution and to the prediction of the local loads against structures.

In the present work, we deal with sloshing phenomena in shallow water conditions since they represent a very demanding problem for both numerical schemes and theoretical models because of the major role played by nonlinearities. To assess the accuracy of the δ-SPH scheme, comparisons have been made with experimental measurements and with an analytical model. In the former case, the experimental campaign of Lepelletier & Raichlen [8] and the experimental data available at the INSEAN on the shallow-water sway motion of a rectangular tank has been considered. For what concerns the comparison with the analytical model, a modal system based on depth-integrated equations (Boussinesq-type equations with a linear dispersive term) has been adopted.

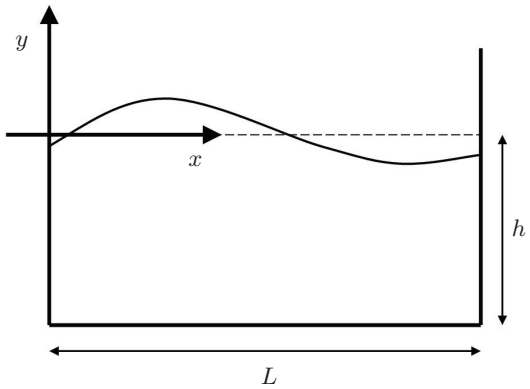

Fig. 1. Sketch of the tank and of the frame of reference.

In all the cases/experiments which follow, a rectangular tank is used and the frame of reference is set like in figure 1. Specifically, L and D indicate the tank length and breath respectively, h the filling height and $k = \pi/L$ the wave length.

II. δ-SPH scheme

The δ-SPH scheme is based on the assumption that the fluid is barotropic and weakly-compressible. Te reference equations for the flow evolution are the Euler equations along with a linear state equation. A proper artificial diffusive term is used into the continuity equation in order to remove the spurious numerical high-frequency oscillations in the pressure field and, similarly to the largest part of the weakly-compressible SPH schemes, an artificial viscous term is added inside the momentum equation for stability reasons (see for example

Author Index