The University of Manchester

Proceedings of the

5th International SPHERIC Workshop

22-25 June 2010
School of Mechanical, Aerospace and Civil Engineering
University of Manchester, U.K.

Edited by B.D. Rogers

Dr Benedict Rogers
School of Mechanical, Aerospace
and Civil Engineering
The University of Manchester
Pariser Building, Sackville Street
Manchester, M13 9PL, U.K.

www.mace.manchester.ac.uk
benedict.rogers@manchester.ac.uk

June 2010

Foreword to the 5th International SPHERIC Workshop

Dear Delegate,

The School of Mechanical, Aerospace & Civil Engineering (MACE) at the University of Manchester is delighted to host the 5th International SPHERIC workshop. This will be the premier event of 2010 in the field of Smoothed Particle Hydrodynamics (SPH) focussing on the method and applications.

The SPH European Research Interest Community (SPHERIC) was founded in 2005 as a Special Interest Group of the ERCOFTAC community and aims at encouraging and facilitating the spread of the method throughout Europe and the wider international community. Since that time, the SPHERIC community continues both to grow and to play an important role in helping the development of SPH for academia, industry and government organisations. SPH is one of the most exciting new areas in the field of computational methods and is opening up the possibility of research into fields that were beyond any modelling capability but are now being actively pursued in a whole range of fields.

As in previous years, the SPHERIC Committee received a record number of abstracts (eighty this year) demonstrating just how active the field is in fields ranging from solids and fluid-structure interaction to heterogeneous HPC. In keeping with the workshop ethos with no parallel sessions, the committee were unable to accept them all. In the papers presented, you will see the very high quality of work in SPH currently underway.

For the 5th International SPHERIC Workshop, it is with great pleasure that I welcome you all to the City of Manchester, a city whose technological advances have played and continue to play a crucial role in the development of the modern world.

Thank you for your participation.

Benedict Rogers
Chair of the Local Organising Committee of the 5th SPHERIC Workshop 2010

Committees

SPHERIC Steering Committee (Workshop Scientific Committee)

- Damien Violeau (Electricité de France, LNHE, Chatou, France)
- Dr Nathan Quinlan (National Univ. of Ireland, Galway) - Secretary
- Dr Benedict Rogers (Univ. of Manchester, UK) - Webmaster
- Prof. Moncho Gómez-Gesteira (Univ. de Vigo, Spain)
- Dr 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)
- Dr David Le Touzé (Ecole Centrale de Nantes, France)
- John Biddiscombe (CSCS, Switzerland)
- Dr Xiangyu Hu (Technical University of Munich, Germany)

Local Organising Committee – University of Manchester

- Dr Benedict D. Rogers - Chair
- Prof. Peter K. Stansby
- Pourya Omidvar – Webmaster
- Dr Daniel Valdez Balderas
- Dr Christophe Kassiotis
- Roberto Lotto
- Martin Ferrand

Acknowledgements

The Local Organising Committee would like to thank the sponsors and supporting organisations:

Lloyds Register – Sponsorship of the 1st Keynote Lecture: Dr N. Quinlan
BAE Systems – Sponsorship of the 2nd Keynote Lecture: Prof. Nathan Quinlan
nVidia – Donation of the Student Prize
SuperMicro

The Chair would like to express his sincere gratitude to the following people who really helped to make this workshop happen:

School of Mechanical, Aerospace and Civil Engineering (MACE)
Confercare: Janet Adnams, Andy Tickle
MACE: Steve Manifould
The Local Organising Committee, in particular:
- Pourya Omidvar for website construction and many good organisational suggestions
- Christophe Kassiotis and Martin Ferrand for helping me produce these proceedings

Printing:

Published by 5th International SPHERIC Workshop Local Organising Committee

CONTENTS

Improvement of multiphase model using preconditioned Riemann solvers

J. Leduc, F. Leboeuf, M. Lance
Laboratory of Fluid Mechanics and Acoustics
Ecole Centrale de Lyon, University of Lyon
Ecully, France
julien.leduc@ec-lyon.fr

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

***Abstract*—This work focuses on the improvement of a multiphase method for Smooth Particles Hydrodynamics (SPH) simulations. Taking advantages of the arbitrary Lagrange-Euler (ALE) formalism, this model uses an acoustic Riemann solver in the SPH-ALE method which allows sharp interfaces between fluids and is fully compliant with a purely lagrangian description of the interface. The particular case of low Mach number flows is addressed in this paper. Preconditioned Riemann solvers are used, in the core flows but also at the interface, to limit the numerical diffusion linked with upwind schemes.**
Surface tension effects are also included in the model. The Continuum Surface Force (CSF) model and the Local Laplace Pressure Correction model (LLPC) are studied in this paper in terms of spatial convergences and impact of preconditioned Riemann solvers. The different approaches are successfully validated on well documented academic cases: gravity waves, dam break and static/oscillating droplet.

I. Introduction

High speed water flows as jets from Pelton hydraulic turbines nozzles, can lead to important interface deformations without phase changes. These kinds of deformations are really hard to catch with eulerian numerical methods since the diffusion of the interface smoothes the sharpness of the physical interface perturbations. On an other hand mesh-less/lagrangian methods as SPH offers the possibility to naturally simulate flows with large deformations. Furthermore previous work on SPH-ALE formalism brought improvement in term of precision, stability of single phase flows [8]. Different approaches were developed to simulate multiphase flows with SPH method [5], [3], [6] or [7]. The goal of the present study is to bring improvements of the methods presented in [7]. In the particular case of low Mach numbers, preconditioned Riemann solvers are introduced to limit the numerical diffusion linked with the method.

II. The multifluid formalism

The main characteristics of the multifluid formalism presented in [7] are recalled here.

A. The multifluid model in ALE Formalism

The arbitrary Lagrange-Euler (ALE) formalism considers conservative form of Euler equations in a moving frame of reference at the velocity v_0.

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

where Ω is the volume of control, S its boundary, Φ the vector of conservative variables and Q_S and Q_V the surface and volume source terms. If surface terms are restricted to pressure term, the equation (1) can be rewritten:

$$L_{v_0}(\Phi) + div\left(F_E(\Phi) - v_0\Phi\right) = Q_v \tag{2}$$

where L_{v_0} is the transport operator associated to v_0 and F_E is the flux vector of the Euler equations. The discretization of this conservative formulation leads to the appearance of one dimensional Riemann problems between each pair of control volumes [13]. If we consider two control volumes i and j, their interaction results in contributions along the direction joining i and j, with a discontinuous state evolution at mid-point. It can be expressed as:

$$\begin{cases} \dfrac{\partial}{\partial t}(\Phi) + \dfrac{\partial}{\partial x^{n_{ij}}}\left(F_E(\Phi).n_{ij} - v_0(x_{ij},t).n_{ij}\Phi\right) \\ \Phi(x^{(n_{ij})},0) = \begin{cases}\Phi_i \ if \ x^{(n_{ij})} < 0 \\ \Phi_j \ if \ x^{(n_{ij})} > 0\end{cases}\end{cases} \tag{3}$$

where n_{ij} is the unit vector between i and j (from i to j), x_{ij} is the mid-point between i and j, $x^{n_{ij}}$ is the curvilinear abscissa along the straight line between i and j, whose origin is taken at x_{ij} and Φ_i and Φ_j the vector of conservative variables at i and j respectively. The final set of discrete equations in the SPH-ALE formalism is:

$$\begin{cases} \dfrac{d}{dt}(x_i) = v_0(x_i,t) \\ \dfrac{d}{dt}(\omega_i) = \omega_i \sum\limits_{j\in D_i} \omega_j (v_0(x_j) - v_0(x_i))\nabla_i W_{ij} \\ \dfrac{d}{dt}(\omega_i\rho_i) + \omega_i \sum\limits_{j\in D_i} \omega_j 2\rho_{E,ij}(v_{E,ij} - v_0(x_{ij},t)).\nabla_i W_{ij} = 0 \\ \dfrac{d}{dt}(\omega_i\rho_i v_i) + \omega_i \sum\limits_{j\in D_i} \omega_j 2[\rho_{E,ij} v_{E,ij}\otimes(v_{E,ij} - v_0(x_{ij},t)) + p_{E,ij}].\nabla_i W_{ij} = \omega_i\rho_i g \end{cases} \tag{4}$$

where $(\rho_{E,ij}, v_{E,ij})^t = \Phi_{ij}(\lambda_0^{ij})$ is the upwind solution of the moving Riemann problem.This formalism has different

Modeling Surface Tension in SPH by Interface Reconstruction using Radial Basis Functions

Björn Andersson, Stefan Jakobsson, Andreas Mark, Fredrik Edelvik
Fraunhofer-Chalmers Centre
Chalmers Science Park
Göteborg, Sweden
{bjorn.andersson, stefan.jakobsson, andreas.mark, fredrik.edelvik}@fcc.chalmers.se

Lars Davidson
Department of Applied Mechanics
Chalmers University of Technology
Göteborg, Sweden
lars.davidson@chalmers.se

***Abstract*—A novel method for reconstructing the interface between two fluids is described and evaluated. The method uses a different basis for approximating the color function than what is common practice in SPH simulations. The key feature of the new method is the ability to omit small length scale structures in order to obtain a smoother representation. A smoother interface is more suitable to derive a surface tension force from, as the magnitude of the force is proportional to the curvature which is related to the second derivative along the interface.**

I. Introduction

At an interface between two different fluids, or a fluid and a solid wall, there is in general a surface tension associated with it. Several attempts have been made to model this effect within the framework of *Smoothed Particle Hydrodynamics* (SPH). Morris [1] describes a method where the surface tension is modeled on a macroscopic level using the curvature of the interface. Tartakovsky and Meakin [2] model the surface tension as inter-particle forces, which cancel out in the bulk of the fluid. Both methods are so-called *Continuum Surface Force* (CSF) methods [3], where the surface tension is modeled as a volume force in a narrow region close to the interface. In this way more particles than those immediately close to the interface experience the surface tension.

The method described here is similar to that of Morris in that it attempts to estimate the curvature of the interface in a macroscopic sense. The general motivation for doing so is that the surface tension force acting on a fluid element can be written

$$\boldsymbol{F}_s = \sigma\kappa\hat{\boldsymbol{n}} - \nabla_S\sigma, \tag{1}$$

where σ is surface tension coefficient, κ is the mean curvature of the interface, $\hat{\boldsymbol{n}}$ is the interface normal and ∇_S is the interface tangential differential operator. The latter term, known as the Marangoni effect, is in the following assumed to be zero and is not considered.

The surface acting as an interface between the different phases is tracked by means of a *color function*, $C(\boldsymbol{x})$. Each of the phases is assigned a different color which is propagated with the fluid. In a Lagrangian method, such as SPH, this is particularly easy as each particle is assigned a color at the start of the simulation and it is then kept at that same color throughout the simulation. The color field is then used to evaluate the surface tension. The direction of the surface tension force is evaluated as the gradient of the color function, and the strength is proportional to the second derivative along the interface. Therefore it is important to have a sufficiently smooth color function that does not vary too rapidly in space compared to the size of the SPH particles.

The standard SPH framework offers a couple of different options for discretization of the curvature and surface normals based on the color function [1], [4]. They all have in common that the length scale of the correlation of the color function is of the same order as the size of the particles. The reason for this is that the color field is determined from the particles alone, using SPH interpolation. However, the length scale of the second derivative will be much shorter, and as a result the estimate of the curvature will be strongly dependent on the particle distribution.

The remedy to this problem suggested here is to use a different basis for interpolating and/or approximating the color function. The framework chosen is *Radial Basis Function*s (RBF) [5], which shares some common features with SPH. One significant advantage of the RBF approach is the possibility to introduce a relaxation parameter that can be varied continuously, resulting in pure interpolation in one limit, and a very smooth, but crude approximation in the other limit. This parameter is then used to obtain a reconstruction of the interface having appropriate smoothness on the length scale of the SPH particle radius.

The smoothness is particularly important if the two phases solved for have a large density difference. If, for example, water droplets in air are studied, the density ratio is about 1000 to 1, and depending on the situation the air flow may have negligible influence on the water droplets. If that is the case, it is enough to solve for the denser phase, in effect treating the other as massless. This will in the following be the case; only the denser phase is treated, turning the interface into a free surface.

The outline of the paper is as follows: In Section II the governing equations and their discretization is described. Section III gives an overview of the RBF results relevant to the present work. In Section IV the reconstruction of the interface is described and the expression for the surface tension force

3D drop deformation and breakup in simple shear flow considering the effect of insoluble surfactant.

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

***Abstract*—We have developed a multi-phase SPH method to simulate arbitrary interfaces containing surface active agents (surfactants) that locally change the properties of the interface, such the surface tension coefficient [1]. Our method incorporates the effects of surface diffusion, transport of surfactant from/to the bulk phase to/from the interface and diffusion in the bulk phase. Neglecting transport mechanisms, we use this method to study the impact of insoluble surfactants on drop deformation and breakup in simple shear flow.**

I. Introduction

Exposing drops to extensional flows such as e.g. a simple Couette flow, the viscous forces along the interface tend to deform the drop and elongate it to an ellipsoid-type shape. The balancing force to stop the deformation due to the shearing is the surface tension. When two pure fluids of different types are in contact, the resulting surface tension force is only proportional to the local curvature and normal to the interface. Depending on the strength of this force and the viscosity ratio between the two fluids λ, drops are deformed to a steady ellipsoid shape or break up. The correlation between the breakup behaviour and the flow parameter is known as the Grace curve [2].

Adding surface active agents (surfactants) to a multiphase system can strongly alter the flow phenomena. Neglecting the effect of such an additive on other material properties, surfactants mainly change the surface tension coefficients between two fluids when replacing fluid molecules at the interface with surfactant molecules. Hence, surface tension gradients along the interface can occur resulting in the so-called Marangoni forces [3].

Here, we only focus on the case of insoluble surfactants, i.e. surfactants are initially added to the interface and cannot dissolve to the adjacent fluid phases. Bazhlekov et al. [4] studied the effect of insoluble surfactants on drop deformation and breakup in simple shear flow with a boundary-integral method and clearly describe the different breakup modes. But due to the nature of their method, an interface capturing scheme is required and breakup is detected manually. By the use of a Lagrangian particle method we avoid these algorithms and handle interface deformations naturally.

In the following section we briefly introduce the governing equations for multiphase flows with surfactants. Exemplary we show a 3D simulation with steady deformation and validate it against analytical data. Finally we present a detailed study of the different breakup modes where we focus on the effect of tip streaming.

II. Governing equations

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

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

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

where ρ, p, $\mathbf{v}$, and $\mathbf{g}$ are material density, pressure, velocity and body force, respectively. $\mathbf{F}^{(\nu)}$ denotes the viscous force and $\mathbf{F}^{(s)}$ is the interfacial surface force.

Following the weakly-compressible approach [5], an equation of state (EOS) is used to relate the pressure to the density

$$p = p_0\left(\frac{\rho}{\rho_0}\right)^{\gamma}+b\ , \tag{3}$$

with $\gamma = 7$, the reference pressure p_0, the reference density ρ_0 and a parameter b. These parameters and the artificial speed of sound are chosen following a scale analysis presented by Morris et al. [6] which determines the threshold of the admissible density variation.

Assuming incompressibility, the viscous force $\mathbf{F}^{(\nu)}$ simplifies to

$$\mathbf{F}^{(\nu)} = \eta\nabla^2\mathbf{v}\ , \tag{4}$$

where η is the dynamic viscosity. Following the continuum-surface-tension model [7], the surface force can be expressed as the gradient of the surface stress tensor with the surface tension coefficient α

$$\mathbf{F}^{(s)} = \nabla\cdot\left[\alpha\left(\mathbf{I}-\mathbf{n}\otimes\mathbf{n}\right)\delta_\Sigma\right] = -\left(\alpha\kappa\mathbf{n}+\nabla_s\alpha\right)\delta_\Sigma\ . \tag{5}$$

The Capillary force $\alpha\kappa\mathbf{n}\delta_\Sigma$ is calculated with the curvature κ, the normal vector of the interface $\mathbf{n}$ and the surface delta function δ_Σ. This expression describes the pressure jump condition normal to an interface. In case of surface tension variations along the interface (e.g., due to non-uniform temperature or surfactant concentration) the Marangoni force $\nabla_s\alpha\delta_\Sigma$ results

An Alternative Approach to Modelling Complex Smoothed Particle Hydrodynamics Boundaries

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

***Abstract*—This contribution presents a strategy to treat boundaries based on surface normals that are calculated from local feature information of the triangular surface mesh. The presented strategy involves the calculation of boundary forces as well as an approach to prevent particles from interacting through solid boundaries. The strategy is capable to handle arbitrarily thin and highly nonconvex surfaces. It is illustrated that the presented strategy is especially useful in parallel simulation environments with load balancing that is based on domain decomposition. There, treating surface triangles as independent entities allows for an efficient combined dynamic distribution of SPH particles and elementary surface geometry to balance workload and to minimize memory consumption.**

I. Introduction

There exist a variety of approaches to model solid boundaries in Smoothed Particle Hydrodynamics (SPH) simulations, see [1], [2], [3]. The probably most widely used approach is to introduce fixed boundary particles as presented e.g. in [4]. Another way to treat boundaries that is especially viable in engineering applications, where boundary geometries are usually provided by CAD software, is to use polygonal surface discretizations. To incorporate CAD data into SPH simulations using triangular surface meshes is simple, flexible and robust. Furthermore, combined with boundary potential forces, triangular meshes are advantageous in regions with low boundary curvature. There the feature resolution through triangles can be rough which significantly helps to reduce the memory and computing overhead in comparison to the use of fixed boundary particles.

When it comes to parallel simulations, it is common to divide the simulation work by means of domain decomposition and particle grouping [5]. In such simulations the whole triangle based boundary geometry can either be identically stored on every computing node or it can be divided into subsets that represent those triangles that are situated within a processor's associated subdomain. The latter is advantageous if surface geometry triangles are treated as individual entities that are equivalent to particles in a particle simulation sense. Then, no mesh topology needs to be stored and the triangles can be reassigned along with the SPH particles among the different computing nodes. This approach speeds up the boundary force computation if very complex boundary geometries are considered. Individual interactions between SPH particles and the surface geometry are thereby treated by superimposition of SPH particle-triangle interactions.

In contrast to a boundary description that assigns a copy of the whole boundary geometry to every computing node, the aforementioned approach requires alternative means to compute boundary forces as there is no explicit information about boundary topology, such as triangle adjacencies or surface curvature. Furthermore, regarding slender parts where inter boundary distances across solid regions are smaller than the SPH cutoff radii, additional thought has to be spend on preventing moving particles from interacting through the boundary geometry. To overcome this restriction, in this work local surface informations are calculated for each particle that can be used to compute boundary forces and to decide if a pair of SPH particles is separated by a solid boundary. Different decision strategies are introduced and evaluated.

The software used in this work is Pasimodo [6], a Lagrangian simulation framework for the 3D simulation of granular materials and fluid models. The implementation of SPH in the program is based on [7].

II. Boundary normal calculation

Unfortunately, there is no knowledge of other particles in the vicinity when evaluating an interaction between a pair of particles. There is only information about the two particles in contact. Considering interactions between pairs of SPH particles, this yields inaccurate results if separating solid boundaries are ignored. Furthermore, it can lead to unphysically strong forces for SPH particle-triangle interactions if the boundary curvature is unknown. Therefore, some kind of information needs to be calculated in advance about the surrounding boundary geometry for each SPH particle. Then, this information is used to calculate a boundary force on the particle and to help decide, whether a pair of SPH particles truly interacts or not. One approach is to calculate a boundary normal for each SPH particle at the begin of each simulation step. It points from the triangular surface geometry towards the particle. In addition to the boundary normal, a minimal distance to the boundary surface is calculated and stored for each SPH particle.

To begin with, the boundary normal $\boldsymbol{q}$ of an SPH particle results from a summation over the triangles within its contact

SPH no-slip BC implementation analysis at the continuous level

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

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

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

***Abstract*—The aim of this paper is to discuss whether there are structural problems in three of the most representative velocity mirroring techniques used to force no-slip boundary conditions (BC) in SPH for Newtonian incompressible flows. We refer specifically to fixed fluid particles, ghost particles, and Takeda *et al.* [1] boundary integrals. In Newtonian incompressible flows, the viscous related terms in the momentum conservation equation depend on the evaluation of the Laplacian of the velocity field. In order to analyze such techniques, the continuous version of the Laplacian approximation by Morris et al. [2] and Monaghan-Cleary-Gingold [3] viscous terms has been considered. It has been shown that there are intrinsic inaccuracies in the computation of the Laplacian close to the boundaries and the onset of singularities in such evaluation for some flows and mirroring techniques combinations. The impact of these deviations in the SPH simulation of viscous flows is not clear at this stage.**

Symbols

MVT	Morris et al. viscous term [2]
MCGVT	Monaghan-Cleary-Gingold [3] viscous term
BC	Boundary condition(s)
FFP	Fixed fluid particles
GP	Ghost particles
ASM	Anti-symmetry model for velocity mirroring
U0M	Zero velocity mirroring model

I. Introduction

The modeling of no-slip BC plays an important role in the simulation of many important physical phenomena like boundary layers, separation, transition flows, etc.., and in the computation of important magnitudes in Engineering like the viscous drag force. Our aim with this paper is to discuss whether a set of very representative mirroring techniques of the velocity field used to force no-slip BC for Newtonian incompressible flows have essential problems in doing so, at the continuous level of the SPH formulation. In Newtonian incompressible flows, the viscous related terms in the momentum conservation equation depend on the evaluation of the Laplacian of the velocity field, which presents difficulties in general, and in particular, close to the boundaries. Therefore, the analysis should focus not in the value of the velocity at the boundary but in the value of the Laplacian. Although some general results are outlined, the analysis presented here is treats mainly linear and quadratic velocity fields.

In a previous paper [4], we had already started the study of this problem by analyzing the accuracy of the continuous version of Monaghan-Cleary-Gingold viscous term [3] for the evaluation of the Laplacian of quadratic velocity fields. The ghost particle (GP) technique had then been considered. The analysis had a limited scope but some incongruities in the evaluation of the Laplacian close to the boundaries were shown. In the present paper, such analysis is extended by using as well the general formulation of Español and Revenga [5] for the computation of the Laplacian. The SPH discretization of such term leads to the well know Morris et al. viscosity term [2]. On top of this extension, fixed fluid particles as well as the evaluation of the boundary effects with Takeda *et al.* [1] integrals have been considered.

The paper is organized as follows: first, the field equations and the SPH formalism considered is presented and discussed. Second, the selected implementations of no-slip BC techniques are introduced. Third, two $2D$ given velocity fields are considered in order to check the performance of the viscous terms at the continuous level close to the boundaries. Finally the evolution of a plane Couette and Poiseuille flows is studied in order to assess the validity of the conclusions obtained from the previous analysis.

II. Governing equations

A. Field equations

The aim of this paper is to deal with Newtonian incompressible flows. The incompressible Navier-Stokes equations in Lagrangian formalism are hence taken as the field equations:

$$\frac{D\boldsymbol{r}}{Dt} = \boldsymbol{u}\,, \quad \nabla\cdot\boldsymbol{u} = 0\,, \quad \frac{D\boldsymbol{u}}{Dt} = \boldsymbol{g} + \frac{\nabla\cdot\mathbb{T}}{\rho}\,, \tag{1}$$

In these equations, ρ is the fluid density and $\boldsymbol{g}$ is a generic external volumetric force field. The flow velocity, $\boldsymbol{u}$, is defined as the material derivative of a fluid particle position $\boldsymbol{r}$. $\mathbb{T}$ is the stress tensor of a Newtonian incompressible fluid:

$$\mathbb{T} = -p\boldsymbol{I} + 2\mu\,\mathbb{D}\,, \tag{2}$$

Improved Solid Boundary Treatment Method for the Solution of Flow over an Airfoil and Square Obstacle by SPH Method

Mostafa Safdari Shadloo, Amir Zainali, Mehmet Yildiz
Faculty of Engineering and Natural Sciences
Sabanci University, Orhanli 34956 Tuzla, Istanbul, Turkey
mostafa@sabanciuniv.edu, amirz@sabanciuniv.edu, meyildiz@sabanciuniv.edu

Abstract— **In this article, we present numerical solutions of flow over an airfoil and square obstacle using Incompressible Smoothed Particle Hydrodynamics (ISPH) method with an improved solid boundary treatment, referred to as Multiple Boundary Tangents (MBT) method. It was shown that MBT boundary treatment technique can be used effectively for tackling boundaries of complex shapes. Also, we have proposed the usage the repulsive component of the Leonard Jones Potential (LJP) in the advection equation to repair the fracture occurring in SPH method due to the tendency of SPH particles to follow the stream line trajectory. Additionally, for the solution of these two benchmark problems, we have derived a correction term in SPH discretization. Numerical results suggest that the utility of MBT method, fracture repair algorithm and corrected ISPH discretization scheme together enables one to obtain very stable and robust SPH simulations. The square obstacle, and NACA airfoil geometry with the angle of attacks between 0-15 were simulated in a flow field with Reynolds numbers as high as 1600. The SPH results are validated with a mesh-dependent Finite Element Method (FEM) solver, and excellent agreements among the results were observed.**

I. INTRODUCTION

Smoothed particle hydrodynamics (SPH) is one of the members of meshless methods used to solve partial differential equations widely encountered in scientific and engineering problems. Due to being a relatively new computational method for engineering applications (roughly for two decades old), there are still a few significant issues with SPH that need to be scrutinized. It is still a challenge to model physical boundaries correctly and effectively [1]. In addition, there are various ways to construct SPH equations (discretization), and a consistent approach has not gained consensus. Highly irregular particle distributions which occur as the solution progress may cause numerical algorithms to break down, thereby making robustness another significant issue to be addressed.

In most engineering problems, the domain of interest is, in general, bounded. The SPH formulations being valid for all interior particles are not necessarily accurate for particles close to the domain boundary since the kernel function is truncated by the boundary. Therefore, the application of boundary conditions is problematic in the SPH technique, and the correct boundary treatments have been an ongoing concern for an accurate and successful implementation of the SPH approach in the solution of engineering problems with bounded domains.

Hence, over the years, several different approaches have been suggested for the boundary treatment such as specular reflections, or bounce-back of fluid particles with the boundary walls, Lennard-Jones Potential (LJP) type force as a repulsive force, and ghost particles [2]. The current status of ghost particle implementation is limited to relatively simple geometries. Therefore, in our previous research efforts on SPH, we suggested an improved solid boundary treatment, as an extension to the ghost particle approach, which eliminates many shortcomings of the aforementioned boundary treatments. This approach is called the multiple boundary tangent (MBT) method [3]. The approach was tested on several bench mark flow problems such as lid-driven cavity flow, flow over a cylindrical obstacle, and flow over a backward facing step [4]. The test problems solved using the MBT method was relatively simple. To test and then understand the full power and possible limitations of the MBT method, we have solved geometrically more complex flow problems such as flow over an airfoil and square obstacle. Therefore, this paper presents a follow-up work on the implementation of MBT method to more complex solid boundary geometries.

II. CORRECTED SPH FORMULATION

The three-dimensional Dirac-delta function $\delta^3(r_{ij})$, also refer to as a unit pulse function, is the starting point for the SPH approximation technique. This function satisfies the identity

$$f(\vec{\mathbf{r}}_i) = \int_{\Omega} f(\vec{\mathbf{r}}_j)\delta^3(r_{ij})\, d^3\vec{\mathbf{r}}_j, \tag{1}$$

where $d^3\vec{\mathbf{r}}_j$ is a differential volume element and Ω represents the total bounded volume of the domain.

The SPH method to simulate the model test of a sandy river levee on seepage induced failures

Hirotoshi Mori, Yukiko Saito, Tetsuya Sasaki
Soil Mechanics and Dynamics Research Team
Public Works Research Institute
Tsukuba, Japan
hi-mori@pwri.go.jp

Kenichi Soga
Dept. of Engineering
University of Cambridge
Cambridge, UK

Reiko Kuwano
Institute of Industrial Science
University of Tokyo
Tokyo, Japan

Abstract—**The failure of river levees induced by seepage is generally designed or evaluated by the circular slip method. The method has been used for decades and is the simple method to calculate whether a failure would take place. However, it is impossible to simulate the process of failures in detail.**

The SPH method is used to simulate the failure mode in the model test of a sandy river levee. The experiment used a large scale model and added a static water pressure and rainfall. The progressive failure with a large deformation was observed as the result of the experiment.

The code for the SPH method was developed with a soil-water coupling model. The model consists of 46,051 the SPH particles using the Mohr-Coulomb model. The effects of the pore pressure, the internal erosion and the matrix suction were considered in the model. The result of calculation shows a progressive failure with large deformation similar to the experiment. The failure mode is that the initial shear band would be generated at the toe of the model. Then, it develops to the inside of the model. The internal erosion or the matrix suction affects the failure mode in the size or the shape of the failure. The results of the simulation indicate that the cause of progressive failures with large deformation would be the combination of the circular slip and the internal erosion.

The SPH method successfully give the insight of the progressive failure observed in the model test.

I. Introduction

A. Background

Floods are natural disasters which can cause catastrophic damages. For example, Hurricane Katrina, which hit New Orleans in the United States in August 2005, killed 1,836 people (with 705 people still missing) and caused 81.2 billion U.S.dollars of damage [1]. In Japan, more than 20 rivers flooded in 2004 and several river levee failures occurred as shown in Fig. 1. This is due to the unusually large number of typhoons, which hit that year (10 compared to the annual average of 2.6 between 1971 and 2000). If we turn our eyes to a developing country, 130,000 people are dead or missing from Cyclone Nargis in Burma in 2008. Hence, flood control is still an important issue both in developed and developing countries.

A river levee is one of the major earth structures for flood control. Although, river levees are known to fail by (1) overtopping, (2) soil erosion and (3) increased seepages. Overtopping occurs when the height of river water exceeds

Fig. 1. River levee failure (Maruyama River, Japan, 2004)

that of the river levee. At the initial stage, overtopped river water erodes the top/toe of the back slope of the river levee. Then, the erosion expands to the whole slope. Finally, river water takes away all the river levee and causes flooding. Soil erosion is caused by the rapid flow of river water. The tractive force of river water removes the front slope surface of the river levee leading to river levee failures. The failure by soil erosion is observed especially in fast-flowing rivers. The failures of river levees by seepage have two major mechanisms: (a) a progressive failure and (b) a progressive piping. The progressive failure of river levees by a circular slip and/or an internal erosion are caused simultaneously by the increased pore pressure and the weakened strength of soil by seepage water as shown in Fig. 2. If the failure reaches the front slope, the river levee would fail completely. The progressive piping by increased hydraulic gradient has the possibility to take away the inner soil of the river levee basement by the fast flush of seepage flow. The loss of the soil in the basement rapidly leads to the instability or the settlement of the river levee.

The major mechanism of river levee failures by seepage is an interaction between soil and seepage water, whereas that by overtopping and soil erosion is caused mainly by the tractive force of river water.

Hydraulic jump simulation by SPH

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

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

***Abstract*— In a hydraulic jump, a supercritical open-channel flow dissipates a large part of its kinetic energy, usually through a strong turbulent roller, and turns into a subcritical flow. Several hydraulic phenomena (strong turbulence effects, downstream wave propagation, unsteadiness, air entrainment) characterize the different jump types (undular, weak, steady, strong. etc..) which can occur depending on the value of the upstream Froude number. The hydraulic jump appears therefore to be a suitable test case to validate different SPH models developed to analyse unsteady turbulent open-channel flows. The present paper investigates the influence of numerical parameters and turbulence models by comparing 2D and 3D SPH solutions with reference literature data (Chow, 1959), as well as with laboratory data (Ben Meftah et al, 2008). The comparison between SPH and experimental results shows a relevant influence of smoothing procedures and of different turbulence models, such as mixing length (De Padova et al, 2009) or Standard k-ε, on the free-surface profile description; however, the agreement of computed velocity profiles and water elevations with laboratory data confirms the adequacy of the SPH description.**

I. INTRODUCTION

Hydraulic jumps occur when a supercritical open-channel flow dissipates a large part of its kinetic energy and turns into a subcritical flow. Dissipation takes place usually through a strong turbulent roller which characterizes the jump. Different types of hydraulic jump (undular, weak, steady, strong. etc..) can occur depending on the value of the upstream Froude number *Fr*. The present paper discusses the capability of the SPH method to model different types of hydraulic jumps, with a particular attention on the modelling of the development of high-Froude undular jumps. Owing to the variety of different hydraulic phenomena (strong turbulence effects, downstream wave propagation, unsteadiness, air entrainment) which characterize the different jump types, the hydraulic jump appears to be a suitable test case to validate different SPH models developed to analyse unsteady turbulent open-channel flows.

In a previous study (De Padova et al., 2009), 2D SPH modelling was applied to the simulation of an undular jump at $Fr = 3.9$ and successfully tested using physical experiments on supercritical flow motion by Ben Meftah et al. (2007; 2008). However, since SPH simulations of hydraulic jumps over sills (Gallati and Braschi, 2003) showed a certain degree of dependency of the results on the choice of the SPH numerical parameters, the present paper intends to investigate the influence of some of these parameters and of the possible turbulence models by comparing 2D SPH solutions of a weak jump at $Fr = 2$ with reference literature data.

The application of SPH modelling to the analysis of high Froude-number undular jumps is then discussed. 2D results regarding a jump at $Fr = 8.3$ are compared with laboratory data obtained in a very large channel of the Coastal Engineering Laboratory of the Water Engineering and Chemistry Department at Politecnico di Bari (Ben Meftah et al, 2008). The agreement of computed velocity profiles and water elevations with laboratory data confirms the adequacy of the SPH description.

However, a complete description of this type of jump requires a full 3D representation because, when the channel is very large, the hydraulic jump front is trapezoidal (Ben Meftah et al., 2007; 2008). In order to analyse and reproduce also the lateral shock waves (Ben Meftah et al., 2007; 2008), a 3D SPH model was used to simulate the jump at $Fr = 3.9$.

II. EXPERIMENTAL SET UP

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

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

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

For the measurement of velocity, a two-dimensional Nortek ADV system was used, together with CollectV software for the data acquisition and ExploreV software for the data analysis, all of them products of Nortek. Water height was measured using an ultrasonic measuring system UltraLab

Prediction of Sediment Scouring through SPH

Manenti S., Sibilla S., Gallati M.
Dipartimento di Ingegneria Idraulica e Ambientale
Università degli studi di Pavia, Via Ferrata, 1 – 27100
Pavia (Pv), Italy
sauro.manenti@unipv.it

Agate G., Guandalini R.
Environment and Sustainable Development Dept.
ERSE s.p.a., Via Rubattino, 54 – 20134
Milan, Italy

***Abstract*—This work presents the advances in modelling scouring phenomena of non-cohesive sediments induced by free-surface water flows through the SPH approach; in particular this paper is focused on the selection of the strength criterion for defining the failure mechanism and the incipient motion of the bottom sediments under hydrodynamic friction. Two approaches have been analyzed: one is derived from the Mohr-Coulomb yield criterion, in which some improvements have been realized to include the effect of the fluid properties; the other is based on the more classic Shields approach [11]. A preliminary sensitivity analysis has been carried out in order to provide qualitative assessment of the influence of the model parameters on the erosion process; subsequently an experimental test case has been simulated for evaluating the accuracy provided by both yielding criteria.**

I. Introduction

Sediment scouring induced by free-surface water flows represents a topic of great concern in several fields of applied engineering such as operation of an artificial reservoir [6] and design-maintenance of partially submerged structure [4].

In this context, the SPH method appears a promising numerical approach for simulating the scouring process in non-cohesive sediment deposits induced by a rapidly varied water flow; actually, a first model was presented for simulating local sediment erosion and flushing hydrodynamics induced by opening manoeuvre of dam's bottom outlet [7]. Comparison between numerical results and measured laboratory data from an experimental reservoir proved that the SPH model can reproduce most of the relevant physical and engineering aspects of the process; however, in peculiar test conditions it seems to suffer from some intrinsic limitations, such as the dependency of the sediment erosion criterion on the fluid properties.

Some improvements to the numerical model have been therefore implemented, tested and compared with the previous version, in order to develop a more general tool which is able to predict the fluid-solid coupled dynamics to quantify the scouring levels in other engineering fields of application.

In Sect. II a brief description of the first model based on the Mohr-Coulomb theory is given, including some improvements that have been introduced; the theoretical aspects concerning an alternative erosion criterion derived from Shields theory for sediment transport mechanics in river flow is then described.

Sect. III illustrates the numerical results obtained with both erosion criteria: sensitivity of the erosion process to changes in models parameters is firstly evaluated from a quantitative point of view; then the results of the validation test are presented, in which a two dimensional flushing experiment is simulated and the final eroded profiles obtained with both Mohr-Coulomb and Shields criteria are compared with laboratory data.

Finally Sect. IV illustrates the main conclusions of the work.

II. Theoretical Aspects

In the following basic theoretical details of the two yielding mechanisms implemented in the SPH code are provided.

It should be stated that, in general, non-cohesive sediments are modelled by means of particles treated differently whether they are at rest or suspended in the water column.

In the first condition, granular particles are treated as a fixed boundary and excluded from the computation of the velocity and density fields; their pressure is imposed according to the lithostatic condition and included in the pressure smoothing of the surrounding fluid particles.

Once the grains are moved they are treated as a viscous slightly compressible pseudo-fluid whose motion responds to the same discretized governing equation as for water [7].

The decision on whether a solid particle is at rest or not is based on two alternative erosion criteria described below.

A. Mohr-Coulomb Erosion Criterion

This yielding mechanism is based on the theoretical model described in [7] and here briefly recalled.

It assumes that the behaviour of the granular material could be schematized through a non-Newtonian rheological model; the critical condition for incoming motion of a solid particle (i.e. the hydrodynamic frictional stress at the fluid-grain interface overcomes the critical strength threshold of the sediment) is translated into the inequality expressed by Eq. (1) which relates the critical viscosity threshold μ_{max} to the dynamic viscosity: the latter is calculated on the basis of the fluid and solid parameters, i.e. the second invariant of the rate of deformation tensor Π_D and, respectively, internal frictional angle ϕ, cohesion c and effective pressure p'.

SPH Modelling of Water/Soil-Suspension Flows

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

***Abstract*—The paper reports the development and validation of an SPH based water/soil-suspension model and its application to marine engineering flows. Adequate models have been implemented into the massively parallel hydrodynamic SPH code GADGET-$^{\mathrm{H}}2^{\mathrm{O}}$ which is a modification of the cosmological TreeSPH code GADGET-2. Validation studies refer to two multiphysics cases, i.e. an erosional dam-break and an idealized flushing manoeuvre, supplemented by a sequence single phase cases. The investigated application focuses on the generation of potholes in front of quay walls induced by transverse thrusters during an unberthing manoeuvre of a ship. Such erosions can significantly weaken the wall support and lead to cost-intensive counter measures.**

I. INTRODUCTION

The interaction of water, soil and structures poses challenges to marine and hydraulic engineering. In the field of ocean and harbour engineering, the generation of potholes (scours) around platform legs or next to quay walls is a frequent problem. The predicition of related erosion processes requires accurate numerical models for the involved water/soil-suspensions.

The present study aims to enhance the modelling capabilities of the massively parallel hydrodynamic SPH-code GADGET-$^{\mathrm{H}}2^{\mathrm{O}}$ [12] towards multi-phase flows involving water/soil-suspensions. GADGET-$^{\mathrm{H}}2^{\mathrm{O}}$ [12] is a modification of the cosmological TreeSPH-code GADGET-2 [11]. It shows a linear speed-up using several hundred CPU-cores when applied to large-case hydrodynamic problems with a couple of million particles. The fluid is assumed to be Newtonian and turbulence is modelled by means of a LES approach. The soil model considers the granular material as a fluid with a variable viscosity which is evaluated in line with the Mohr-Coulomb yield-stress criterium for cohesive or cohesionless materials. As opposed to this rather simple dry-soil model, the suspension model is given more attention. A concentration based approach to mimic the stresses inside the fictitious suspension layer is introduced which is derived from a Chézy-relation between the shear stresses and the local flow velocity as proposed by Fraccarollo and Chapart [4].

Validation examples refer to pressure and shear driven single phases as well as suspension flows. A complex application example shows the simplified unberthing of a container ship.

The paper is structured as follows: In section II, the employed governing equations are described. Section III shows validation cases while parameter sensitivity of the suspension is discussed in section IV. The application example is presented in section V. Final conclusions are summarized in section VI.

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 j refers to its neighbours. Mind, that Einstein's summation is employed over repeated Greek superscripts.

A standard symmetric cubic spline-kernel function W is used which complies with the Delta-Dirac-function δ_D in the continuous limit and adheres to normalisation principle. It drops to zero when the particle distance approaches the kernel length h. The kernel length can vary in GADGET-$^{\mathrm{H}}2^{\mathrm{O}}$, but is kept constant (2.4 times particle spacing) in the present research, as no benefits are expected from a variable smoothing length for incompressible fluids.

A. Conservation of Mass and Momentum

The transient evolution of the particle density follows from the continuity equation

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

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

The particles' momentum is governed by the Navier-Stokes equation, which is approximated using the following integral representation

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

Here, f^α refers to a volumetric force and $\sigma^{\alpha\beta}$ denotes to the stress tensor. The latter is split into an isotropic pressure portion and a remainder

$$\sigma^{\alpha\beta} = -p\delta^{\alpha\beta} + \tau^{\alpha\beta} \,, \tag{3}$$

with the pressure p, the unit tensor $\delta^{\alpha\beta}$.

Modelling the flow of self-compacting concrete by SPH

Kulasegaram S, Karihaloo BL, Ghanbari A
School of Engineering
Cardiff University
Cardiff, CF24 3AA, UK

***Abstract*— A Lagrangian particle based method, smooth particle hydrodynamics (SPH), is used to model the flow of self-compacting concretes (SCC) with or without short steel fibres. An incompressible SPH method is presented to simulate the flow of such non-Newtonian fluids whose behaviour is described by a Bingham-type model, in which the kink in the shear stress vs shear strain rate diagram is first appropriately smoothed out. The viscosity of the self-compacting concrete is predicted from the measured viscosity of the paste using micromechanical models in which the second phase aggregates are treated as rigid spheres and the short steel fibres as slender rigid bodies. The basic equations solved in the SPH are the incompressible mass conservation and Navier-Stokes equations. The results of the numerical simulation are benchmarked against actual slump tests carried out in the laboratory.**

I. INTRODUCTION

In recent years, self-compacting concrete (SCC) is increasingly replacing conventional concrete in the construction industry. The need for very high durability structures demands excellent flow-ability of SCC into the formwork. The filling behaviour of SCC is even more difficult to predict in the presence of reinforcing steel and formworks of complex shape. Therefore, in order to produce high quality SCC structures it is vital to understand fully the flow characteristics of SCC. The most cost-effective way to gain such understanding is by performing numerical simulations. These will not only enable us to understand the filling behaviour but will also provide insight into how the fibres will orientate themselves during the flow of SCC containing fibres.

In practice it is not easy to model SCCs or any such heterogeneous materials flow due to their very constitutive properties. Any computational model for SCC flow should be able to describe accurately the rheological behaviour of SCC and to follow the large deformation and Lagragian nature of the flow. A number of computational strategies have been attempted in the past to simulate the SCC flow by assuming concrete as a homogeneous viscous fluid and using either the discrete element method [1],[2] or the Lagrangian finite element method [3]. An overview of various computational techniques used in the past to model concrete flow and their advantages and disadvantages is given in [4]. Due to the Lagrangian nature of the SCC flow and due to the fact that an SCC mix is essentially an aggregate of particles of different sizes and shapes, the use of particle-based Lagrangian numerical techniques to simulate such flows is both more appropriate and simpler than the traditional mesh-based methods [1]-[4]. Therefore, in this paper, a Lagrangian particle-based technique, the so-called smooth particle hydrodynamics (SPH) method [5]-[8] is chosen for simulating the SCC flow.

The simplicity and Lagrangian nature of SPH have been exploited in the past to model many free-surface fluid flows and related engineering problems [5]-[8],[10],[11]. To simulate the SCC flow, an incompressible SPH methodology is adopted in this paper. The formulation relating to incompressible SPH and coupling of SPH with a suitable Bingham model to represent the rheological behaviour of SCC are briefly discussed in Section II. In Section III, numerical results for SCC flow with and without steel fibres are compared with some available experimental observations. Section IV concludes the paper by highlighting the advantages of the presented numerical approach.

II. NUMERICAL MODELLING

This section introduces the basic governing equations and numerical procedures associated with the modelling of SCC flow.

A. Governing Equations

The self-compacting concrete is assumed to have the characteristics of a viscous non-Newtonian fluid, described by a bi-linear Bingham-type rheological model in which the fluid flow only initiates once the shear stress has reached a critical value called the yield stress τ_0 [13][14]. Thereafter, the shear stress varies linearly with the shear rate $\dot{\gamma}$, the slope being equal to the plastic viscosity η of the SCC mix. Ghanbari and Karihaloo [14] have shown recently how to predict the plastic viscosity η of self-compacting concretes with and without short steel fibres from the measured viscosity of the paste alone using micromechanical models in which the second phase aggregates are treated as rigid spheres and the short steel fibres as slender rigid bodies. They have also argued the yield stress τ_0 of SCC mixes is practically unchanged over a very large

Simulations of liquid impacts using a two-fluid parallel SPH model

G. Oger, P.-M. Guilcher, E. Jacquin
HydrOcean
Nantes, France

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

N. Grenier, D. Le Touzé
Laboratoire de Mécanique des Fluides
Ecole Centrale de Nantes
Nantes, France

***Abstract*— The fluid-structure interaction model within the parallel solver SPH-flow has been successfully applied to various single-phase sloshing impact problems (Oger et al., 2009).] Ecole Centrale Nantes and HyrdOcean have carried out the developments of this model, with support from GTT (Gaztransport & Technigaz). As a result of this partnership, SPH-flow has been recently extended, enabling the treatment of interactions between several fluids. In this paper, the theoretical model of SPH-flow for the two-phase formulation is described. Applications to liquid impacts are then presented, confirming the strong influence of the gas on flow evolution, and impact pressure peaks. These applications deal with the free fall of a bi-dimensional liquid patch through a gas, and the simulation of a breaking wave impacting a rigid wall involving an entrapped gas pocket.**

I. INTRODUCTION

The influence of the gas phase during a liquid impact event on a wall (such as a sloshing impact), appears to be crucial for a good estimation of the impact pressures. This has been confirmed experimentally by sloshing model tests when comparing statistical pressures obtained with gases of different densities within the tank [15]. When assumed as non condensable, the gas properties that significantly influence the liquid/gas interaction are its density and compressibility (hence the speed of sound, regardless of the equation of state considered).

As the membrane containment systems designer for LNG carriers, *GTT* (*Gaztransport & Technigaz)* continues to work on sloshing related R&D in order to obtain improved predictions of their design loads. Numerical simulation is considered as a complement to experiments, in order to gain further understanding on the physics of sloshing impacts. Such work enables parametric studies of impacts in ideal situations (simplified initial shape of the input wave, ideal physical properties of liquids and gases, and the simplified list of physical phenomena involved), that would not be possible by experimental analysis alone.

Special attention has been paid to the Smoothed Particle Hydrodynamics (SPH) method. The SPH method offers advantages over classical numerical methods when simulating sloshing type problems. No connectivity is required for the free surface, enabling the simulation of violent flows with possible fragmentation and interface reconnection. The Lagrangian formulation cancels the interface diffusion, resulting in a sharp definition of interfaces between gas, liquid and structure. Moreover, SPH can be applied to any continuum description, resulting in an ability to easily approach multi-physics. Therefore, the method can theoretically solve compressible multi-phase/structure interaction problems occurring during sloshing phenomenon.

In this paper, the theoretical aspects of the two-phase SPH developments are discussed. Applications to liquid impacts in the presence of a gas are then presented.

II. SPH THEORY FOR TWO-FLUID FLOWS

The developments performed for upgrading the SPH-flow software from a mono-fluid/structure parallel version to a bi-fluid/structure parallel version are described in this section.

A. Main characteristics of SPH-flow software

SPH-flow is a 2-D and 3-D parallel SPH solver developed by *Ecole Centrale Nantes* and *HydrOcean* [8,17,7,11,10]. It enables solving complex fluid and multi-physics problems through massive HPC resources. Complex geometries in free or forced motions can be modelled, with a variable space discretization (variable-h) solver for increased resolution simulations.

The solver was first developed for fluid flow simulations dedicated to complex, non-linear free surface problems. Within this context, the conservation laws for a compressible fluid are solved together with the Tait equation of state. Inspired by Finite Volume formalisms, Vila [24] proposed to rewrite the SPH formulation initially developed by Monaghan [16]. A Riemann solver in a Lagrangian framework was introduced, leading to increased stability properties of the scheme. Combined with the MUSCL (Monotone Upwind Centered Scheme for Conservation Law) extrapolation [22], this scheme provides very stable and low-diffusive results and can be written in the more general Arbitrary-Lagrange-Euler (ALE) formalism. Accuracy and convergence order of the scheme have been increased, leading to improved spatial derivative

IB-SPH simulations of wave-body interactions.

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

Elie Rivoalen
Laboratoire de Mécanique de Rouen
INSA de Rouen
Rouen, France
elie.rivoalen@insa-rouen.fr

***Abstract*—The present study deals with the use of the Immersed Boundary Method within the Smoothed Particle Hydrodynamics framework, for the investigation of hydrodynamics problems. The Navier-Stokes equations are solved by a parallel weakly compressible SPH code. By the use of moving least squares (MLS), the computation of forces between the boundary and the fluid particles has been significantly improved. As a matter of validation, the SPHERIC validation case 6 has been investigated.**

I. Introduction

Our goal is to model immersed, possibly moving, complex and rigid bodies, interacting with an oscillatory flow in an open channel. The free surface can't be accurately defined if particles are remeshed at every time step (like in RSPH [1] and in our previous paper [2]). So combining a non-conforming boundary model and an accurate, fully lagrangian, no-slip condition at this boundary seems to be a good compromise for future investigations. The numerical scheme and an improved Immersed Boundary technique applied to rigid bodies will be presented. Some validation results will also be given considering the SPHERIC Benchmark Case 6. Finally, preliminary results of applications to free surface flows and interaction with a rigid immersed body will be presented.

II. Numerical Scheme

A. SPH Model

In our model, the Navier-Stokes equations are written in the following way:

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

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

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

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

$$\boldsymbol{F}_i^v = \sum_j \frac{2\mu}{\rho_i \rho_j} \boldsymbol{u}_{ij} \frac{\boldsymbol{x}_{ij} \cdot \boldsymbol{\nabla} W_{ij}^*}{|\boldsymbol{x}_{ij}|^2} m_j \quad (5)$$

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

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

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

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

$\boldsymbol{\nabla} W_{ij}^*$ is the corrected kernel gradient at $\boldsymbol{x}_j$ centered in $\boldsymbol{x}_i$:

$$\boldsymbol{\nabla} W_{ij}^* = \frac{\boldsymbol{L}_i + \boldsymbol{L}_j}{2} \boldsymbol{\nabla} W_{ij} \quad (9)$$

$\boldsymbol{L}_i$ is the renormalization matrix at the location $\boldsymbol{x}_i$ [3]:

$$\boldsymbol{L}_i = \begin{pmatrix} \sum_j x_{ij} \nabla_x W_{ij} \frac{m_j}{\rho_j} & \sum_j x_{ij} \nabla_y W_{ij} \frac{m_j}{\rho_j} \\ \sum_j y_{ij} \nabla_x W_{ij} \frac{m_j}{\rho_j} & \sum_j y_{ij} \nabla_y W_{ij} \frac{m_j}{\rho_j} \end{pmatrix}^{-1} \quad (10)$$

The smoothing kernel W_{ij} is the *modified* Gaussian function given by Colagrossi and Landrini [4]. In (1), following the work by Ferrari *et al.* [5], a Rusanov flux has been added to enhance the stability of the scheme by reducing density fluctuations, which are often observed with weakly compressible models. The choice of introducing this flux instead of the well known Moving Least Sqaure (MLS) density re-initialization has been motivated by the lower numerical cost it provides and by the fact that, no parameter calibration (*i.e.* the number of time steps between two re-initialization) is required.
The viscous term (5) is written as proposed by Morris *et al.* [6]. This formulation is known to preserve the linear momentum exactly, but the angular momentum only approximately [7]. Using a gaussian kernel, this term takes the same form as an approximation of the laplacian operator using the Particle Strength Exchange (PSE) by Eldredge *et al.* [8].

B. Immersed Boundary Method

Boundaries of the computational domain are modeled using the technique of *ghost particles* to enforce no-slip boundary conditions. Since the use of ghost particles to model arbitrary shaped boundaries is not straightforward, the Immersed Boundary Method (IBM) has then been preferred to model complex geometries when immersed bodies have to be simulated.

Simulating free-surface channel flows through SPH

I. Federico[1], S. Marrone[2,3], A. Colagrossi[2,4], F. Aristodemo[1], P. Veltri[1]

[1]Department of Soil Defence, University of Calabria, Italy
[2]INSEAN, The Italian Ship Model Basin, Italy
[3]Department of Mechanics and Aeronautics, University of Rome - La Sapienza, Italy
[4]CESOS, Centre of Excellence for Ship and Ocean Structures, NTNU, Trondheim, Norway

***Abstract*—In this work, a study of 2D open-channel flows has been performed through a SPH solver. An in/out-flow algorithm has been developed to treat this class of hydraulic problems. In order to assign different upstream and downstream flow conditions, two new sets of particles (in/out-flow particles) have been introduced. These boundary particles affect the fluid particles but, on the opposite, are not affected by fluid particles.**

Two kinds of simulations have been carried out. First, a viscous laminar flow has been considered with Reynolds number of order $\mathcal{O}(10^2)$. A comparison between the in/out-flow boundary conditions and periodic conditions has been proposed. The simulations are carried on for a time interval long enough to reach a steady state condition. Three different spatial resolutions have been compared with analytical solution in order to heuristically check the convergence of the numerical scheme.

As a second test case we studied the hydraulic jump. This represents a strongly dissipative hydrodynamic problem. Different types of jumps, obtained varying the Froude number, are investigated with particular reference to the location of the jump and the velocity field. The model has been validated comparing the numerical water depths with the theoretical ones.

I. Introduction

The study of open-channel flows is a key topic in hydraulic river engineering. In SPH context the simulations of open-channel flows need suitable in/out-flow boundary conditions. It is well known that the enforcement of these kind of boundary conditions is not trivial for particle methods. Conversely, in lagrangian numerical approach, periodic conditions can be easily treated even if this method is useful just for academic problems. In most hydraulic applications, different upstream and downstream conditions are needed and, therefore, a more general boundary treatment has to be used. For this reason, in this work, a new algorithm for in/out-flow conditions is proposed.

Firstly, it is applied to viscous free-surface channel flow in laminar regime. The suitability of the in/out-flow algorithm is highlighted by the comparison of the obtained particles' distribution and velocity field against the ones resulting from the periodic boundary conditions. Furthermore, the numerical velocity profile is compared with the analytical Poiseuille solution.

Then the efficiency of the algorithm is tested through a hydraulic problem characterized by different upstream and downstream conditions. Under specific boundary conditions, this problem gives rise to a sharp discontinuity in the flow field. This phenomenon is known as hydraulic jump and is characterized by strong dissipative effects. Varying the flow Froude number, different types of jumps are obtained: undular, breaking undular and weak jump. The results are validated with the classical theory of hydraulic jump [1].

The governing equations of the flow evolution for the SPH model adopted are:

$$\begin{cases} \frac{D\rho_a}{Dt} = -\rho_a \sum_b (\mathbf{v}_b - \mathbf{v}_a) \cdot \nabla_a W(\mathbf{r}_{ab})\, dV_b \\ \rho_a \frac{D\mathbf{v}_a}{Dt} = -\sum_b (p_b + p_a) \nabla_a W(\mathbf{r}_{ab})\, dV_b + \rho_a \mathbf{f}_a + \\ \qquad\qquad + \mu \sum_b \pi_{ab} W(\mathbf{r}_{ab})\, dV_b \\ p_a = c_0^2 (\rho_a - \rho_0) \end{cases}$$

where $\pi_{ab} = \frac{8(\mathbf{v}_b - \mathbf{v}_a)\cdot \mathbf{r}_{ab}}{\|\mathbf{r}_{ab}\|^2}$, $\mathbf{r}_{ab} = -\mathbf{r}_{ba} = \mathbf{r}_a - \mathbf{r}_b$, $\mathbf{r}_a$ and $\mathbf{r}_b$ are the positions of the a-th and b-th particles.
The symbols ρ_a, p_a, $\mathbf{v}_a$ and dV_a denote the a-th particle density, pressure, velocity and volume; $\mathbf{f}$ is the body force field, ρ_0 is the density at the free surface, c_0 is the sound speed and μ is the dynamic viscosity. A renormalized Gaussian kernel function $W(\mathbf{r}_{ab})$ (for more details see *e.g.* [2]) has been adopted. The bottom channel is modeled by the solid boundary particles [3].

II. In/out-flow boundaries

The proposed algorithm for in/out-flow boundaries is here illustrated. In order to assign different upstream and downstream flow conditions two new sets of boundary particles have been defined. Therefore, including the solid boundary particles [3] which simulate the channel bottom, four sets of particles are used: fluid (f), solid boundary (s), inflow (i) and outflow (o) particles.

Similarly to the solid boundary particles, the in/out-flow particles affect the fluid particles but not vice versa and cover a region which is at least as wide as the kernel radius.

Fig. 1a shows an initial sketch of the computational domain: different colours are associated to different sets of particles. The flow extends along the x-axis and it is limited by an *inlet* and an *outlet* boundary. The algorithm procedure is characterized by the use of an *inflow* and an *outflow* threshold. The particles that cross these thresholds change the set they belongs to (see fig. 1b). For reasons of computational efficiency, the particles belonging to each set have been listed subsequently in one only array. Through the use of in/out-flow particles,

Improved time scheme integration approach for dealing with semi analytical boundary conditions in *SPARTACUS2D*

Martin Ferrand
School of Mech., Aero. & Civil Engineering
University of Manchester
Manchester, United Kingdom
École des Ponts Paritech
Paris, France
martin.ferrand@ponts.org

Dominique Laurence[1], Benedict D. Rogers[1], Damien Violeau[2]
[1]School of Mech., Aero. & Civil Engineering
University of Manchester
Manchester, United Kingdom
[2]Saint-Venant Laboratory for Hydraulics
Université Paris-Est
Chatou, France

***Abstract*—The present paper aims to provide robust and physical boundary conditions in the most basic *SPH* formulation, that is without renormalization of the kernel *gradient* or Riemann solver approach. The present method is analogous to Kulasegaram *et. al*'s approach [1] in the sense that it is based on a geometrical parameter measuring the missing area in the kernel support when a particle is in the vicinity of a solid boundary, but improve accuracy for linear fields, such as a hydrostatic pressure. This geometrical term is computed with a governing equation, and the integration in time of the continuity equation is modified in order to ensure density conservation during long-time simulations. Results are presented for (*i*) a channel flow where hydrostatic pressure is better reproduced and density is sustained, (*ii*) still water and dam break in a basin with a wedge, (*iii*) the *SPHERIC* benchmark of a moving square solid inside a rectangular tank.**

I. Introduction

Dealing with wall boundary conditions is one of the most challenging parts of the *SPH* method and many different approaches have been developed by authors including Kulasegaram *et al.* [1], Oger *et al.* [2], Di Monaco *et al.* [3], Monaghan and Kajtar [4], Marongiu *et al.* [5] and De Leffe *et al.* [6]. Accurate boundary conditions are essential since in many applications the target values are precise loading on walls (forces on floating bodies or shoreline structures, tank walls, wind-wave exchanges, fluid-structure interactions in power-plants etc.), and is an obvious prerequisite of the current research project also aiming at improving turbulence modelling near walls.

The present work is based on [1] which consists of renormalizing the density field near a solid wall with respect to the missing kernel support area. This methodology, combined with the Lagrangian formalism, defines intrinsic *gradient* and *divergence* operators which are variationally consistent and ensure conservation properties.

The time integration scheme used for the continuity equation requires particular attention, and as already mentioned by Vila [7], we prove there is no point in using a dependence in time of the particles' density if no kernel gradient corrections are added. Thus, by using a near-boundary kernel-corrected version of the time integration scheme proposed by Vila, we are able to simulate long-time simulations ideally suited for turbulent channel flow with accurate boundary conditions.

As mentioned by De Leffe *et al.* [6], the method of Kulasegaram *et al.* defines an inaccurate gradient operator which provides non-consistent behaviour. Herein we introduce corrections of differential operators analogous to Di Monaco *et al.* [3] and De Leffe *et al.* [6] for slightly compressible viscous Newtonian flows, but all boundary terms issued from the continuous approximation are given by surface summations which only use information from a mesh file of the boundary. In order to compute the kernel correction, Feldman and Bonet [8] use an analytical value which is computationally expensive whereas Kulasegaram *et al.* [1] and De Leffe *et al.* [6] use polynomial approximations which can be difficult to define for complex geometries. We propose here to compute the renormalization term of the kernel support near a solid with a time integration scheme, thereby more easily accounting for any shape of boundary.

All the present developments have been numerically tested using *SPARTACUS2D*, a *FORTRAN* code initially developed by *EDF R&D*.

II. State-of-the art

A. Original formulation and operator consistency

The slightly compressible Newtonian fluid is modelled by a set of particles denoted by the subscripts $(.)_a$ and $(.)_b$ in a domain Ω. Each particle a possesses information such as its mass m_a (assumed constant), its position $\mathbf{r}_a$, its velocity $\mathbf{u}_a$ which is the Lagrangian derivative of the position, its density ρ_a, its volume $V_a = \frac{m_a}{\rho_a}$, and its pressure p_a. The spacial discretization is based on a kernel function w with a compact support. Let Ω_a be the support of the kernel function centred in $\mathbf{r}_a$. We generally denote by the subscripts f_{ab} the difference of the quantity f between the positions a and b. For

SPH Shallow Water Equation Solver for real flooding simulation

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

B.D. Rogers & P.K. Stansby
School of Mech., Aerosp. and Civil Engin.,
The University of Manchester
Manchester, M60 1QD, UK
e-mails: benedict.rogers@manchester.ac.uk,
p.k.stansby@manchester.ac.uk

***Abstract*—In this paper an SPH numerical model for the Shallow Water Equations (SWEs) is to be presented with the aim of simulating flood inundation events due to rapidly varying flow such as dam breaks, tsunamis, levee breaches etc. The SPH-SWEs method has been limited so far by the variable kernel size or smoothing length being inversely proportional to water depth causing poor resolution at small depths; this is addressed in this work by introducing a particle splitting procedure which conserves mass and momentum by varying the smoothing length, velocity and acceleration of each refined particle. Moveover, in order to make the simulation of real events feasible, an algorithm to impose open boundary conditions, based on a simplified version of the characteristic method, is developed. Both supercritical and subcritical inflow and outflow boundary conditions can be handled. The code is tested against a number of reference solutions such as transcritical flow over a hump in order to validate it and afterwards it has been applied in a flood hazard analysis with the aim to show the capability of the simulation method to reproduce real-scale events.**

I. Introduction

The numerical simulation of flooding events over initially dry area due to dam breaks or tsunamis is one of the most important tools relevant in flooding risk analysis. To date, this kind of study has been conducted using the Shallow Water approximation and using classical Eulerian numerical methods. Meshless models can offer a great advantage because the calculations are conduted just where the fluid is present and large dry areas can be easily simulated. Recently various authors extended the SPH technique to Shallow Water Equations (SWEs) obtaining promising results: Ata and Soulaïmani [**?**] proposed a new formulation for the stabilization term in order to avoid the artificial viscosity, Rodriguez-Paz and Bonet [12] introduced an SPH-SWE formulation derived from variational approach, where the variable smothing length is introduced consistently; and de Leffe et al. [3] presented a SWEs scheme based on the approach proposed by Vila [16]. Some limitations that affects these models are addressed in this paper: the problem of poor resolution at small depths is solved by a conservative particle splitting procedure [15], the closed boundary simulation is successfully handled by the Modified Virtual Boundary Method [6], [14] and implementation of source terms over real bathymetries can be described by the bottom particle approach [14]. Finally, open boundaries are introduced in the SPH-SWEs model in order to remove the last limitation that prevents the use of SPH-SWEs models for real flooding simulations.
This paper is organized as follows: in Section II the main aspect of the SPH-SWEs numerical models based on the variational approach are briefly recalled, in Section III the new methodogy for the open boundary is presented. In section IV the SPH-SWEs model is validated against some reference solutions, and finally in Section V the simulation of a real flood-hazard analysis is made in order to show the capability of the SPH-SWEs scheme to simulate flood events over real bathymetries.

II. SPH Shallow Water Equation solver

The SWEs are formally identical to the Euler equations if we re-define the density ρ as the amount of fluid per unit of area in a 2-D domain; given this new definition of ρ we can connect it to the depth of water d with: $\rho = \rho_w d$, where ρ_w denotes the constant 3D (conventional) density. The density ρ_i of a particle i can vary enormously during a simulation; therefore an SPH scheme with variable smoothing length h in time and space is used in order to keep the number of neighbours particles roughly constant during the processes of water inundation and retreat. Using these definitions and the Lagrangian derivatives the SWEs can be written as follow:

$$\begin{aligned} \frac{d\rho}{dt} &= -\rho \nabla \cdot \mathbf{v} \\ \frac{d\mathbf{v}}{dt} &= -\frac{\rho}{\rho_w} \nabla \rho + g \left(\nabla b + \mathbf{S}_f \right) \end{aligned} \tag{1}$$

where $\mathbf{v}$ is the horizontal velocity vector, b is the bottom elevation, g is the acceleration due to gravity and $\mathbf{S}_f$ is the bed friction source term.
The particles position and the velocities are integrated in time by means of a leap-frog scheme and a Lax-Friedrichs term is used in order to keep the solution stable even in presence of shock waves, moreover a MUSCL reconstruction is applied to increase the accuracy of the SPH interpolation [15]. The closed boundaries (walls) are simulated using the Modified Virtual Boundary Particle method, described in [14].

Violent Fluid-structure impacts solved through a δ-SPH model

Marrone S.
INSEAN, DMA
s.marrone@insean.it

Antuono M.
INSEAN

Colagrossi A., Colicchio G.
INSEAN, CESOS

Le Touzé D.
ECN

Graziani G.
DMA

INSEAN, The Italian Ship Model Basin, Rome, Italy
DMA, Department of Mechanics and Aeronautics University of Rome "Sapienza", Rome, Italy
CESOS, Centre of Excellence for Ship and Ocean Structures, NTNU, Trondheim, Norway
ECN, Fluid Mechanics Lab, Ecole Central de Nantes/CNRS Nantes, France

***Abstract*—The diffusive weakly-compressible SPH scheme described in Antuono *et al.*[1] (hereinafter denoted as δ-SPH model) has been challenged in the analysis of impacts against structures generated by dam-break flows. Specifically, the 3D SPHERIC benchmarks number 1 and 2 have been studied as well as a novel 2D test case characterized by complex solid boundary profiles.**

In all the test cases the solid boundaries have been modelled through the boundary particle technique described in Colagrossi *et al.*[2] and a free-slip condition has been imposed along the solid profiles. For the SPHERIC benchmarks, the numerical results have been validated through comparison with the experimental data already available in the literature. Conversely, the new test case has been compared with the numerical results obtained through a Navier-Stokes Level-Set scheme. Finally, a brief insight on viscous effects has been provided for the latter test case.

All test cases proved that the δ-SPH scheme with the boundary particle technique is an accurate and robust solver for the prediction of the global fluid evolution and the local loads against structures.

I. Introduction

In many physical phenomena concerning violent water/structure interactions, the evaluation of global and local loads is of fundamental importance. For example, this is a key issue for coastal or ocean structures under extreme sea conditions (see *e.g.* [19], [17]) and for problems of water on deck (see *e.g.* [10]).

Since impact events are characterized by strong dynamics and large fluid deformations, their analysis is very complex: analytical results are only available for simple geometries while the experimental data are of difficult interpretation because of interaction of several physical features.

The objective of the present work is, therefore, to prove that the Smoothed Particle Hydrodynamics model with diffusive terms proposed by Antuono et al. [1] is accurate and well suited to simulate the dynamics of strong impacts against structures. A set of test cases related to impact flows generated by dam breaks has been investigated. In all the simulations the solid boundary has been modeled through the boundary particle technique described in Colagrossi *et al.* [5]. This technique was proven to give reliable results when comparing to experimental data or to other solutions (see [15]).

The first and second test cases are the 3D SPHERIC benchmarks number 1 and 2 while as a third simulation a novel 2D test case characterized by complex solid boundary profiles has been considered. Due to its geometrical configuration, the latter test case leads to violent fluid-structure interactions and pressure loads and, therefore, is suitable to become a new benchmark inside the SPHeric community.

The simulations have been performed by assuming the fluid to be inviscid and, therefore, by imposing a free slip condition along the solid profiles. Further, to give a brief insight on viscous effects, the last test case has been run again with Reynolds numbers equal to 1000 and 10000 and using a no-slip condition along the solid boundary.

The paper is organized as follows: we introduce the diffusive SPH model and the solid boundary treatment in the first section while in the second section we focus on the test cases. Each test case has a dedicated sub-section in which comparisons with theory, experiments or other numerical results are described in depth.

II. The δ-SPH scheme

In the present work we adopt the SPH scheme proposed Antuono et al. [1] in which 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. This scheme reads:

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

Propagation of gravity wave-packets through a δ-SPH method

Antuono M.
INSEAN
m.antuono@insean.it

Marrone S.
INSEAN, DMA

Colagrossi A., Lugni C.
INSEAN, CESOS

INSEAN, The Italian Ship Model Basin, Rome, Italy
DMA, Department of Mechanics and Aeronautics University of Rome "Sapienza", Rome, Italy
CESOS, Centre of Excellence for Ship and Ocean Structures, NTNU, Trondheim, Norway

***Abstract*—In the present work we study the capability of the diffusive weakly-compressible SPH scheme described in Antuono *et al.* [1] (hereinafter denoted as δ-SPH) to reproduce the propagation of 2D gravity waves generated by a wavemaker into a 2D wave basin. In the first part we consider regular waves with height-to-depth ratios spreading to deep to shallow water and steepness values chosen to emphasize nonlinear behavior in wave propagation. The results of the Mixed-Eulerian-Lagrangian Boundary Element Method (hereinafter BEM-MEL) developed in [3] are used for comparison. This comparison allows inspecting the accuracy of the δ-SPH to approximate inviscid fluids. In the second part we study the propagation of a wave packet. The experimental data of Dommermuth *et al.* [4] have been considered for validating the capability of the SPH solver to follow the evolution of a wave train towards the focusing. The results clearly show a good match between δ-SPH, BEM-MEL and experiments. The influence of the weakly-compressibility assumption on the results is inspected and a convergence analysis is provided in order to understand the minimal spatial resolution needed to get a good representation of gravity waves.**

INTRODUCTION

In the last years the use of particle methods to simulate complex flow has been largely increased (see e.g. [5], [6]) As a primary feature those methods do not need a structured topological connection (grids) between the computational nodes. These nodes are treated as fluid particles followed during their motion, while their physical properties evolve in time according to the governing equations. One of the main advantages of such methods consists in the capability to deal with complex free-surface flows without an explicit enforcement of the dynamic condition along it (for details see [7]).

In this context, the SPH scheme has widely proved to be an accurate and efficient solver to simulate violent flows with strong deformations of the free surface (see for example [8], [9]). Notwithstanding that, in the SPH literature only a few works deal with the modeling of gravity waves (see for example [10]). Then, in the present work we study the capability of the weakly-compressible SPH scheme with diffusive terms described in Antuono *et al.* [1] (hereinafter denoted by δ-SPH) to correctly reproduce the propagation of 2D gravity waves generated by a wave maker into a 2D wave basin. This also stands as a further validation for the numerical technique used to model the solid boundary and the wave maker first described in [2]. In the first part of the paper regular waves have been considered and a large number of simulations have been performed for steepness and the height-to-depth ratio spreading from deep to shallow water. Wave heights have been chosen in order to emphasize nonlinear features and to avoid wave breaking. This allows a comparison between the SPH results and the predictions of a Mixed-Eulerian-Lagrangian Boundary Element Method (hereinafter BEM-MEL) [3]. The BEM-MEL solver, based on the potential theory (that is, inviscid and irrotational flows), is the most appropriate scheme to describe the propagation of gravity waves (see for example [11], [12], [13], [14], [15], [16]). With respect to this, the present work aims to understand the capability of the SPH scheme (which is generally implemented by using an artificial viscosity) to approximate inviscid fluids. In the second part of the paper we focus on the propagation of wave packets. A wave packet is given by a spectrum of waves whose phases are adjusted so that their superposition gives rise to a rapid increase in wave amplitude at a prescribed distance (called focusing point) away from the wave maker. Following Dommermuth *et al.* [4], the wave packet is used to get a breaking wave at the focusing point and the numerical results are then compared with the experimental measures provided in that work.

Finally, the influence of the weakly-compressibility assumption on the results is inspected and a convergence analysis is provided in order to understand the minimal spatial resolution needed to get a good representation of gravity waves.

The paper is organized as follows: first, we introduce the δ-SPH scheme adopted here and the solid boundary treatment and, then, we deal with the test cases and the discussion of the results.

I. THE δ-SPH SCHEME

In the present work we adopt the SPH scheme proposed Antuono et al. [1] in which a proper artificial diffusive term is used into the continuity equation in order to remove the spurious numerical high-frequency oscillations in the pressure

The influence of the truncated kernel to free-surface predictions with ISPH and a new solution

R. Xu and P. K. Stansby
School of Mechanical, Aerospace, and Civil Engineering
University of Manchester
Manchester, UK

***Abstract*—Projection-based incompressible SPH is a meshless method, providing accurate prediction to fluid fields comparing with Weakly Compressible SPH (WCSPH). A new divergence-free ISPH approach is used here to simulate the free-surface flows. This new ISPH approach maintains accuracy and stability without increasing computational cost by slightly shifting particles away from streamlines while correcting their hydrodynamic characteristics. This avoids the highly distorted particle spacings which causes instability. The extra artificial diffusion is introduced around the free surface, and effectively reduces the noise caused by truncated-kernel error without contaminating predictions for the velocity and pressure. A new Laplacian operator proposed by Schwaiger is applied in the code, which highly improves the accuracy of the simulation. The thorough quantitative validation has been first-time conducted for free-surface predictions in the context of SPH method. It is proved through test cases that no matter for the free-surface prediction, or for the velocity and pressure field, this ISPH method can provide accurate predictions. Good agreements are obtained for the studied cases.**

I. Introduction

Traditional Eulerian methods have some difficulties in describing the free surface, while Lagrangian methods possess natural ability to describe the details of complicated phenomena. It is much easier for Lagrangian method to include complicated integration. The traditional WCSPH method has predicted some highly transient flows quite well [1]–[3], pressure fields were noisy and the method highly dissipative [4]. Therefore, the application of ISPH method in free-surface prediction is desirable.

A highly accurate, incompressible, mesh-free, noise-free method for arbitrary free-surface flows is attractive for many problems in engineering which involve both fluid-structure interaction and multi-phase fluid simulations. In 2009, a new ISPH approach with improved stability and efficiency has been proposed in [5]. The objective of this paper is to validate the new ISPH approach with free-surface predictions. However, it is shown by Schwaiger that larger numerical error could appear around the free surface even with the improved Laplacian operator in [6]. And this numerical error will be represented as noisy free surfaces in simulations here. In this paper, an artificial damping viscosity calculated from the maximum local Peclet number has been introduced for free-surface particles and particles around free surface. Rigorous validation is desirable against analytical or highly accurate solutions as undertaken for internal flows. The following phenomena are covered in different simulations separately:

1) free-surface evolution with very high curvature,
2) wave propagation.

The dam-break flow has received considerable attention but has not been fully exploited. For the wet-bed case we use the highly accurate solution for flow acceleration at zero time, with a singularity in the free surface at the lower gradient discontinuity [7]. With an analytic initial condition a highly accurate high-order boundary integral method for potential flow [8] is available for small times showing jet-like mushroom behavior [7]. This is preferable to direct experimental comparison, which shows similar phenomena, since the plate removal time is of the same order as the time scale for the formation of the initial flow structures. Finally non-dissipative wave propagation should be demonstrated. Regular waves are generated by a piston-type paddle for several periods and surface profiles, velocities and pressures are compared against accurate stream-function theory [9]. This is undertaken for small and moderate waves for which linear wavemaker theory is realistic.

In this way an algorithm for arbitrary incompressible free-surface motion will be established. This may be generalized to 3-D motion with complex boundaries through massively parallel processing but this is not considered here. In this paper, the improved Laplacian operator [6] will be introduced in §2. The truncated kernel error and one of its solutions are introduced in §3. Two test cases, 2-D dam break and the regular-wave propagation, are defined in §4. The results and discussion are presented in §5. The conclusion is drawn in §6.

II. Improved Laplacian operators

One of the main error in incompressible SPH is coming from the inaccuracy of viscous term and Laplacian operators. It has been fully discussed in [6] that the traditional Laplacian (or viscosity) operator could not provide accurate interpolation around the free surface, and Schwaiger proposed a new

Theoretical analysis of SPH in simulating free-surface viscous flows

Colagrossi A. INSEAN, CESOS
Antuono M. INSEAN
Souto-Iglesias A. ETSIN
Le Touzé D. ECN
Izaguirre-Alza P. ETSIN

INSEAN, The Italian Ship Model Basin, Rome, Italy
CESOS, Centre of Excellence for Ship and Ocean Structures, NTNU, Trondheim, Norway
ETSIN, Naval Architecture Department,Technical University of Madrid (UPM). 28040 Madrid, Spain
ECN, Fluid Mechanics Lab, Ecole Central de Nantes/CNRS Nantes, France

***Abstract*—A theoretical analysis on the performance, close to a free surface, of the most used SPH formulations for Newtonian viscous terms is carried out in this paper. After an introduction of the SPH formalism, the SPH expressions for the viscous term in the momentum equation are analyzed in their continuous form. Using a Taylor expansion, a reformulation of those expressions is undertaken which allows to characterize the behavior of the viscous term close to the free surface. Under specific flow conditions, we show that the viscous term close to the free surface is singular when the spatial resolution is increased. This problem is in essence related to the incompleteness of the kernel function close to the free surface and appears for all the formulations considered. In order to assess the impact of such singular behavior, an analysis of the global energy dissipation is carried out, which shows that such a free-surface singularity vanishes when the integral quantities are considered. Notwithstanding that, not all the SPH viscous formulas allow the correct evaluation of the energy dissipation rate and, consequently, they may lead to an inaccurate modelling of viscous free-surface flows.**

I. Introduction

Smoothed Particle Hydrodynamics (SPH) is a method devised to obtain approximate numerical solutions of the Navier-Stokes equations. The SPH discretization of these equations does not lie on a mesh-based decomposition of the fluid domain, but it relies on the transport of the fluid properties by a set of particles.

Due to its underlying structure, SPH was initially thought to solve astrophysical problems [1], [2] and a few years later, its application range was extended to gravity related incompressible inviscid free-surface flows [3]. In the first formulations, the incompressibility was achieved by choosing a stiff equation of state and an adequate numerical sound speed.

In order to extend the method to viscous flows, and due to several disadvantages that arise from the direct SPH interpolation of the second derivatives [4], some numerical viscous terms have been developed. The most widespread is surely the Monaghan & Gingold one [5]. This is main due to the fact that this formula preserves both linear and angular momenta. Another famous formulation is the one given by Morris *et al.* [6] which, although does not conserve exactly angular momentum, it is the straightforward discretization of the Laplacian using the general differentiation formulas of Español et al [7]. To recover the conservation properties, Watkins *et al.* [8], Bonet & Lok [9] and Flebbe *et al.* [10] obtained formulas that preserve both linear and angular momentum. Anyway they were extremely expensive from a computational point of view because they need to evaluate two times the particles' interactions. Due to this drawback these formulations are rarely used in practice. A further important contribution to the analysis of viscosity in SPH using direct differentiation to obtain second derivatives has been given by Takeda *et al.* [11]. Though their formula has not been used much for free surface flows, and though it does not conserve angular momentum, it allows to take into account the effects related to the compressibility of the fluid.

This paper is organized as follows: the first part (section II), is dedicated to present the flow field continuum equations, emphasizing the structure of the Newtonian stress tensor and the free surface boundary conditions for viscous flows. In section III, the SPH formalism and notation is presented. Section IV is the core of the paper, presenting an analysis of the behavior of the viscous term near the free surface using the Monaghan & Gingold formula [5] and the Morris *et al.* [6] formulation both in their continuous form. In section V, we study the global energy dissipation corresponding to those viscous formulas. Finally two test cases aimed at demonstrating the validity of the conclusions obtained from the continuous analysis are presented.

II. Governing equations

A. Field equations

The compressible Navier-Stokes equations for a barotropic fluid in Lagrangian formalism are the continuum model of the flow:

$$\begin{cases} \dfrac{D\rho}{Dt} + \rho \nabla \cdot \boldsymbol{u} = 0\,, \quad \dfrac{D\boldsymbol{u}}{Dt} = \boldsymbol{g} + \dfrac{\nabla \cdot \mathbb{T}}{\rho}\,, \\ p = c_0^2(\rho - \rho_0)\,, \end{cases} \tag{II.1}$$

In these equations, ρ is the fluid density, ρ_0 is the reference density, p is the pressure, c_0 is the sound speed (assumed constant) and $\boldsymbol{g}$ is a generic external volumetric force field.

Three SPH Novel Benchmark Test Cases for free surface flows

E. Botia-Vera
A. Souto-Iglesias
Naval Architecture Dept.(ETSIN)
Technical University of Madrid (UPM)
Madrid, Spain
elkinmauricio.botia,antonio.souto@upm.es

G. Bulian
Dept. Naval Architecture, Ocean, Env. Eng.
University of Trieste
Trieste, Italy
gbulian@univ.trieste.it

L. Lobovský
Dept. Mech. & Biomed. Eng.
National University of Ireland
Galway, Ireland
libor.lobovsky@nuigalway.ie

***Abstract*—Benchmark Test Cases have been used by SPHERIC interest group members for the validation of SPH models and their corresponding computer implementations. Since the use of SPHERIC benchmark test cases as validation reference for SPH implementations has slightly declined in the most recent editions, we think it might be interesting to document three novel test cases with the aim of enriching the database with complementary validation data. The first proposed test case is a wave impact problem in a rectangular tank. The time history of the motion of the tank and the pressure of the first instances of lateral and roof impacts for both water and oil are provided. An analysis of the two-dimensionality and repeatability of the pressure peaks is provided. The second proposed test case treats the coupling of the angular motion of a sloshing tank and a single degree of freedom structural system. Finally, the third proposed test case, is a canonical fluid structure interaction problem consisting in the interaction between a free surface sloshing flow and an elastic body. As both SPH practitioners and experimentalists, regardless of the discussion provided in this paper, we are committed to improving these test cases for future use. We hope to increase our experimental skills and capabilities not only in light of experience from our own simulations but mainly by receiving a feedback from the SPH community.**

I. Introduction

Benchmark Test Cases have been used by the SPH community for the validation of SPH models and their corresponding computer implementation. One of the aims of the SPHERIC Workshops, as stated in the preface of the 2008 SPHERIC Workshop proceedings book, is to define and run benchmark test cases. In 2008 Workshop, only two papers made use of the SPHERIC benchmark test cases in order to validate computations. In 2009, that figure was the same. It may be therefore interesting to provide some new test cases that could serve as basis for validation procedures as well as providing room for some competitiveness between the different codes. There is a specific space in the SPHERIC web site from where the information corresponding to those cases can be downloaded as well as an application form for proposing new test cases. Although it is possible to use such a procedure through the SPHERIC web site directly, we think it would be interesting for the SPH community to discuss the newly proposed benchmark cases in the open forum of the 2010 Workshop. This is the reason why we think it can be adequate to organize those materials as a proper paper.

One of the main interests of the authors in the past has been related with free surface flows [1] as well as with the interaction between free surface flows and solid mechanics, either in the field of ship motions [2] or in the field of solids deformations and fluid structure interactions [3]. The three new proposed test cases spring naturally from those interests since they incorporate a wave impact problem in a sloshing flow in a rectangular tank, the coupling of the angular motion of a sloshing tank and a single degree of freedom structural system and the interaction between a free surface sloshing flow and an elastic body.

The cases are canonical in the sense that we have tried to simplify them as much as possible, aiming at making them useful for validating the corresponding SPH computational models. Such simplicity is related to their two-dimensionality which is in-depth discussed for the first proposed benchmark. The three proposed cases incorporate, we believe, all the necessary information to implement them in SPH codes. They are presented and discussed in this paper but detailed information necessary for the implementation and posterior validation assessments is available from the following link: http://canal.etsin.upm.es/ftp/SPHERIC_BENCHMARKS/

II. Wave Impact problem

A. General

The first test case focuses on a wave impact problem, by providing time histories of the pressure recorded at specific locations, together with the corresponding roll angle history of the periodic angular motion of a sloshing tank. This test case is an improved version of the test presented in [1] which focused on lateral impact problems in a rectangular tank and that has been already taken as reference data for the validation of an ISPH code by Khayyer et al. [4]. In the present paper, significant improvements and contributions with respect to reference [1] are presented. First, the test case focus on what is expected to be the most deterministic impact event, which is the first pressure peak, for which a repeatability analysis is provided. Second, the influence of liquid viscosity

Lyapunov Stability Analysis of semi-discretized SPH

R Vignjevic, S Powell
Department of Structures, Impacts and Machine Dynamics
School of Engineering, Cranfield University, UK
v.rade@cranfield.ac.uk

***Abstract* - It is well known that Standard SPH suffers from the "tensile instability". Previous analyses have been of the von-Neumann type. This work will present an analysis of Eulerian SPH in 1D using Lyapunov's method. It will be seen that this method can reproduce past results and under differing assumptions a new instability criterion is established.**

I. INTRODUCTION

A well documented flaw in the basic formulation of SPH is an instability due to the nature of the space discretization. Commonly this manifests as an unphysical clumping of particles under tension which can in turn result in purely numerical fracture. Many papers propose solutions to this problem but fewer [1-3] have a performed a rigorous stability analysis of SPH and all these have been von-Neumann analyses.

In a von-Neumann type stability analysis [4] the equilibrium state is perturbed with a displacement field assumed to be of the form.

$$u_j^n = \xi^n e^{-ikx_j}$$

Here n refers to the time step, j is the particle index and $i = \sqrt{-1}$. If the amplification factor $|\xi| > 1$ then the perturbation will grow and the solution is unstable. In some special cases including but not limited to linear PDEs with constant coefficients von-Neumann stability analysis provides necessary and sufficient conditions for stability.

The first papers to analyse the stability of SPH were [1; 3] the analysis is restricted to equally spaced particles in 1D with Eulerian kernels. Swegle *et al* [1] derive a sufficient condition for SPH to be unstable; $W''\sigma > 0$. In tension ($\sigma > 0$) if the particles are spaced such that the neighbouring particles' kernel's second derivative is positive the position is unstable.

Belytchko *et al* [2] extended there analysis to SPH with Lagrangian kernels (Total Lagrangian SPH), non-collocational SPH and to a two dimensional case. Three distinct modes of instability are identified; Tensile instability as identified by Swegle, instability due to the growth of zero-energy modes [5] and the instability present in the original PDEs. Total Lagrangian SPH does not suffer from tensile instability. Neither does non-collocational SPH in 1D, but the authors find that in 2D stability depends on the careful placement of the stress or slave nodes. Non-collocational SPH eliminates zero-energy modes and the authors state that a combination TL-SPH with stress point closely mirrors the instability properties of the PDE.

Lyapunov's direct method allows one to analyse the stability of equilibria of dynamical systems, see [6; 7]. The analysis is performed directly on the equations of motion, hence direct method, without any explicit solutions being required. Lyapunov's method can be applied to systems which evolve in discreet time steps as well as well as continuous time with little modification. Before applying Lyapunov's method to SPH there follows a brief list of definitions all are modified from definitions given in [6; 7]

A. *Lyapunov Stability analysis*

This section will present a minimum necessary background and terminology for those unfamiliar with Lyapunov's method. Lyapunov's method allows one to examine the stability properties of dynamical systems. The definition of a dynamical system is very broad but includes ordinary differential and difference equations. To illustrate consider a system of ordinary differential equations

$$\dot{\mathbf{x}} = f\left(\mathbf{x}(t), t\right)$$
$$\mathbf{x}(0) = \mathbf{x}_0$$

The vector $\mathbf{x}$ represents a point in phase space which traces a curve parameterised by (the time) *t.* This curve originating at the point $\mathbf{x}_0$ is called a motion of the system, by varying the initial condition a family of motions is generated. This family of motions together with the phase space, the set of admissible initial conditions and the time parameter define a dynamical system[7]. If $\mathbf{x}(t) = \mathbf{x}_e$ for all $t \geq 0$ then $\mathbf{x}_e$ is an equilibrium point. A change of coordinates can always be performed to bring the equilibrium to the origin and therefore in the remainder of this work we will assume $\mathbf{x}_e = \mathbf{0}$.

Lyapunov's method requires the definition of auxiliary functions and from the behaviour of these functions the stability properties of a system can be inferred. The main weakness of Lyapunov's method is that no general procedure exists for generating these functions. Lyapunov's definition of stability is a rigourous statement of the idea that a motion of a dynamical system which begins 'near' enough to a stable equilibrium will stay 'close' to the equilibrium for all time. Conversely an unstable equilibrium will have motions that begin arbitrarily close but nevertheless diverge unboundedly. Formally the definition of stability is:

WSPH AND ISPH CALCULATIONS OF A COUNTER-ROTATING VORTEX DIPOLE.

L. M. González, J.M. Sánchez, F. Macià, D. Duque, J. Gómez-Goñi
Naval Arquitecture School
Universidad Politécnica de Madrid
Madrid, Spain
leo.gonzalez@upm.es

M.A. Rodríguez-Perez
Cellular Materials Group (CellMat), Condensed Matter Physics Department
Universidad de Valladolid
Valladolid, Spain

***Abstract*—Viscosity and vorticity are magnitudes playing an important role in many engineering physical phenomena such as: boundary layer separation, transition flows, shear flows, etc., demonstrating the importance of the vortical viscous flows commonly used among the SPH community.**

The simulation presented here, describes the physics of a pair of counter-rotating vortices in which the strain field felt by each vortex is due to the other one. Different from the evolution of a single isolated vortex, in this case each vortex is subjected to an external stationary strain field generated by the other, making the streamlines deform elliptically. To avoid the boundary influence, a large computational domain has been used ensuring insignificant effect of the boundary conditions on the solution. The performance of the most commonly used viscous models in simulating laminar flows, Takeda's(TVT), Morris'(MVT) and Monaghan-Cleary's(MCGVT) has been discussed comparing their results. These viscous models have been used under two different compressibility hypotheses.

Two cases have been numerically analyzed in this presentation. In the first case, a 2D system of two counter-rotating Lamb-Oseen vortices is considered. At first, the system goes through a rapid relaxation process in which both vortices equilibrate each other. This quasi-steady state is obtained after the relaxation phase is advected at a constant speed and slowly evolves owing to viscous diffusion. The results of the different Lamb-Oseen numerical solutions have been validated with good agreement by comparison with the numerical results of a finite element code (ADFC) solution. A second case, somewhat more complex than the previous one, is a 3D Batchelor vortex dipole obtained by adding an axial flow to the system of the first case. The Batchelor vortex model considered here is a classical option normally used to model the structure of trailing vortices in the far-wake of an aircraft.

I. Introduction and Motivation

Modeling low Reynolds number vortical viscous flows does not present excessive difficulties for industrial focused CFD methods like finite element method (FEM). Nevertheless, these methods encounter difficulties when dealing with problems such as highly distorted free surface flows, where SPH is probably the first option. Laminar solutions of the incompressible Navier-Stokes and continuity equations, describing the evolution of vortex laminar systems, have been efficiently obtained by both weakly compressible SPH (WSPH) and incompressible SPH (ISPH) methods. Several examples of simulation of isolated vortices [1]–[3] and periodic Taylor-Green flows [4], [5] can be found in the literature. The motivation for the present study is to extend the SPH method to model the interaction of the counter-rotating vortices created at the far wake of an aircraft, an interesting aeronautical application. Aircraft in flight leave behind large-scale swirling flows (vortices), which can represent a significant hazard to following aircraft, and therefore are of great importance for practical applications concerning safety and capacity of air transport, see [6].

Trailing vortices are a natural byproduct of airplanes, and other vehicles, with finite-span lifting wings. Lift is generated by pressure differences on the upper and lower surfaces of the wings. These pressure differences lead to the creation of vortices that trail behind the airplane and persist well downstream of the vehicle. The vortices (sometimes referred to as wake vortices, or wake turbulence) are characterized by a rotational flow field, analogous to a set of tornados turned sideways with their cores aligned in the flight-path direction. The circulation strength of the vortices is proportional to the total lift (which supports the airplane weight), and inversely proportional to the vortex span (which depends on the geometric wing span and the distribution of lift along the wings) and the airplane speed. Practical interest in trailing vortices is motivated by their potential impact to following airplanes that might encounter them. As an airplane flies into the vortices of another aircraft it can experience a significant upset. This can result in the rapid rolling of the airplane due to the rotational flow, and/or a drop in the airplanes altitude. Depending on the severity of the upset and the proximity to the ground, a vortex encounter can be a safety hazard. This is a critical issue for airplanes when arriving to and departing from airports. In order to avoid unsafe encounters near the ground, regulators have imposed airplane separation requirements for operations

Spurious atomistic viscosities in Smoothed Particle Hydrodynamics

Marco Ellero
Institute of Aerodynamics
Technical University Munich
Munich, Germany
marco.ellero@aer.mw.tum.de

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

***Abstract*—We consider a standard microscopic analysis of the transport coefficients, commonly used in Non-Equilibrium Molecular Dynamics (NEMD) techniques, and apply it to the Smoothed Particle Hydrodynamics (SPH) method in steady-shear flow conditions. As previously suggested by Hoover et al. [Physical Review E, 52 (1995) 1711-1720], we observe the presence of non-zero microscopic (kinetic and potential) contributions to the total stress tensor in addition to its dissipative part coming from the discretization of the Navier-Stokes continuum equations. Accordingly, the dissipative part of the shear stress produces an output viscosity equal to the input model parameter. On the other hand, the non-zero atomistic viscosities can contribute significantly to the overall output viscosity of the method. In particular, it is shown that the kinetic part, which acts similarly to an average Reynolds-like stress, becomes dominant at very low viscous flows where large velocity fluctuations occur. Remarkably, in this kinetic regime the probability distribution function of the particle accelerations is in a surprising good agreement with non-Gaussian statistics observed experimentally.**

I. Introduction

Smoothed Particle Hydrodynamics (SPH) is a meshless particle method able to discretize an arbitrary set of partial differential equations in a Lagrangian framework [1], [2]. In spite of its robustness and flexibility, SPH still suffers of problems related to its numerical accuracy. The existence of spurious transport coefficients in the method is well-known and it is usually associated to the occurrence of disordered particle configurations and artificial mixing. This is particularly evident for high Reynolds number flows where the particle inertia is large, effectively limiting the applicability of standard SPH for accurate direct numerical simulations (DNS) of turbulent flows [3].
In [4], a systematic study of the spurious SPH viscosities was performed and a direct connection with the the numerical SPH particle diffusion was highlighted. Alternatively, in [5], [6] a different approach was pursued in order to characterize the transport coefficients: by invoking the isomorphism linking the SPH equations describing an inviscid fluid with the ones governing the motion of an atomistic Lucy fluid, the authors performed Non-Equilibrium Molecular Dynamics (NEMD) simulations of the latter under homogenous shear flow to deduce the spurious transport coefficients of SPH. The presence of two *atomistic* viscosities in a thermostated Lucy fluid, analogous to the microscopic kinetic and potential ones, was reported. Intrinsic viscosity as well as other transport coefficients (i.e. thermal conductivity) [7] were lately observed and their dependence on applied shear rate and averaged velocity fluctuations (kinetic temperature) numerically quantified.
An interesting point highlighted by Hoover was that the the continuum interpretation of the two microscopic viscosities is very different. Indeed, although the potential one has no continuum analogous, the kinetic viscosity which is based on particle velocity fluctuations, acts formally as an averaged Reynolds stress in standard turbulence closures [8]. It is therefore very important to separate the two contributions in order to quantify the different source of errors and to understand whether intrinsic SPH dissipative mechanisms can be used as sub particle-scale (SPS) turbulent models in the spirit of Implicit Large Eddy Simulations (ILES) [9].

In this paper, we study the behaviour of the SPH spurious viscosity under homogeneous shear flow differentiating the several contributions. Unlike the study made in [5], here no microscopic Hose'-Hoover (or similar) thermostat is employed to achieve a non-equilibrium stationary state. The presence of an input viscosity through the SPH discretization of the full Navier-Stokes equations [10] allows to stabilize the simulations without imposing *a priori* a specified level of velocity fluctuations. As a consequence, particle velocity fluctuations are unconstrained, emerge naturally and eventually become dominant in very large Reynolds number flows: the corresponding regime is denoted in this paper as *kinetic regime*. The behaviour is studied under different choice of the Mach number and it is found to be predominant in strongly compressible situations. In this latter case, we observe a phase-transition in the averaged fluid density corresponding to the onset of the kinetic regime. Although the statistics of the particle velocity fluctuations remain Gaussian, the probability distribution function of the corresponding acceleration are stretched exponential increasing their flatness as the effective Reynolds number increases. This intermittency feature has been already observed in Voronoi simulations [14] and, despite a difference in the Mach number, it is in surprisingly good

Resolution study on Smoothed Particle Hydrodynamics with mesoscopic thermal fluctuations

Xin Bian
Institute of Aerodynamics
Technical University Munich
Munich, Germany
xin.bian@aer.mw.tum.de

Marco Ellero
Institute of Aerodynamics
Technical University Munich
Munich, Germany
marco.ellero@aer.mw.tum.de

Nikolaus A. Adams
Institute of Aerodynamics
Technical University Munich
Munich, Germany
nikolaus.adams@tum.de

***Abstract*—Recently, *thermal* fluctuations have been introduced into SPH [1] in order to simulate mesoscopic fluid phenomena. Developed within the GENERIC framework [2], the new model is denoted as Smoothed Dissipative Particle Dynamics (SDPD) and fulfills the *fluctuation-dissipation* theorem automatically.**

SPH is a scale-free method, in which convergent dynamics is achieved when the dimension of fluid particles is decreased; Differently, in SDPD the magnitude of thermal fluctuation is associated to the particle size, therefore defining the local dynamics. Although the local dynamics can differ for different particle resolutions, physical properties should not depend on the coarse-graining level [3].

A simulation code of SDPD has been implemented within the framework of the Parallel Particle Mesh library (PPM) [4], which takes care of interchanging information between processors efficiently. We first ran 2D mesoscopic Poiseuille flow for a simple fluid with thermal fluctuations. Although the thermal fluctuations of particles have different magnitudes for different resolutions, the ensemble velocity profiles converged to the theory at low Reynolds number. We have also modeled rigid colloidal particle suspended in a mesoscopic fluid. The velocity probability distribution function (PDF) of the fluid particles scales with different resolutions while for a colloid with predefined mass/size is invariant. This is confirmed also by the mean square displacement (MSD) which shows convergent behaviors.

I. Introduction

Since its inception, Smoothed Particle Hydrodynamics (SPH) has been explored extensively to solve various macroscopic flow problems in last decades [5], which demonstrated its capability of discretizing continuum systems on large scale. However, if one wants to study complex mesoscopic fluids, e.g., colloidal systems, thermodynamics has to be taken into account. The microscopic heterogeneity of colloidal particles is mainly determined by the ratio between viscous and Brownian forces and represents one of the key features to understand the complex rheological properties.

The most widely used mesoscopic simulation technique for colloidal particles suspended in a fluid is probably the Stokesian Dynamics (SD) [6], where a Langevin equation for the motion of Brownian colloids is considered and hydrodynamic interactions are computed through a grand resistance tensor. Despite the notable computational advantages of this approach, colloids with irregular shapes can not be simulated, therefore restricting the range of applications of SD for spherical objects only. An alternative approach is to model the thermal fluctuations directly in the fluid, instead of in the colloid equations. One method using this approach is Dissipative Particle Dynamics (DPD) [7] [8], which is a coarse-graining method. Starting from the stochastic differential equations for the DPD particles, hydrodynamics equations with expected Navier-Stokes form can be obtained via standard projection techniques [9]. However, there are several conceptual shortcomings about DPD. For example, the transport coefficients of the simulated system and the scale on which the simulation is performed are not defined a priori. Preliminary runs usually have to be performed to measure the viscosity and temperature of the fluid particles. The physical scale is not given as input parameter, which is a crucial problem when the physical dimensions of the suspended objects determine whether and, more importantly, to which extent thermal fluctuations come into play.

Starting from discrete SPH hydrodynamics equations, Español and Revenga [1] introduced thermal fluctuations by using the GENERIC formalism [2]. The new method, denoted as Smoothed Dissipative Particle Dynamics (SDPD), represents an extension of SPH. Indeed without temperature it solves the classical hydrodynamics, just like SPH; with temperature switched on, it handles mesoscopic flows and therefore it can be considered as a general multiscale framework linking SPH to DPD.

In section II, we present the formulation of SDPD and point out the extended part of SPH. Other aspects such as state equation for weakly compressible fluid, kernel function and time integrator will be also discussed. In section III, a simple fluid in Poiseuille flow is simulated with thermal fluctuations. Resolutions issues for the ensemble velocity profile are investigated. A model of hard solid colloid suspended in a Brownian solvent is introduced in the second part of this section. Both spherical and general shape of colloidal particles are simulated. Again, attention is paid to the resolution issues

SPH truncation error in 3D simulations

Andrea Amicarelli[1], Francis Leboeuf[1], Le Fang[1], Joëlle Caro[1]
[1,]Ecole Centrale de Lyon- LMFA
Lyon (France)
Andrea.Amicarelli/Francis.Leboeuf/Le.Fang/Joelle.Caro
@ec-lyon.fr

Jean-Christophe Marongiu[2], Julien Leduc[1,2]
[2]ANDRITZ HYDRO
Vevey (Switzerland)
Jean-Christophe.Marongiu/Julien.Leduc
@andritz-hyro.com

Abstract—**Following the procedure proposed by Quinlan et al. (2006) for a 1D generic derivative, we derived and validated 3D formulations of the SPH truncation error when reproducing a generic function (ε_{Tf}) and a first derivative (ε_{Td}). We then underlined the difference between non-consistent and consistent (Shepard's correction, renormalization) estimations.**

I. INTRODUCTION

The Smoothed Particle Hydrodynamics (SPH) is a mesh-less method based on an interpolation technique. A continuous field is approximated from discrete data points through the use of a regular function W, usually called kernel function, which is equal to zero outside of its support V_h, h being the smoothing length. As SPH standard formulation is affected by relevant truncation errors, some correcting methods have been developed in order to reduce it or to give some consistency to the SPH approximation: the Reproducing Kernel Particle Method (RKPM, Liu et al., 1995), the moving least square particle hydrodynamics (Dilts, 1999), the corrected SPH (Bonet and Lok, 1999), Liu and Liu's method (2006). Several properties of the truncation error have been remarked, from Monaghan (1992) to Vaughan (2009). In particular Quinlan et al. (2006) derived an explicit formulation for the SPH truncation error when reproducing a generic function gradient in 1D simulations. Adopting simple and generic hypotheses on the distribution of the particle volumes, we applied their procedure to get 3D formulations for the SPH truncation error in estimating both a generic function and a first derivative (Par.II). We then validated them on simple test cases of a fluid box (Par.III) and a channel sill (Par.IV). The corresponding conclusions are finally reported (Par.V).

II. 3D FORMULATIONS

We can define the SPH truncation error ε_{Td} at a computational point $(\underline{x_0})$, as the difference between the SPH particle approximation of a generic function f, or its derivative, and its actual value. We applied Quinlan et al. (2006) procedure, which allowed to obtain a formulation of ε_{Td} in 1D, using simple hypotheses on the particle volume geometry, in order to derive 3D formulations. The demonstrations are not reported here because of the size of this document. They are just cited as reference. In the following, we use a simple notation for the kernel function W centred at $(\underline{x_0})$, instead of the traditional one $W(\underline{x_0}-\underline{x},h)$ with h the smoothing length linked with the size of the kernel support; dx^3 is the differential element of volume.

The SPH truncation error ε_{Tf} is the difference between the SPH particle approximation and the actual value of the function:

$$\varepsilon_{Tf} = \sum_b f\big|_{\underline{x_b}} W_b \omega_b - f\big|_{\underline{x_0}} \qquad (1)$$

where the summation is performed on the particles "$_b$" included inside the support. We developed a second order Taylor's series approximation for f around $(\underline{x_0})$ within the kernel support of the computational particle and for W around $(\underline{x_b})$ within each particle volume. We adopt Einstein's notation for the subscripts "$_j$" and "$_k$", hereafter. In order to generalise the formulation and to underline the dependence of the truncation error on the kernel support length (h), we introduce non-dimensional kernel $(\hat{W})$ and distances:

$$\hat{W} \equiv h^3 W, \qquad \underline{s} \equiv \frac{\underline{x}-\underline{x}_0}{h}, \qquad \underline{s}_b \equiv \frac{\underline{x}_b-\underline{x}_0}{h} \qquad (2)$$

being ds^3 the differential increment of the non dimensional kernel support volume $(\hat{V}=V/h^3)$. We then assume the particle volumes to be non intersecting parallelepipeds, covering the whole fluid domain as a partition of it. Let's define $\underline{\delta}$ as the vectorial difference between the neighbouring particle position and its volume barycentre $(\overline{\underline{x_b}})$. We even introduce the normalised particle volume parallelepiped sides $(\underline{\Delta s}_b)$ and a normalised distance $(\overline{\underline{s_b}})$, difference between the particle barycentre and the computational particle position:

$$\underline{\delta_b} \equiv \frac{\underline{x}_b-\overline{\underline{x}_b}}{\underline{\Delta x}_b}, \qquad \underline{\Delta s}_b \equiv \frac{\underline{\Delta x}_b}{h}, \qquad \overline{\underline{s}_b} \equiv \frac{\overline{\underline{x}_b}-\underline{x}_0}{h} \qquad (3)$$

We finally define the vector of the ratios between the sides of the particle volume and the volume itself at the power of 1/3, as an anisotropy geometrical index:

$$\underline{I_b} \equiv \frac{\underline{\Delta s}_b}{(\hat{\omega}_b)^{\frac{1}{3}}} \qquad (4)$$

We could finally derive a 3D formulation for the SPH truncation error in estimating a generic function (Amicarelli et al. (a), in revision), in which Kronecker's delta $(\tilde{\delta}_{ij})$ appears:

In-Situ Visualization and Analysis of SPH Data using a ParaView Plugin and a Distributed Shared Memory Interface

Jerome Soumagne
Swiss National Supercomputing Centre
Manno, Switzerland
soumagne@cscs.ch

John Biddiscombe
Swiss National Supercomputing Centre
Manno, Switzerland
biddisco@cscs.ch

Jerry Clarke
US Army Research Laboratory
Aberdeen Proving Ground, MD, USA
clarke@arl.army.mil

***Abstract*—As SPH codes become more powerful and more interactive, it is desirable to monitor the simulation *in-situ*, performing not only visualization, but analysis of the incoming data as it is generated - so that the simulation may be stopped, modified, or potentially, steered, thereby conserving CPU resources. Monitoring or post-processing simulation data *in-situ* has obvious advantages over the conventional approach of saving to - and reloading data from - the file system. The time and space it takes to write and then read the data from disk is a significant bottleneck for both the simulation and subsequent post-processing. In order to be able to post-process data as efficiently as possible with minimal disruption to the simulation itself, we have developed a parallel virtual file driver (VFD) for the HDF5 library which acts as an MPI-IO file layer, allowing the simulation to write in parallel to remotely located distributed shared memory (DSM) instead of writing to disk.**

This paper describes an implementation of an *in-situ* framework which has been integrated into the ParaView visualization package alongside existing SPH analysis modules. The ParaView application acts as a server/host for the DSM and can read the file contents directly using the HDF5 API as if reading from disk. The transfer of data between simulation and visualization machines may be done using either an MPI based communicator shared between the applications, or using socket based communication. The management of both ends of the network transfer is transparently handled by our DSM VFD layer, meaning that an application using HDF5 can make use of in-situ visualization without significant code changes.

The complexity of HDF5 is such that many different data layouts are possible and it is not feasible to read all HDF5 files using a single common standard reader. To mitigate this problem, we support several readers/formats within our ParaView plugin, including the H5Part particle format. To enable the import of other arbitrary HDF5 data, we have made use of the Xdmf format which permits a *description* of the data to be supplied in XML form telling the reader how to convert data arrays into objects for visualization. Once data has been loaded, all existing ParaView filters and display methods are available and the user may setup an analysis workflow which is updated automatically as new data is received.

I. Introduction

The HDF5 library [1] provides the user with several different file drivers, which are the core pieces of code responsible for the transfer of user controlled memory onto disk. They act as an abstraction layer between the high level HDF5 API and the low level file system API. The drivers provided by the HDF5 package include the `core` (memory based), `sec2` (posix compliant serial IO), `mpio` (parallel file IO) and a streaming driver [2] has been created for the purpose of providing live access to simulation data by transfer to remote grid servers or via sockets to a waiting application. The `mpio` driver uses MPI-IO built on top of an HDF5 MPI layer to write data in parallel to the file system - our driver emulates this behaviour but instead routes the data in parallel to a Distributed Shared Memory (DSM) buffer over multiple TCP connections using either an MPI or a socket based protocol. Compared to other systems such as the ADIOS library [3] which defines an API and lets the user decide about which IO interface to use, we use here the flexibility of HDF5 so that codes already using HDF require only a few changes or no modification at all.

The original DSM implementation (upon which this work is based), referred to as the Network Distributed Global Memory (NDGM), was created by Jerry Clarke [4], it was used for CFD code coupling between applications modeling fluid-structure interactions [5], [6] using very different models (and hence partitioning schemes) to represent the domains. Since the DSM can be considered as a flat memory space, one of the principle advantages that it provides is that coupled simulations do not need to be aware of the parallel domain decomposition used by the other partners in the simulation/analysis. Separate codes may write their data using any HDF5 structures suitable for the representation, providing the other coupled processes are able to understand the data and read it with their own partitioning scheme. This effectively abstracts the data model used by either partner away and leaves the HDF5 API as the mediator between the coupled applications. The original NDGM implementation supported the transfer of data between processes using only a single channel of serial MPI based traffic and therefore had a limited capacity. Our new DSM based virtual file driver (VFD) allows very high speed *parallel* transfer of data directly between coupled simulations, or a simulation and a post-processing application such as ParaView [7] - for which we have created a custom plugin which allows full control of the visualization of live data. Our design is

A SPH model with C1 particle consistency

XU Haihua
Department of Civil Engineering
National University of Singapore
Singapore
g0600365@nus.edu.sg

DAO My Ha*, CHAN Eng Soon
Department of Civil Engineering
National University of Singapore
Singapore
*tmsdmh@nus.edu.sg

***Abstract*— In this paper, a weakly compressible SPH model with C1 particle consistency was established to simulate the free surface flow. The one-step searching method was used to restore the C1 particle consistency both on the continuity and momentum equations. The accuracy, energy conservation and free surface tracking of the standard SPH were improved, especially in long time simulation. The model was validated against several benchmarking problems, including a circular patch evolution, water oscillation in a tank and solitary wave propagation and breaking on the slope.**

I. INTRODUCTION

The Smooth Particle Hydrodynamics (SPH) is a Lagrangian particle method, which was first developed by Gingold and Monaghan (1977) to study the fluid dynamics in astrophysics. After its invention, SPH has become one of the most popular meshless methods and has been used in many areas such as astronomy, fluid mechanics, aerodynamics, solid mechanics and so on. Comprehensive reviews of the SPH method can be found in references [1, 2].

However, the standard SPH suffers several drawbacks, which include lack of particle consistency (completeness), tension instability, difficulty in apply essential boundary conditions and so on. In all of these drawbacks, the lack of particle consistency is one of the most severe problems. The particle consistency is the ability of particle method to approximate the polynomial functions. If the particle method can approximate the constant function, then it has C0 (0th order) consistency. If the method can approximate the linear function, then it has C1 (1th order) consistency. The detailed definition of the particle consistency can be found in the reference [1]. The lack of particle consistency can lead to poor accuracy in modeling slow dynamic problems, unexpected energy dissipation in long time simulations, un-accurate free surface tracking in free surface simulations. The tensile instability may also be caused by the lack of particle consistency [1].

Many approaches have been published to restore the particle consistency [2-4]. One drawback of these methods is the CPU intensive of the correction procedure. An additional searching step is needed to compute the correction coefficient. In SPH simulation, the searching step is the most CPU intensive work. Hence, the above methods are not quite efficient in SPH simulation. Based on Taylor series analysis, Liu, Xie et al. [5] proposed a one-step correction method. This method can restore C1 particle consistency and compute the particle-particle interactions in one searching step and then improve the accuracy of SPH simulation. Promising results can be found in the simulations of Poiseuille and Couette flow. Fang, Parriaux et al. [6] applied this method to the free surface flow problems. However, only the consistency of the surface particles was restored in their model. The non-surface particles which take most part of the particles in the simulation domain were not restored. Furthermore, the correction method only tested on very simple problems and simulated a relative short time. The improvements of the free surface problems with boundaries, the accuracy of free surface tracking in long time simulations of the correction method have yet been studied.

The purpose of this paper is to restore the particle consistency of the standard SPH to C1 particle consistency for free surface problems in an efficient way. By restoring the particle consistency, the new model was used to simulate the low speed free surface flow in a relative long time. The free surface problems with boundary conditions were studies by the new model. The accuracy of the free surface tracking and the energy conservation of the model were also studied in the paper.

II. GOVERNING EQUATIONS AND SPH APPROXIMATION

A. Governing equations

The free surface flow is governed by the Navier-Stokes equations which take the form of

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

$$\frac{d\mathbf{u}}{dt} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\mathbf{u} + \mathbf{g} + \nabla\cdot\boldsymbol{\tau} \tag{2}$$

where, ρ is the density. In 2D case, $\mathbf{u} = (u_x, u_y)$ is the velocity and p is the pressure of the fluid, $\mathbf{g} = (g_x, g_y)$ is gravitational acceleration, ν is kinetic viscosity and $\boldsymbol{\tau}$ is turbulent shear stress.

B. SPH approximation

The SPH is an interpolation method, which allows any function to be expressed by the value of a set of disordered particles (points) and weighted by the values of another function (called kernel function) at these particles. This expression of function has the form of:

$$f(\mathbf{x}) = \int_{\Omega} f(\mathbf{x}')W(\mathbf{x}-\mathbf{x}', h)\,d\mathbf{x}' \tag{3}$$

where $W(\mathbf{x}-\mathbf{x}', h)$ is called kernel function and h is the smoothing length. In this study the 5th order Quintic kernel function was used:

$$W(q) = \alpha_D\left(1-\frac{q}{2}\right)^4(2q+1),\ 0\le q\le 2 \tag{4}$$

Advanced Pre-Processing for SPHysics

Mayrhofer A.
Faculty of Mathematics
University of Vienna
Vienna, Austria
arnom@amconception.de

Gomez-Gesteira M., Crespo A.J.C.
EPHYSLAB Environmental Physics Laboratory
Universidad de Vigo
Ourense, Spain

Rogers B.D.
School of Mech., Aerospace and Civ. Eng.
University of Manchester
Manchester, U.K.

***Abstract*—Until now the amount of setups that could be simulated with SPHysics was limited by SPHysicsGen that initialises the program. With this paper we want to present a new add-on that allows the creation of arbitrary geometries. Compared to a similar 2D add-on the 3D version features improved usability and new possibilities when it comes to the use of external data, e.g. from CAD programs. Furthermore the changes to SPHysics are kept to a minimum so that the program can easily be adapted to the GPU and parallel versions.**

I. Introduction

Recent advances in the open-source software SPHysics make it possible to simulate millions of particles within reasonable time. This new capability paves the way for tackling real-life engineering problems. However, one major bottleneck needs to be mastered in order to achieve this, namely the particle initialisation. This is currently solved by a program called SPHysicsGen, which is capable of creating a limited set of geometries and objects.

In this paper, a new add-on shall be presented that combines the open-source software packages Blender, ParaView and SPHysicsGen. A combination thereof allows the creation of near arbitrary setups, e.g. the simulation of real bathymetries and geometries. The additional programs can both handle various file formats. This amongst others offers the possibility of importing CAD files.

In section II we will give an overview over a 2D program that allows the implementation of complex geometries but is severely limited by its usability. This will be overcome and extended to the third dimension by the add-on that will be presented in-depth in section III. To illustrate the new capabilities created by add-on we will show several exmples in section IV. We conclude this paper with a short résumé and an outlook to future developments.

II. MyGeo

The add-on MyGeo was introduced in [1] to simulate wave energy converters. Since the program is restricted to 2D, each boundary can be considered as parameterised curve. More precisely the user can implement a function $\gamma : [0,1] \times [0,T] \to \mathbb{R},\ (i,t) \mapsto \gamma(i,t)$ in Fortran code.

When creating complex geometries the usual filling of square regions is no longer appropriate. Thus a completely new filling algorithm had to be implemented. This happens by stepping through the whole domain and checking whether a given point on a regular grid is allowed to be filled. This is decided within a Fortran routine that has to be adapted by the user. An example of a filled geometry created with MyGeo can be seen in Fig. 1. It displays the lip of a Limpet (Land Installed Marine Powered Energy Transformer). The model also includes a piston connected on the left-hand side to generate waves (not displayed).

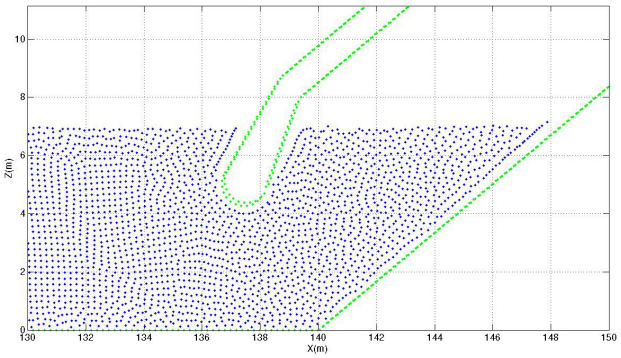


Fig. 1. Setup of a Limpet

To account for moving particles specific parts of the curve γ can be identified as moving. During the simulation itself they move according to a pre-specified pattern that again has to be hard-coded into the program.

During the development of the program it became apparent that many researchers are in thorough need for such an add-on. Amongst others there was the question about importing CAD files, but this is impossible so far with MyGeo. An important downside is that everything needs to be hard-coded in Fortran which requires a certain level of programming skills.

Since 3D simulations are considered to be the future of SPH simulations this add-on is no longer actively maintained. It proved to be an important and useful exercise when designing the 3D advanced pre-processing code since most of the problems have been eliminated.

Before moving on to the next section it shall be mentioned that this code only works for Dalrymple boundary conditions (with stationary fluid particles) [2], [3]. This is due to the fact that it is nearly impossible to calculate the normals for arbitrary geometries which are needed for Monaghan repulsive force boundary conditions [2], [4]. As this is even more complicated in 3D this is one of the only restrictions that remains.

Evaluation of SPH Capability to Model Internal Transient and Oscillating Flow Regimes

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

***Abstract*—Transient start up flows, variable imposed body forces and moving boundaries result in unsteady flow features in fluids. In literature, most SPH works dealing with internal unsteady flow simulations have presented the results for transient flows with constant driving forces at low Reynolds numbers. However, most of the fluid flow phenomena in nature and industry happen at higher Reynolds numbers. In this study, four benchmark unsteady cases are simulated to evaluate the ability of SPH to simulate internal transient and oscillating flows at moderate to high laminar Reynolds numbers. This allowed the evaluation of the performance of the two most commonly used formulations for viscous term modeling. Also, the effect of XSPH variant is tested. All SPH results are compared with analytical solutions.**

I. INTRODUCTION

In literature, SPH has shown that it is able to simulate internal transient start up of steady flows with a constant driving force when the viscous forces are comparable or larger than inertial forces (low Reynolds number, $Re << 1$) [1,2] and for Reynolds numbers $Re < 55$ [3]. However, most of the fluid flow phenomena in science and engineering have unsteady nature and happen at a variety of flow characteristics such as moderate to high Reynolds numbers. Of course, SPH is still in an introductory stage for simulation of such high Reynolds number showing turbulent features.

Here, we emphasize on SPH applications to simulate a variety of unsteady internal flows and discuss some features of SPH methodology we encountered. The transient behaviour of Poiseuille and Couette flows are analyzed in a range of Reynolds numbers. In each case, the two standard formulations for viscosity suggested by Monaghan et al. in 1993 [4,5] and made by Morris et al. in 1997 [1] are examined.

Also, our numerical tests show that XSPH variant can create disorder in particle movements especially in low particle resolution cases subjected to high velocity gradients.

The unsteady flow problems due to the oscillating boundary and pressure gradient at different frequencies and amplitudes are also simulated in this study. SPH was capable of simulating the correct velocity field even in the cases with high frequency in which the phase lag between the variation in velocity and the displacement of the moving plate or the pressure gradient force oscillations exists. This study has many applications in nature and industry: cardiac flows are unsteady by nature due to periodic applied pressure forces and moving boundaries and flows in industrial piping systems usually experience pulsatile effects in which the unsteadiness of pressure waves is an important factor.

II. FORMULATIONS AND METHODOLOGY

In fluid mechanics, the conservation of mass and momentum for an incompressible, Newtonian fluid flow leads to Navier-Stokes equations as

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

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

where ρ is fluid density, $\vec{V}$ velocity vector, $\vec{f}_{body}$ is called external body force indicating the applied forces per unit mass of the fluid element due to the external fields, P is pressure, μ presents fluid dynamic viscosity and t stands for time.

The fluid acceleration due to a directional pressure difference can be interpreted as the effect of an external body force. So, the pressure difference body force per unit of mass can be formulated as

$$f_x = -\frac{1}{\rho}\frac{dP}{dx} \tag{3}$$

Here, the classical SPH, weekly compressible, is used to formulate the governing equations of fluid flow.

The time derivative form of the conservation of mass leads to [4]

$$\frac{d\rho_a}{dt} = \sum_b m_b \vec{V}_{ab}.\vec{\nabla}_a W_{ab} \tag{4}$$

where $\vec{V}_{ab}$ is relative velocity.

Infiltration Induced Collapse in Coastal Cliffs

Johan Vandamme, Qingping Zou.
Centre for Coastal Dynamics and Engineering
University of Plymouth
Drake Circus, Plymouth, Devon, PL4 8AA.
Johan.Vandamme@plymouth.ac.uk

This paper examines the effect of infiltration and seepage on coastal bluffs using a hybrid particle method combining an adapted Weakly Compressible Smoothed Particle Hydrodynamics (WCSPH) with a static particle stability model and a one dimensional seepage model, based on Darcy's Law. Case studies are modelled and the results presented. A number of different static external water tables are imposed on the same initial numerical model domain, and the subsequent infiltration and soil pore pressure changes are captured. The variation in stability is analysed and discussed, and the failure of the cliff, where appropriate, is examined. Finally, a dynamic water table is analysed to provide insight into the effect of pressure variations on the soil surface in a storm surge. The research presented shows the importance of understanding the dynamics of external and internal water tables when considering the individual collapse mechanisms that cause cliff retreat.

I INTRODUCTION

Coastal sand dunes and bluffs provide an important, often naturally occurring, soft defence against coastal flooding and storm damage. The increasing value of coastal land, combining with climate change and rising sea levels, presents an important challenge to coastal scientists and engineers. The physical processes that drive coastal erosion, in particular the sudden collapse of cliffs, bluffs and dunes, needs to be understood with sufficient clarity to inform policies and coastal protection measures in order to maximise the impact of the limited resources used to control coastal damage.

Recent research [1] has shown the importance of including infiltration from storm surges when modelling the morphology of dunes and cliffs, and catastrophic collapse modelling requires a model which allows for sudden abrupt changes of geometry.

Many current models, such as EDUNE, SBEACH and the Coastal Construction Control Line (CCCL), use an equilibrium profile to predict the morphology of a beach face; however this approach is nonphysical when considering the individual cliff and dune collapses themselves, as opposed to considering an averaged profile over a longer time period. Other models, including work by Larson, Erikson et al. [2] and the Overton and Fisher model [3] use wave impact to drive the model, however this tends towards and over prediction in the quantity of erosion, and hence retreat.

XBEACH is also a frequently used model. This method uses a 2DH methodology to model the nearshore response to wave climates including infragravity waves. It uses a grid-based method to solve mass transfer and momentum balance across a staggered grid. Shallow water momentum is solved using the equations of Walstra [4]. XBEACH, however, has been shown to be highly sensitive to storm surges, and generally overestimates erosion in these conditions. This could be due to the avalanching methodology used for dune collapse [5].

As opposed to using an avalanching algorithm, there are many techniques to study the stability of a soil slope. Many techniques consider rotational or translational slips only, depending on the soil types. The majority of slope stability techniques use the Swedish (Fellinius) method of slices, or a derivative of this, for analysing circular failure mechanism [6, 7]. Standard errors of this technique range from 5-20% [8] depending on the soil conditions. Non circular failure planes can be analysed with methods of slices [9, 10], or a multiple wedge method [11, 12].

In order to consider in detail the effects of the water table and pore water pressures within the soil, it is important to model the infiltration and subsequent seepage of a storm surge or rainfall event. This is done in this model using a one dimensional application of Darcy's law, using saturation of the model particles, representing small areas of the soil, as a scalar parameter within the domain.

In this paper a number of simplified test cases are modelled in order to begin to quantify the effect of infiltration and seepage on Coastal Bluff stability. Both storm surge and rainfall events are considered, and a comparison of the distribution of the Factor of Safety (FoS) throughout the coastal bluff is discussed.

The Factor of Safety is a geotechnical measurement of the ratio of the theoretical maximum force available to resist failure within the soil geometry, to the force currently required for stability. Thus, the higher the FoS, the more stable the soil body, and a FOS equal to one infers a at the point of collapse.

In this paper, the fluctuation of the FoS due to infiltration and seepage within a dune or coastal cliff is examined, with the changing stability presented for a number of cases.

Improving the Performance of a Trapezoid Sloshing Vibration Absorber

Sean Kennan[1,2], Mahesh Prakash[2], Adam P. Marsh[3], S. Eren Semercigil[1] and Özden F. Turan[1]

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

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

[3]LFM Ecole Centrale de Nantes, FRANCE

A sloshing absorber is a vibration control device which utilizes intentional sloshing of a liquid in a container, to suppress excessive structural oscillations. Due to its simplicity, sloshing absorbers require no regular maintenance and are cost effective. The shape of the container of a sloshing absorber is a potentially significant design parameter with opportunities to enhance its rate of energy dissipation.

Earlier work suggested a trapezoidal shape with diverging walls as a promising candidate for further study [1]. The advantage of diverging walls, as opposed to the vertical walls of a rectangular container, is its superior ability to establish strong dynamic interaction (dynamic tuning) between the structure to be controlled and the oscillating free surface. To be an effective controller, however, strong interaction alone is not sufficient. An effective controller must also be able to rapidly dissipate energy.

The objective of this investigation is to study the effect of different shapes of obstructions, on the performance of a diverging trapezoid container. These obstructions are intended to increase the rate of shear dissipation during sloshing. The investigation is conducted numerically using Smoothed Particle Hydrodynamics (SPH) method.

I. INTRODUCTION

Sloshing is the low frequency oscillation of a liquid in a container with a free surface. Sloshing liquid has inherent ability to dissipate kinetic energy via shearing. Hence, it is possible to employ liquid sloshing as an effective energy sink in structural control applications, providing protection for structures exposed to excessive levels of vibration [2-4]. The basic principle in this approach, is to design the sloshing frequency to be the same as the critical frequency of the structure. This design, also called tuning, ensures strong interaction between sloshing and structural oscillations, similar to that of a classical tuned vibration absorber. Shear dissipation is employed to dissipate the harmful energy extracted from the structure as a consequence of tuning. Hence, a sloshing absorber is practically free of maintenance unlike the dissipative components of a classical tuned absorber.

Liquid sloshing in rectangular containers have long been studied to increase the energy dissipation performance [5-7]. These include using a net [8], wedge shaped objects on the container bottom [9], baffles on the container walls [10] and wall flexibility [11]. These studies have demonstrated that shallow depths are more effective at dissipating energy than deep liquid levels [12-15]. The effect of container shape, on the other hand, has not yet attracted much attention. An earlier work of the authors demonstrated that a trapezoid container possesses significant promise, as it is able to mobilise large amounts of liquid easily, and establish a tuning stronger than that of a rectangular container [1]. The problem with this strong interaction, and the large amounts of kinetic energy extracted from the structure as a result of the strong interaction, is the insufficient rate of dissipation. Poor dissipation causes the extracted energy being returned back to the structure periodically, significantly deteriorating the control performance.

The motivation of the present work is to explore the possibility of increasing the rate of energy dissipation in a trapezoid container, by introducing obstructions in the path of the sloshing liquid. These designed obstructions are expected to increase the velocity gradients and to enhance the shear dissipation as a result.

Smoothed Particle Hydrodynamics (SPH) is used in this study due to its grid free nature, and ability to predict the free surface behaviour closely [16,17]. Close correspondence reported earlier between SPH predictions and experiments [18] justify the purely numerical nature of the present study.

II. NUMERICAL MODEL

Smoothed Particle Hydrodynamics (SPH) code has been developed by CSIRO's Mathematics, Informatics and Statistics Division. SPH is used due to its ability to predict complex free surface behaviour accurately in this particular application. The container of a sloshing absorber is represented by a partially constrained moving boundary.

A schematic representation of the structure is given in Figure 1(a) where tethers represent structural stiffness (k) and viscous damping (c). The structure's stiffness, and mass (m) are 4260 N/m and 60.5 kg respectively, to give a natural frequency of 1.33 Hz. A 1% critical damping ratio is used to represent a lightly damped resonant structure. The sloshing fluid is water.

A particle size of 0.8 mm by 0.8 mm has been found to be fine enough in a two-dimensional space [18]. Time stepping is explicit and is limited by the Courant condition modified for the presence of viscosity [19], with a time step of integration of $1\times10^{-5}\,s$.

The fluid within the container is allowed to settle under gravity for a period of 5 s to a state of rest. The structure is then subjected to an initial velocity of 0.5 m/s over one time step.

Development of a smoothed particle hydrodynamics code for the numerical predicition of primary atomization of fuel injecting nozzles

C. Hoefler, R. Koch and H.J. Bauer
Institute of Thermal Turbomachinery
Karlsruhe Institute of Technology
Karlsruhe, Germany
Corresponding author: corina.hoefler@kit.edu

***Abstract*—A meshless particle code based on the Smoothed Particles Hydrodynamics (SPH) method [1] [2] has been developed. It is capable of simulating shear driven flows, free surface flows and two-phase flows of incompressible liquids using the Weakly Compressible SPH approach. Benchmark-tests of shear flows for validation of periodic boundary conditions and the viscosity model were conducted. By applying a modified Tait equation it is possible to account for surface tension effects. With this prerequisite the simulation of a two-dimensional liquid jet with fluids of different viscosities using the artificial viscosity model [6] was conducted. Preliminary results are presented which underline the promising performance of this approach. Our final goal is the generation of a so called virtual atomization test rig, which provides the possibility to simulate three-dimensional flow dynamics during the atomization process of various fluids.**

I. Introduction

Nowadays civil aviation is subject to rigorous emission restrictions, which will become even more restrictive in future. Consequently this comes along with the requirement for higher efficiencies and thus more effective jet engines. Jet engines operate on liquid fuels, therefore the kerosene has to be atomized, vaporized and mixed with air before an efficient combustion can take place. The atomization in turn can be optimized by small droplet sizes of the liquid fuel and dilute sprays. Furthermore, by enhancing prevaporization, the residence time of the droplets in the combustion chamber is reduced and its length can be minimized. Hence huge savings concerning space and weight can be accomplished in accordance with reduced emissions of CO_2 and NO_x.

The droplet size can be optimized by using appropriate atomizer concepts. Unfortunately up to now the numerical prediction of primary atomization of the liquid into ligaments and finally into single droplets using grid-based techniques is not possible or only with a huge effort of computational resources [3] [4] with up to 110 Mio. cells. These computations usually rely on the Volume-of-Fluid method, which includes some inherent inaccuracies. Costly surface-tracking algorithms are employed to model the advection of phase interfaces. Thus a common practical approach for the prediction of such combustors are experimental measurements of droplet-size- and droplet-velocity-distributions in the spray directly downstream of the nozzle exit. Subsequently, those test results are used as initial conditions for the simulation of the spray generation process together with secondary breakup models. Common measurement techniques in use, as e.g. Phase-Doppler-Anemometry, Laser-Doppler-Anemometry or Particle-Image-Velocimetry are expensive and time consuming. Additionally suitable test rigs for measurements at elevated pressure are required. In conclusion this means the atomizer nozzles have to be designed, manufactured, tested and re-engineered if necessary. In order to accelerate or to avoid this iterative process, more effective simulation programs are required. At this point SPH seems to be a promising alternative to the computationally expensive grid-based methods [1] [5] as interfaces are advected naturally and the spatial resolution is adapted automatically to the flow fields.

II. Method

A. SPH formulations

In our code we solve the Navier-Stokes equations as well as the governing conservation equation for the internal energy and an appropriate equation of state in order to close the system of equations. In case of the validation of the periodic boundary conditions the density is directly calculated by the summation density approach

$$\langle \rho \rangle_a = \sum_b m_b W_{ab} \quad . \tag{1}$$

However, for the simulation of the generic nozzle flow, the continuity density approach

$$\left\langle \frac{d\rho}{dt} \right\rangle_a = \sum_b m_b (v_a - v_b) \nabla W_{ab} \tag{2}$$

is more suitable as free surfaces are present.

The terms of the momentum conservation equation are calculated as pairwise, symmetric forces, e.g.

Numerical Study on Fluid Structure Interaction Using Smoothed Particle Hydrodynamics and the Finite Element Method

Qing Yang
Department of Aerospace and Ocean Engineering
Virginia Tech
Blacksburg, Virginia, USA
royyang@vt.edu

Van Jones
Department of Aerospace and Ocean Engineering
Virginia Tech
Blacksburg, Virginia, USA
cerchio@vt.edu

Leigh S. McCue
Department of Aerospace and Ocean Engineering
Virginia Tech
Blacksburg, Virginia, USA
mccue@vt.edu

***Abstract*—A loosely-coupled two-dimensional numerical model is proposed to investigate the fluid structure interaction (FSI) problem. The fluid domain is simulated by Smoothed Particle Hydrodynamics (SPH) and the structural domain by the Finite Element Method (FEM). This paper investigates the application of the SPH-FEM model to the FSI problem. Validation is performed by comparison with a dam break flow through an elastic gate done by Antoci et al. [1]. The results obtained in this paper show fair agreement with Antoci et al. [1] and suggest from this initial SPH-FEM study, that the SPH-FEM model is a viable and effective numerical tool for the FSI problem.**

I. Introduction

Meshless methods typically require no special treatment of fluid interfaces. This reduced complexity is especially useful when applied to simulations involving large free surface deformations or fluid interactions with highly deformable structures. Smoothed Particle Hydrodynamics(SPH) [10, 14] has been successfully applied in fluid structure interaction problems such as dam break flow [5], breaking waves on a beach [20], wedge slamming [23], sloshing tank [8, 24], and complicated geometry such as high speed ship flow [13], etc.

Most FSI problems using SPH have focused on fluid interactions with rigid bodies. Antoci et el. [1] extended the use of SPH into problems involving fluid interaction with a deformable structure. Antoci et al.'s [1] work shows encouraging results but also shows a need for improvement in the simulation of elastic body dynamics. Although SPH has been applied to elastic body dynamics [11, 13], additional terms such as artificial stress are needed to avoid tensile instability. Boundary deficiency artifacts also remain problematic. In this paper, finite elemental analysis is considered due to its maturity as a numerical tool in the analysis of structural dynamics without additional modification of governing equations. Because of the accuracy of FEM in solving structural dynamics and the convenience of SPH in simulating non-linear fluid dynamics, an SPH-FEM model is proposed to investigate the FSI problem. This paper is philosophically similar to the work of Groenenboom and Cartwright who have published using coupled SPH-FEM methods, see for example [12]. The most obvious difference lies in the modeling of the interface. Whereas Groenenboom and Cartwright use a penalty formulation [12], in this work a Monaghan type repulsive force is incorporated. Groenenboom and Cartwright's work [12] lends validity to pursuing this line of SPH-FEM-based analysis.

SPH formulations for the fluid domain are first described. A Couette flow case is presented to validate the viscosity model. The SPH boundary condition is discussed, with a Monaghan repulsive boundary chosen due to the simplicity of its implementation. Next, the structural theory of the finite element method applied to both small and large deformation cases is briefly given. Then the SPH-FEM coupling scheme is illustrated. SPH-FEM coupled simulation results are presented and analyzed. Improvements to the fluid structure interaction model and future work are discussed.

II. SPH Formulation

A. Continuity Equation

$$\frac{\mathrm{d}\rho_a}{\mathrm{d}t} = \sum_b m_b \, v_{ab} \cdot \nabla_a W_{ab} \tag{1}$$

This formulation preserves the Galilean invariance.

B. Momentum Equation

$$\frac{\mathrm{d}v_a}{\mathrm{d}t} = -\sum_b \left(\frac{P_a}{\rho_a^2} + \frac{P_b}{\rho_b^2} + \Pi_{ab} + R f_{ab}^n \right) \cdot \nabla_a W_{ab} \tag{2}$$

The quintic Wendland kernel [28] was chosen as the SPH kernel function W_{ab}. Morris et al. [21] and Monaghan and Kajtar [19] demonstrate the superior performance and stability of the Wendland kernel function relative to the commonly used cubic spline kernel.

Rf_{ab}^n is the artifical stress proposed in Monaghan [17]:

$$f_{ab} = \frac{W(r_{ab})}{W(\mathrm{d}p)} \tag{3}$$

SPHysics Code Validation Against a Near–Shore Wave Breaking Experiment

Christos V. Makris, Yannis N. Krestenitis
Dept. of Civil Engineering
Aristotle University of Thessaloniki (AUTh)
Thessaloniki, Greece
cmakris@civil.auth.gr, ynkrest@civil.auth.gr

Constantine D. Memos
School of Civil Engineering
National Technical University of Athens (NTUA)
Athens, Greece
memos@hydro.ntua.gr

***Abstract*—The use of the academic 'open source' code SPHysics, in the framework of wave breaking simulation over a relatively mild–sloping beach, is discussed in this paper. Thorough calibration of the relevant code's wide range of parameters and assumptions is attempted through the comparison between numerical and experimental data. A general validation of the Smoothed Particle Hydrodynamics (SPH) method's ability to capture the dynamics of near–shore wave breaking features and the characteristics of surf and swash zone turbulence, is implemented. Plausible qualitative agreement is achieved and inherent drawbacks and calibration weaknesses of the model are detected, based upon quantitative discrepancies. Moreover, particular amendments of the classic Smagorinsky–type turbulence model, incorporated in SPHysics, are suggested and the use of a more efficient one is introduced, setting the grounds for prospective research.**

I. INTRODUCTION

Coastal engineers and scientists are deeply concerned about the comprehension and the description of the detailed near–shore wave pattern evolution, in their effort to examine the extremely complex character of coastal processes. Among those, wave propagation, shoaling and depth–induced breaking are dominant. Specifically the latter is of major significance in assessing the surf/swash zone characteristics, such as the breaker wave height and type, the velocity and vorticity profiles, the undertow return–type flow, the shoreward net drift–type motion (Stokes's drift), the overall coherent and intermittent turbulent structures etc. All the above primarily control coastal sediment movements, thus long– and cross–shore morphodynamic evolution and secondarily the aeration and mixing processes inside the surf zone. The latter combined with the descending turbulent eddy formation, are responsible for the definition of quality and safety criteria for recreation and related activities. Moreover, climate change may aggravate consequent hazardous inundation events by extreme waves surging on especially low–land beach formations. Relative run–up on mild–sloping coasts as well as scouring due to turbulence at the toe of the associated steeper coastal protection works are of great importance.

Accordingly, near–shore wave breaking and related turbulence have been investigated adequately both physically and numerically, throughout the last decades, mostly accounting for the surf zone and secondarily for the swash and run–up ones. Despite that, the hydrodynamics describing the respective processes are far from completely elucidated. In this framework, laboratory experiments have been conducted, implementing various measuring techniques and introducing physical modelling of wave generation, propagation and breaking of the spilling or plunging type, on inclined beach slopes placed inside small scale wave flumes. Such are the photographic depiction of the breaking wave with simultaneous measurements of the free surface at specific gauge points throughout the propagation, surf and swash zone regions and the more elaborate modern Acoustic/Laser Doppler Velocimetry (ADV/LDV) or Anemometry (LDA), e.g. those used by [1] to trace obliquely descending eddies and more recent researches presented below. Furthermore Particle Image Velocimetry (PIV) methods [2] are frequently used nowadays, covering broader areas than gauges by recording high frequency frames of the flow field, thus depicting its overall turbulent patterns and structures. Extended and thorough reviews on the matter can be found in [3] and [4], while [5] focuses on the swash zone dynamics. On the other hand modern computational approaches comprise modelling of the full Navier–Stokes (NS) equations in combination with averaging and surface tracking techniques like RANS–VOF or Large Eddy Simulation (LES) models combined with Sub–Grid Scale (SGS) turbulence closure ones. Smoothed Particle Hydrodynamics (SPH) [6], [7] is the most widely discerned mesh–free (particle) method, used in several fields, especially standing out as a pledging modern technique in dealing with highly deformed free surface flows (e.g. plunging breakers), incorporating Lagrangian numerical formulation and rendering dispensable the strenuous employment of a toggling computational grid [8]. The implementation of SPH to a broad range of problems has guided researchers to important numerical corrections of the original SPH method, like moving the particles with the XSPH variant [9], re–initialising the density of the particles [10], incorporating Moving Least Squares (MLS) approach [11], introducing kernel [12] and kernel gradient corrections and dealing with tensile instabilities [13]. All of the above are taken into account in the present study through use of the recently issued academic 'open source' code SPHysics [14]. Some of the very recent proficient

Application of SPH-ALE method to Pelton hydraulic turbines.

Jean-Christophe Marongiu, Etienne Parkinson, Stefan Lais
Research and Development department
ANDRITZ HYDRO
Switzerland
jean-christophe.marongiu@andritz.com

Francis Leboeuf, Julien Leduc
Laboratory of Fluid Mechanics and Acoustics
Ecole Centrale de Lyon, University of Lyon
Lyon, France
francis.leboeuf@ec-lyon.fr

***Abstract*—The SPH-ALE method has been developed as a variant of the standard SPH method that includes an Arbitrary Lagrange Euler framework and the use of upwind fluxes between particles but also at boundaries. The method has been validated on reference cases and is now mature enough to be involved in the industrial process. The paper will give an overview of its use within the hydraulic Research and Development department of ANDRITZ HYDRO. In particular its usefulness in the design process of components of Pelton turbines or in the assessment of the mechanical stresses on components under a very high hydraulic load will be highlighted. The paper will also describe how it behaves compared to other numerical methods, showing that SPH-ALE is reliable and quite competitive. SPH-ALE can also be integrated in a multi-level approach in combination with more accurate numerical tools. Finally its unique ability to capture highly distorted free surface flows without interface diffusion will be underlined on the simulation of a complete 2 jets horizontal Pelton turbine, including the complete runner and the casing.**

I. Introduction

Computational Fluid Dynamics (CFD) has been applied for around 20 years in the design of various components of hydraulic machines. It is now a well recognized and indispensable tool in order to achieve better efficiency and meet project contractual performances, in complement to model tests. For the designer, CFD helps improving the hydraulic design and accelerating the development phase compared to a purely experimental development on a test rig. CFD is also used in order to perform the mechanical assessment of components under a hydraulic loading, in combined Fluid-Structure studies. The main numerical method used in CFD for hydraulic turbines or pumps is the Finite Volumes method.

Among the various types of hydraulic turbines, the Pelton turbine has the main specificity that it is an action type turbine, which means that the turbine transforms only the kinetic energy of water into mechanical energy. It is composed of a distributor, one to six nozzles, a rotating runner made of 18 to 26 buckets and a casing that covers the runner and directs the escaping water sheets from the runner to the tail water channel (see Fig. 1). Flows in distributor and nozzles are internal flows, whereas in the runner and casing they are free-surface flows.

Standard CFD techniques have been successfully applied to study internal flows in distributor and nozzles, leading to great

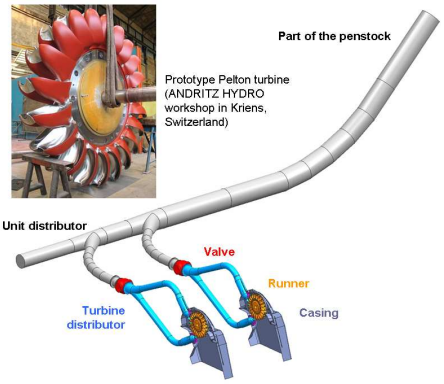


Fig. 1: Schematic view of a Pelton power plant

improvement in their hydraulic design. The numerical study of the free surface flows in runner and casing is more delicate and requires specific techniques. Up to now the Volume Of Fluids (VOF) method has been mainly used. With a great care and know-how in designing the computational mesh and setting the simulation, this method gives good results when dealing with flows in rotating buckets. However the numerical treatment of the rotor-stator interface in the frame of free surface flows introduces artifacts like free surface diffusion and energy losses. The rotor-stator interface can also be responsible of numerical instability when the spacing between static and rotating components is very small. Finally the position of the free surface is very sensitive to the mesh quality. The region of the jet and the bucket must be refined. Refining the outlet region is more difficult, so that it is usually hard to track the out-flowing water sheets on a long distance far from the bucket outlet and at a reasonable cost. The study of casing flows, i.e. the impingement of the water sheets on casing walls and the potential back flows on the runner or jets is not possible with the VOF method in the industrial process.

The above mentioned limitations of standard CFD techniques have motivated the interest for the SPH method. A model based on the SPH-ALE variant has been developed with the purpose of free surface flows in Pelton turbines, including casing flows. It makes use of Riemann solvers to

Flow modeling in a Turgo turbine using SPH

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

***Abstract* – In this paper the Smoothed Particle Hydrodynamics methodology is used for the simulation of the unsteady, free-surface flow in a Turgo water turbine. The standard SPH method was initially tested on a single stationary Turgo turbine blade impinged by a water jet under various angles. A particle resolution dependence analysis was conducted, to determine the optimum number of particles. Results were compared with the commercial CFD package Fluent®, regarding the free-surface flow, the pressure profile and the forces exerted on the blade. Finally the complete geometry of the runner, including the axis-symmetric hub and tip, was simulated using the present algorithm.**

I. INTRODUCTION

Turgo turbines belong to the impulse type turbine and are designed for medium head applications. They have a flat efficiency curve and provide excellent part load efficiency. Thus they can be used as an alternative of other turbine types, especially if there are large flow rate variations. Moreover they can operate for long periods and minimum wear when the water is laden with slit and other entrained matter [1].

The Turgo turbine runner looks like a Pelton runner split in half. The water jet enters from the one side of the runner and exits from the other (Fig. 1). Because of that, the escape of the water does not interfere with the incoming jet, or the other turbine blades. This means that a Turgo turbine is able to handle larger flow rates and jet diameters than a Pelton runner of the same runner diameter. As a result the Turgo turbine has higher specific speed and smaller size than a Pelton turbine of the same power. Moreover the smaller diameter allows the operation at higher angular velocities, which in turn, makes the coupling between the turbine and the generator easier, avoiding the use of a mechanical transmission system decreasing costs and increasing the mechanical reliability of the system [1].

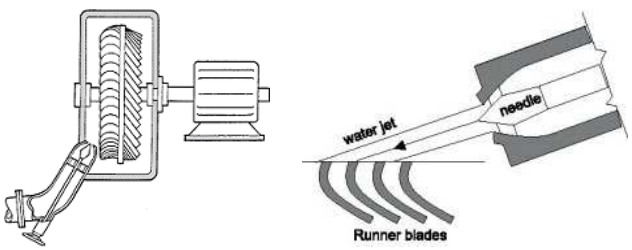


Fig. 1. Sketch of a Turgo turbine (left) and flow (right)

The operation of such turbines involves free-surface unsteady flows, thus making the modelling of the flow difficult. Indeed Turgo turbine dimensioning is based on experimental or empirical data [1]. Traditional grid-based methods are difficult to use for such flows [2], since special techniques are required, such as the VOF technique and mesh adaptation. On the other hand the method of Smoothed Particles Hydrodynamics (SPH), due to its Lagrangian, mesh-free nature, handles easier such complex flows. The objective of the present paper is the flow modelling and simulation in the Turgo turbine using SPH and the validation of the results using Fluent® using the VOF (Volume Of Fluid) method.

II. GOVERNING EQUATIONS

The equations used are those obtained after the standard SPH discretization procedure as it is presented by Violeau and Issa [3]. The same equations were used in the previous work of the authors of the present paper [4].

In short, the continuity equation is used for the density calculation:

$$\frac{d\rho_a}{dt} = \sum_b m_b \mathbf{u}_{ab} \bullet \nabla_{\mathbf{a}} \mathbf{W}_{\mathbf{ab}} \tag{1}$$

and the momentum equation :

$$\frac{d\mathbf{u}_a}{dt} = -\sum_b m_b \left[\left(\frac{p_a}{\rho_a^{\,2}} + \frac{p_b}{\rho_b^{\,2}} \right) \nabla_{\mathbf{a}} \mathbf{W}_{\mathbf{ab}} - \Pi_{ab} \right] + \vec{\mathbf{f}}_{\mathbf{ext}} \tag{2}$$

where Π_{ab} is the Morris viscosity term. In the following simulations the flow will be considered inviscid.

For the pressure calculation the Tait equation of state is used:

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

The parameters in eq.4 are set, taking in consideration the limitations regarding the speed of sound and the maximum density variations [5].

It is known that even if SPH is able to describe the general dynamics of the flow, pressure distribution across particles exhibits large oscillations. For this reason a Shepard density filter [6] is used.

Also for moving the particles more orderly, the XSPH variant [7] is used, to compute particle motion:

The Use of 3D SPHERA Code to Support Spillway Design and Safety Evaluation of Flood Events

Agate G., Guandalini R.
Environment and Sustainable Development Dept.
ERSE s.p.a., Via Rubattino, 54 – 20134
Milan, Italy
roberto.guandalini@erse-web.it

***Abstract*—The study of real structures in hydraulics engineering is a topic of great interest in several research and applied fields. Today, with the advance in computer technology, the behavior of hydraulic structures can be investigated numerically in reasonable time and costs. This fact added to the increase of accuracy in the mathematical models, gives the possibility to the engineers to replace physical model tests and to solve problems related to dam and river hydraulics in a number of different scenarios. In the frame of the projects funded by the Italian Electric System Research fund, the 3D numerical code SPHERA, based on the SPH method, has been developed and tested in order to study different classes of problems connected to the hydro electrical power plant safety and efficiency, having in mind both the design support and the accident prevention point of views. The paper describes the application of the fully 3D SPH model of the SPHERA code to different classes of problems occurring in the hydraulic power plant design and operation, among them the analysis of spillway behaviors and flood events. Results such as level and pressure profiles are compared both with available experimental data and with CFD reference simulations resulting in a very good agreement and low computational costs. These satisfactory results are due to the recent improvements in the code development, as the new pre processing module for the automatic generation of complex geometries, the use of general semi-analytic approach for modeling solid boundaries and different criteria for the treatment of non cohesive granular sediments. The activity described has proved that the SPH approach included in SPHERA can be successfully used to reproduce and to analyze most of the relevant physical and engineering aspects of phenomena involving hydraulic structures with a good accuracy in comparison with traditional CFD approach and with significantly reduced costs with respect to physical models.**

I. INTRODUCTION

The study of real structures in hydraulics engineering is a topic of great interest in several research and applied fields. Today, with the advance in computer technology, the behavior of hydraulic structures can be investigated numerically in reasonable time and costs. This fact added to the increase of accuracy in the mathematical models, gives the possibility to the engineers to replace physical model tests and to solve problems related to dam and river hydraulics in a number of different scenarios. In the frame of the projects funded by the Italian Electric System Research fund, the 3D numerical code SPHERA, based on the SPH method, has been developed and tested in order to study different classes of problems connected to the hydro electrical power plant safety and efficiency.

The definition of the classes of problems of interest comes from the consideration of those problems whose characteristics can be studied successfully using the SPH model and methodology included in the SPHERA code, having in mind the usual operating actions, the design support requirements and the accident prevention point of views. More precisely, the following classes have been considered:

- problems related to the dam bottom outlets including also the water-induced sediment scour. In this class are mainly included the effects due to the opening of the dam bottom outlets. Solid particles are moved and dragged by water flow with the subsequent erosion and transportation of sediments downstream;
- dam-break problems. In this class the dam break effects have been considered mainly from the point of view of the impact of water flow on structures encountered downstream the break location, and therefore the analyses have been limited to the initial time transient;
- spillway problems. In this class the effects caused by a flood event on spillways have been considered both from the point of view of accuracy of the numerical representation of the physical phenomena involved and the potential optimization of the spillway design.

In the frame of each class of problems a number of numerical simulations have been carried out in order to analyze the code features in different significant conditions at real scale; of course, the scenarios described in this paper are not exhaustive of all these simulations, but they represent a set of analyses sufficient to evaluate the accuracy and the quality of the results obtained.

II. SUMMARY OF THE RECENT CODE IMPROVEMENTS

The consideration of different scenarios involving real scale domains with an high level of physical complexity have required some model improvements in the SPHERA code in order to get a more accurate evaluation of the physical aspects and to add special techniques and more efficient algorithm with the goal of increasing calculation performances. As a matter of fact, the main items have been:

High-Performance Fluide-Structure Interactions for Impacts with Fast Dynamic Europlexus Software

Caleyron F., Combescure A.
Université de Lyon
CNRS INSA-Lyon, LaMCoS UMR5259
F-69621 Villeurbanne, France
alain.combescure@insa-lyon.fr

Faucher V.
CEA-Saclay
DM2S/SEMT/DYN
F-91191 Gif-sur-Yvette, France
vincent.faucher@cea.fr

Potapov S.
EDF-Clamart
LaMSID, UMR EDF/CNRS 2832
F-92141 Clamart, France
serguei.potapov@edf.fr

***Abstract*—The work is devoted to the failure prediction of a tank filled with fluid under an impact loading and the resulting fluid leakage. The structural part of the problem is modeled using Mindlin-Reissner finite elements thick shells. The modeling of the fluid is based on an SPH formulation. The problem involves strong fluid-structure interactions which are handled through a master-slave based method and the pinballs method. Computations are run with an high-performance version of the code involving parallel calculations which suits well to the SPH formulation. Numerical results are compared to experimental data. A steel piston filled with water is impacted at a prescribed velocity by a falling mass. The piston is closed at its base with an aluminium plate whose characteristics are allowed to vary. The impact creates a pressure wave into the fluid which is responsible for the deformation of the plate and eventually the propagation of cracks.**

I. Introduction

Some recent and dramatic accidents have launched some studies to assess the possibility of failure and leakage rate of thin structures filled with fluid. These are complex problems that involve highly nonlinear responses both in the fluid and the structure. Moreover, the fluid-structure interface changes drastically during the transient, for instance in the case of large motions of the fluid free surface or when the structure containing fluid looses its integrity and fluid escapes through created tears. As a consequence, fluid-structure interactions modeling is a key feature to obtain reliable numerical simulations.

Meshfree formulations like the SPH formulation suit very well for these complex situations involving important topological changes as well as very nonlinear behaviors. Thus, the choice was made to use an SPH model for the fluid part of the problem. Structural part is modeled through standard finite elements method since it handles nonlinear behaviors well and is reliable. Two fluid-structures interactions approaches were tested, respectively the pinballs method and a master-slave based method.

Numerical results are confronted to an experiment for which a cylindrical reservoir filled with fluid is suddenly compressed by a falling mass and exhibits tearing of the shell and a fluid leakage. The first part of the paper presents the experiment, the second part is dedicated to its numerical modeling and the third part to the results and discussion.

II. Experimental campaign

A. Introduction

In order to study fluid-structure interactions in impacts context, a simple but physically representative experimental campaign has been conceived and conducted on the drop tower of ONERA Lille. The experiments consisted in exhibiting leakage through a suddenly created tear in a piston-like vessel filled with water. The piston was impacted at its top by a dropped mass, which generated a sudden rise of the internal pressure leading to the deformation of the plate closing the piston's bottom and eventually the propagation of cracks.

B. Experimental setup

The main piece of the experimental setup is a thick-walled steel piston-type vessel with an inside diameter D=80 mm filled with water at ambient temperature, see Fig. 1. The initial water column height, is H=230 mm. The piston is closed at its base with an aluminium plate whose characteristics are allowed to vary. A mass of M=245 kg is dropped from a pre-determined height impacting onto the piston. Two impact velocities have been considered $V_i = 2m/s$ and $V_i = 5m/s$. The falling mass is a rigid trolley, which can slide along the drop tower guided by a rail system see. The crash tower is installed on an 80-ton foundation with antiseismic supports, which decouples the crash zone from the infrastructure.

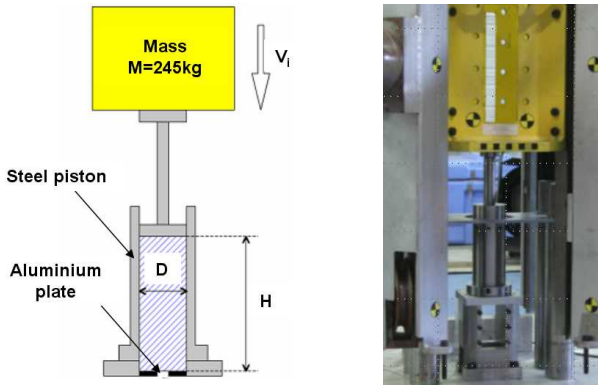


Fig. 1. Experimental setup.

SPH for a 2-D and 3-D heaving body using variable mass particle distribution

Pourya Omidvar*, Peter K. Stansby and Benedict D. Rogers
School of Mechanical, Aerospace and Civil Engineering,
The University of Manchester, Manchester, M60 1QD, UK.
Pourya.Omidvar@postgrad.manchester.ac.uk

Abstract—**A heaving wedge or cone with oscillatory vertical motion is a challenging test case for numerical free-surface schemes, comparing with experimental data. The motion of the wedge/cone in initially still water is forced as the surface motion of a focused wave group. Here, using the open-source code SPHysics [1], we first show how this test case works in 2-D using two forms of repulsive boundary condition [7, 11] and particle boundary force [12]. Using a repulsive boundary condition seems to work better than a particle boundary force. The force comparison between SPH and experiment for 3-D heaving cone is in reasonable agreement in terms of phase and magnitude. Capturing the entire three-dimensional flow field however would require a simulation with an extremely large number of particles and large computing resource. To circumvent this issue, this work employs variable mass distribution around a heaving device generating surface waves. Using variable mass distribution in a pre-selected area avoids the need for a dynamic particle refinement scheme.**

I. INTRODUCTION

Investigation of interactions between water waves and wave energy devices has become a prominent research area. For the case of multiple heaving wave energy devices the flow field is potentially complex since each device is moving and there is a need to study the basic behaviour to enable the design for such a complex problem. Modelling multiple heaving wave energy devices which involve complex interactions of wave breaking, reflection and diffraction processes can help to understand their behaviour in real seas. At the moment there is a limited knowledge of floating bodies in extreme waves, most of which use either physical experimentation or sea trials. Standard approaches such as linear and second-order wave diffraction theory do not represent highly non-linear effects associated with extreme waves.

Smoothed Particle Hydrodynamics has an advantage in dealing with free-surface problems where there is no need for special treatments for the free surface to simulate highly non-linear and potentially violent flows. At the University of Manchester, we are using the SPH method to simulate wave propagation and loading on the wave energy device in a water channel.

Here, following the work by Vila [2], Guilcher *et al.* [3] and Omidvar *et al.* [4], we use SPH to investigate waves generated by a 2-D and 3-D heaving bodies. This paper will include a summary of the experimental data provided by Drake *et al.* [5] followed by a brief introduction to the numerical scheme. We then present the results for 2-D wedge and 3-D cone heaving in still water showing comparisons of force with the experimental data. Finally, after the previous work by the authors [6], the use of variable particle size will be shown for 3-D heaving cone, which is used in a pre-selected area and avoids the need for dynamic particle refinement.

II. EXPERIMENTAL DATA

In this work, we use the experimental data of Drake *et al.* [5] in order to validate our SPH results. Drake *et al.* [5] investigated experimentally the motion of a cone, which is forced to oscillate vertically in still water in a wave tank, and successfully compared the results for force and surface elevation with the linear and non-linear theory. The cone was located centrally across the width of the tank with a depth of 1.01m. The cone draught was 148 mm, equal to the waterline radius in case of a right circular cone. The vertical motion of cone $z(t)$ followed the form of a Gaussian wave packet defined by:

$$z(t) = A\sum_{n=1}^{N} Z(\omega_n)\cos[\omega_n(t-t_0)]\Delta\omega_n \,, \tag{1}$$

where

$$Z(\omega_n) = \frac{1}{\sigma\sqrt{2\pi}}\exp\left[-(\omega_n-\omega_0)^2/2\sigma^2\right], \tag{2}$$

$\sigma = \omega_0/2\pi$, $\Delta\omega_n = 2\omega_0/N$, central circular frequency $\omega_n = n\Delta\omega_n$, A is the largest excursion from the still water level, ω_0 denotes the central frequency of the Gaussian wave packet in rad/s, were given by $\kappa\pi/3$.

For our SPH simulation A is chosen to be +50mm, the number of frequency components N is 50, and κ is 9. In accordance with Drake *et al.* [5] suggestion, the force results are compared with the experimental data in non-dimensional form as $F/\rho g\pi a^2 A$ where F is the time varying component of the vertical fluid force, ρ is the density of fresh water, g is the

A Rectangular Sloshing Absorber with Designed Obstructions to Improve Energy Dissipation

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

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

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

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

Earlier work demonstrated the superior energy dissipation capability of low-level liquid absorbers with travelling sloshing waves, as opposed to deep-level liquids with sustained standing sloshing waves [1]. In this work, travelling waves had a clear wavefront with severe velocity gradients to contribute to the dissipation efforts significantly. The presence of this wavefront offer possibilities of enhancing the shear dissipation further, by placing surface roughness elements (obstructions) on the bottom of the absorber's container. Investigating the effect of the number and location of these designed obstructions is the topic of interest in this paper. Both experimental and numerical predictions (obtained with Smoothed Particle Hydrodynamics - SPH) are presented in the form of design recommendations.

I. INTRODUCTION

Tall structures, such as towers and bridges, are subject to wind and earthquake loads which can cause them to oscillate at excessive magnitudes. Liquid sloshing absorbers can be used to suppress these excessive oscillations for structural control purposes [2-5] by tuning the frequency of the sloshing to the critical frequency of the structure to be controlled. These absorbers are simple structures consisting of a partially full container of liquid with a free surface.

Shallow liquid level rectangular sloshing absorbers are the focus of this paper. This is because shallow liquid level sloshing absorbers have been found to be more effective energy dissipaters than deep liquid level absorbers [6-9]. Rectangular shaped sloshing absorbers have attracted considerable attention in the literature [1, 5, 10-12]. Earlier work demonstrated superior energy dissipation capability of low-level liquid absorbers with travelling sloshing waves, as opposed to deep-level liquids with sustained standing sloshing waves [1]. In this work, travelling waves had a clear wavefront with severe velocity gradients to contribute to the dissipation efforts significantly. The presence of this wavefront offer possibilities of enhancing the shear dissipation further, by placing surface roughness elements (obstructions) on the bottom of the absorber's container.

Variations of the standard rectangular container sloshing absorber to improve the energy dissipation performance have been a focal point of several previous studies. Some variations include flexible container walls [13], a fluid submerged net in the container [14], baffles attached to container walls [15], wedge shaped fluid obstacles on the bottom of the container [16] and a study of various container shapes [1]. However there has yet to be a systematic investigation into the effect of surface roughness elements (obstructions) on energy dissipation performance.

Numerical predictions in this study are undertaken using SPH due to its ability to simulate liquids without the need for a mesh structure. Here SPH is used to model a 2-dimensional rectangular sloshing absorber with obstructions. Due to its Lagrangian nature SPH can accurately capture complex free surface behaviour [17, 18]. Comparisons between SPH predictions and experimental observations using a rectangular liquid sloshing absorber showed good agreement [19]. Here, comparisons between numerical predictions and experimental observations are made for displacement history and free surface behaviour by including surface roughness elements to the base of a rectangular absorber. Investigating the effect of the number and location of these designed obstructions is the topic of interest in this paper.

II. EXPERIMENTAL SETUP

A line drawing of the experimental setup is displayed in Figure 1(a). This consists of a mechanical oscillator whose motion is designed to be rotational around a pivot point. A container to accommodate the sloshing liquid is mounted on top. As this structure is excited, the container on top is subjected to angular oscillations. A photograph of the experimental setup is shown in Figure 1(b). The rectangular absorber's size remains constant for the study with a width of 340mm. Therefore the wavelength of the fluid is also 340mm.

ASPH modeling of Material Damage and Failure

J. Michael Owen
Lawrence Livermore National Laboratory
M/S L-038
P.O. Box 808
Livermore, CA, USA
email: mikeowen@llnl.gov

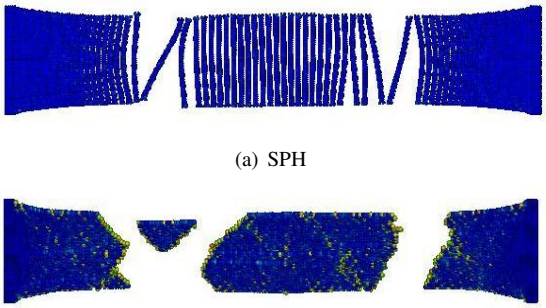
***Abstract*—We describe our new methodology for Adaptive Smoothed Particle Hydrodynamics (ASPH) and its application to problems in modeling material failure. We find that ASPH is often crucial for properly modeling such experiments, since in most cases the strain placed on materials is non-isotropic (such as a stretching rod), and without the directional adaptability of ASPH numerical failure due to SPH nodes losing contact in the straining direction can compete with or exceed the physical process of failure.**

I. Introduction.

Modeling the processes of damage and fracture of materials is an area where meshless methods offer clear benefits over traditional mesh based hydrodynamic approaches. Damage models ([1], [6]) involve following history variables (such as a damage fraction) tied to distinct chunks of mass through large distortions in the fluid flow. These highly dynamic systems usually force mesh based methods to perform some sort of remap of the system in order to maintain a valid mesh, introducing diffusive errors into the history variables. Robust Lagrangian methods such as SPH solve this problem trivially as their Lagrangian nature is ideally suited for following properties tied to the mass distribution, though of course meshless methods come with their own difficulties.

One challenge modeling material fracture presents for a standard SPH approach is that the material strain field is often strongly anisotropic, implying that the nodes representing the material will undergo very anisotropic displacements. Consider a rod being pulled apart along its length: the nodes along the length of the rod will be separating, while the nodes across the width of the rod will actually be approaching one another as the rod compresses in that direction. This presents a problem for adapting the scalar smoothing scale h_i in SPH. The goal of adapting h_i is to keep the number of neighbors for each SPH node roughly constant, which in our tensile rod example will result in a compromise such that we preferentially sample more and more nodes across the width of the rod while simultaneously losing neighbors in the direction of stretching. The worst case scenario as this process continues is that the nodes will ultimately have too few neighbors in the stretching direction in numerical fracture will occur simply due to the nodes becoming decoupled.

One natural solution to this difficulty is to employ Adaptive Smoothed Particle Hydrodynamics (ASPH) [7], which replaces the scalar smoothing scale h_i of SPH with a symmetric tensor $H_i^{\alpha\beta}$. The use of $H_i^{\alpha\beta}$ allows the local sampling volume for each node to be an arbitrary ellipse in 2D/ellipsoid in 3D, freeing the technique from the unit aspect ratio sampling of SPH. The ASPH algorithm strives not just to keep a constant number of neighbors per node but also a constant number of neighbors *in each direction* around each node, ensuring problems such as numerical fracture in our tensile rod do not occur. In Fig. 1 we compare two different models of a 2D rod stretching horizontally, one SPH and one ASPH. The color scale is the magnitude of the damage being evolved on each node. The SPH model in Fig. 1(a) has failed numerically, evidenced by the fact that at the break points the damage variable does not indicate failure. By contrast the ASPH model in Fig. 1(b) has failed where the physical damage model indicates that it should. Fig. 2 compares the evolution of the sampling volume shapes in these two models. The SPH model in Fig. 2(a) shows the spherical sampling around each node overlapping extensively across the width of the rod while separating along the length, whereas the ASPH shapes in Fig. 2(b) demonstrate the successful adaptation of ASPH sampling volumes to distortion of the material.



(a) SPH

(b) ASPH

Fig. 1. Damage magnitude for (a) SPH and (b) ASPH models of a stretching rod in 2D.

II. Choosing the SPH and ASPH smoothing transformations.

The ASPH fluid evolution equations are identical to the SPH forms if one appropriately removes the references to the smoothing scale by working in "normalized" coordinates η^α, where η^α is defined as $\eta^\alpha = x^\alpha/h_i$ in SPH and $\eta^\alpha = H_i^{\alpha\beta}x^\beta$ in ASPH. Note that $H_i^{\alpha\beta}$ has units of inverse length, and ASPH

Modelling Thixotropy with SPH: Application to Ceramic Processing

Andreas Wonisch, Torsten Kraft, Michael Moseler, Hermann Riedel
Fraunhofer Institute for Mechanics of Materials IWM
Wöhlerstr. 11
79108 Freiburg, Germany
andreas.wonisch@iwm.fraunhofer.de

***Abstract*—A SPH implementation of thixotropy is presented which allows simulating the time and shear rate dependent viscosity of complex fluids. The changing internal structure of such non-Newtonian fluids is represented by a scalar parameter for each SPH particle. Thixotropy is then introduced by defining a constitutive equation for the time derivative of the structure parameter and another one for its relation to viscosity. We demonstrate how this SPH extension can be used to model the complex flow behaviour in industrial processes involving free surfaces. Two applications of ceramic processing for multi-layer circuitry are presented: Tape casting in which large, thin ceramic tapes are cast on a moving substrate and the subsequent screen printing process where circuits, resistors and other electronic components are printed on the dried tapes.**

I. Introduction

Many non-Newtonian fluids exhibit thixotropic behaviour: Under shear the viscosity gradually decreases with time until equilibrium is reached. This time and shear rate dependent rheological behaviour has its origin in the breakdown and build-up of structure in the fluid and is usually fully reversible. It should be noted that thixotropy is often used synonymously with shear-thinning, even though the definition of thixotropy specifically addressed the *time-dependent* component [1].

Typical examples of thixotropic fluids are concentrated gels, which can be liquefied by applying shear and resolidify under rest. In many industrial applications this behaviour is desirable because it simplifies usage. For example, paints can be easily applied by stirring them and afterwards slowly become more viscous again. Further applications are e.g. ceramic colloidal shaping processes where the ceramic powder is dispersed in a solvent [2]. In order to simulate the flow behaviour in these processes, which usually involve complex (free) boundaries, a suitable simulation technique is needed. Smoothed Particle Hydrodynamics (SPH), which offers a flexible treatment of free surface flow, has already been demonstrated to be able to simulate industrial processes [3]; however, it lacks a comprehensive treatment of thixotropic flow behaviour.

So far SPH has been mainly used to simulate Newtonian fluids. Even though the extension to generalized Newtonian fluids, where the viscosity is a function of the applied shear rate (but not upon the shear history), is straightforward, it has only been fairly recently implemented [4]. Some recent work also focused on SPH implementations of viscoelasticity [5,6]. However, often viscoelastic effects like stress relaxation or normal stress differences are not important for industrial processes and can be neglected. For these flows a *simple,* yet *robust* implementation of thixotropy is needed.

The Lagrangian approach of SPH makes the implementation of thixotropy much easier compared to traditional, grid based computational fluid dynamics schemes: Each SPH particle can carry information like local viscosity based on its shear history. In this work, we show how a classical, structural kinetics model for thixotropy can be extended straightforward to SPH by adding a scalar structure parameter as an additional internal variable for each particle. In this case, the desired thixotropic behaviour can be easily prescribed by using constitutive equations with fluid-specific parameters that can be determined from experiments.

II. Simulation Model

A. *Thixotropy model*

There has been considerable work on general models describing thixotropy, which broadly fall into three categories [1]. The continuum mechanics approach directly uses time-dependent viscosity functions which can be fitted to experimental data. In contrast, microstructural models try to model the internal properties of the fluid that result in thixotropy and, thus, are specific to a certain fluid and its mechanism. Closely linked to the micostructural approach are structural kinetics models that use a more general structure parameter which is related to viscosity. While in principle all three model groups can be used for a SPH implementation of thixotropy, we will employ the structural kinetics approach because of its general form, which can be applied to a wide range of fluids.

Structural kinetics models of thixotropy are typically based on a one-dimensional measure of structure inside the fluid: a scalar structure parameter λ [1,7]. In this approach, $\lambda = 1$ corresponds to fully developed structure (high viscosity), while for $\lambda = 0$ the structure is completely broken down (low viscosity). The relationship between λ and the viscosity η is determined by a prescribed constitutive equation. If the fluid's

Third Generation RSPH: Towards robust and simple integration with SPH techniques

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

***Abstract*—This is a report on the development of the third generation Regularized SPH (RSPH) method. RSPH has over the last decade or so been shown to give improved descriptions of shock wave problems in compressible fluids compared to that typically provided by conventional SPH. However, the RSPH method has only to a limited degree been applied to problems where conventional SPH normally is preferred over grid-based methods. This revision of key elements of the RSPH method is meant to address issues that are believed to have prevented the RSPH method from being applied more widely. In this paper, characteristic features of the new RSPH method are discussed. In addition, reports on preliminary results are given where the new method is applied to both compressible and weakly-compressible test cases.**

I. Introduction

In general SPH discussions one is often faced with simplified statements from people that have a fairly one-sided view of the method. This can often be ascribed to the fact that the people making the statements do not themselves have any experience in working with the method. Still, even with people having worked quite considerably with SPH, one might meet strong statements regarding the usefulness of the method. This is presumably a direct result of the personal experience people have had working with the method: One the one hand, SPH can be a very powerful tool dealing with problems perhaps no other method can handle. On the other hand, SPH might perform poorly (or fail) at tests considered more or less trivial to many other CFD methods.

When first starting to work on Regularized SPH (RSPH) in 2000, the development was based on three simple observations: (1) SPH has its stronger and weaker sides like any other CFD method. Depending on the application in question, the specific properties of SPH will make the method more or less suitable as a numerical tool. (2) Conventional SPH is not particularly flexible or user-friendly when it comes how fluid is discretized. Particle distributions (and spatial resolution) can only be chosen at the beginning of a simulation. After this, the fluid flow determines not only how the solution evolves but also how this solution is represented. In other words, the evolution of the particle distribution is controlled solely by the fluid flow itself. (3) More work could be done to overcome fundamental limitations of conventional SPH.

RSPH was therefore developed as an extension to SPH [4], [6]. What distinguishes RSPH from the more standard method, is the flexibility with which the resolution can be made adaptive and the regularization technique introduced to prevent the particle distribution from becoming too irregular. In contrast to standard SPH, RSPH is not restricted to resolution being a function of the initial particle distribution and the subsequent, time-dependent flow pattern. RSPH is able to maintain high resolution in regions of interest, for example near shock wave structures, and low resolution elsewhere [1]. The aim has all along been to achieve this flexibility without comprising the nice properties of the original method when dealing with complex free surfaces or interfaces. The first attempts at applying RSPH to free-surface flows were reported already in 2003 [3]. But as discussed in [2], both free surfaces and curved boundaries have remained a challenge to RSPH up to now since newly generated particles always were placed in Cartesian grid configurations.

Another issue so far preventing RSPH from being applied to a wider range of problems, is the increased code complexity of RSPH relative to conventional SPH. In previous versions of RSPH, the smoothing length (h) profile is piecewise constant with steps in h representing a factor of 2 change in the h-value. Quantized h-values made it easier to construct global h-profiles and the corresponding particle distributions. However, the SPH summations do not handle steps in h very accurately. Therefore, alternatives to SPH summations had to be found in regions sufficiently near steps in h. In first generation RSPH, we used a technique referred to as **cell summation** [6], while in second generation RSPH we have been using MLS-type interpolation in combination with an additional type of particles called **auxiliary particles** [4].

As a result, the RSPH-specific features of the method could not be easily separated from the more conventional features. Anyone wanting to apply the RSPH method to a problem would have to write a new code more or less from scratch, even if they already had a well-functioning SPH code. In the third generation RSPH method, h-profiles are roughly piecewise linear instead of piecewise constant, and as a consequence, standard SPH summations work reasonably well. The RSPH-specific features can now in principle be fully separated from the SPH-specific features. And the aim is in future to develop a separate regularization module which could be integrated seamlessly into any existing SPH code.

This paper is organized as follows: Section II describes how

Discrete differential operators for Voronoi particle dynamics

Daniel Duque
Dep. Enseñanzas Básicas Ing. Naval
E.T. S. Ing. Navales, UPM
28040 – Madrid, Spain
daniel.duque@upm.es

Pep Español
Dep. Física Fundamental
Facultad de Ciencias, UNED
28040 – Madrid, Spain

***Abstract*—SPH can be viewed as an method to provide differential operators from a discrete set of spatial points: the particles. These discrete operators, when introduced in the laws of hydrodynamics, provide the laws of motion according to which the particles move. A particularly compact approach to SPH proceeds from the definition of the volume of a particle. From this concept, differential operators may be obtained.**

Other, alternative, particle dynamics can be obtained by considering other choices for the volume. A natural choice is the Voronoi cell volume: the Voronoi cell for a given particle i is the region in space that is closer to i than to any of the other particles. This way, a sort of *Voronoi particle dynamics* results. We discuss here some features of this alternative theory and its associated differential operators, focusing on the Laplacian. A connection to the discrete Laplacian commonly found in the Finite Element Method (FEM) is discussed, as well as other alternatives. Our main conclusion is that, despite its flaws, the FEM-like proposal is an attractive candidate for fluid simulations. We also identify a very promising theory that could be superior, the second-order maximum-entropy method.

I. INTRODUCTION

The Navier-Stokes equations of hydrodynamics can be cast in "particle" form by introducing the Jacobian of the transformation from Eulerian coordinates to Lagrangian one. Its determinant, V, satisfies:

$$\frac{dV}{dt} = V \operatorname{div} \vec{v},$$

and represents the particle volume. Defining particle mass $m = \rho V$ and momentum $\vec{P} = mv$, the Navier-Stokes equations can be cast in a form in which convection terms are absent. The particular formulation of hydrodynamics that we consider here is:

$$\frac{d\vec{R}}{dt} = \vec{v} \tag{1}$$

$$\frac{dm}{dt} = 0 \tag{2}$$

$$\frac{d\vec{P}}{dt} = -\operatorname{grad} p + \nu \Delta \vec{v} \tag{3}$$

These equations are "simulation-ready", since they can be readily interpreted as the laws of motion for a set of particles that move following the flow (1), with invariant mass (2), whose momentum changes according to (3). (One only needs to write an i subscript in each of the objects.) In order to close the system, we consider the additional equation of state (EOS):

$$p = \frac{\rho_0 c^2}{\gamma} \left[\left(\frac{\rho}{\rho_0} \right)^{\gamma} - 1 \right]. \tag{4}$$

The density of a particle is obtained from $m = \rho V$, once the volume is defined. This is an alternative to the more standard route of solving the continuity equation [1]. Indeed, the SPH method (at least, versions thereof) can be derived [2] from the definition of a particle volume for particle i as

$$(V_i)^{-1} = \rho_i / m_i = \sum_j W(\vec{r}_j - \vec{r}_i). \tag{5}$$

From this definition, a number of expressions follow that provide the necessary differential operators to be used in the momentum equation (3), namely the gradient operator (for the pressure forces) and the Laplace operator (for viscous forces). We will not go into details of standard SPH formulation of these operators. Instead, we discuss her another possible choice for the particle volume: the Voronoi volume.

The Voronoi volume of a particle is the volume of the Voronoi cell surrounding the particle. Geometrically, a particle's cell is the region of space that is closer to the particle than to any other particle. We show in Fig. 1 a collection of particles with their cells (called a Voronoi diagram), coloured red. This configuration corresponds to a zoom of the equilibrium configuration in Fig. 6, as we will discuss later.

Notice a Voronoi diagram introduces a useful concept of neighbourhood in a disordered network: two particles are neighbours if their cells are. This definition of neighbourhood is of course related with the distance between particles, but is independent (i.e. two particles may be close-by and still not be neighbours; the opposite may also be true). An important geometrical construction is the Delaunay triangulation, formed by joining neighbouring particles with segments. This triangulation has very interesting features, as we will mention. Mathematically, this network is the dual of the Voronoi diagram. (We are also neglecting in this discussion possible degenerate cases.) In Fig. 1 we also draw the Delaunay triangulation, in blue.

Convergence of the finite volume particle method for viscous flow

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 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). In this work, a brief description of the FVPM model for viscous slightly compressible fluid is provided. The presented study addresses convergence and effectivity of the FVPM simulations for a two-dimensional unbounded viscous flow. The method's sensitivity to some setup parameters is discussed. The main attention is paid to the role of the particle kernel function and to the evaluation of interparticle vectors with respect to further optimisation of the method.**

I. Introduction

The finite volume particle method (FVPM) is a recently developed computational tool capable of solving complex problems of fluid dynamics, and continuum mechanics in general, without the need for a computational mesh. The fundamentals of the FVPM method were derived by Hietel et al. [3] and extended by several authors during the past decade, e.g. [5], [6], [10]. It belongs to the family of particle methods, such as smoothed particle hydrodynamics (SPH), while its formulation is derived from the integral conservative form of the governing equations. In FVPM, computational domain is discretised with a finite number of computational particles which are represented by smooth overlapping test functions. The particle interaction is realised through inter-particle fluxes between neighbouring particles. In a special case where neighbouring particles do not overlap but continuously fill in the computational domain, the method is formally identical to conventional mesh-based finite volume method.

Unlike the standard SPH method, which is purely Lagrangian, in FVPM the particle velocity does not necessarily equal the fluid velocity, i.e. the FVPM particles can be defined either in Lagrangian, Eulerian or Arbitrary Lagrangian-Eulerian coordinate frames. The FVPM is conservative and enables straightforward exact treatment of boundary conditions via definition of appropriate boundary fluxes. Major drawbacks of the method in comparison to weakly compressible SPH are significantly higher computational demands due to numerical integration of interparticle fluxes and difficult implementation of free surface boundary conditions. An attempt to overcome both of these issues by applying special smoothing functions was suggested recently in [9].

The present work is focused on the characterisation of the error behaviour in FVPM simulations, method's convergence and overall sensitivity to the simulation setup. Although the convergence of the method has been investigated in some prior studies, [7], [11], the current results present a more comprehensive evaluation of method's performance for viscous flows.

In the following, the FVPM formulation for weakly compressible viscous fluid is compared with the theoretical solution of Taylor-Green vortex flow in two-dimensional space. A number of simulation series has been carried out and the error in numerical solution has been evaluated as a function of both the initial particle spacing and the particle smoothing length.

II. Method

A flow of viscous fluid can be described by 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 γ equals 7, [1].

The conservative 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 particle at position $\mathbf{x}_i$ and smoothing length h analogous to kernel functions used in SPH. The radius of kernel support domain is defined by a multiple of the smoothing length λh (e.g. λ equals 2 for the quadratic kernel). Similarly to SPH method,

A soft-tissue model coupled with fluid dynamics using SPH

S. Adami, X.Y. Hu and N.A. Adams
Institute of Aerodynamics
Technische Universität München
Garching, Germany
xiangyu.hu@tum.de

***Abstract*—In this work, we show that the fluid-structure interaction (FSI) problem, which includes the modeling of fluid dynamics, soft-tissue dynamics and fully conservative coupling, can be formulated within the monolithic frame work of smoothed particle hydrodynamics (SPH) method. In order to implement the strain-stress relation, an extra evolution equation is introduced according to the soft-tissue models. Two material models, one is the linear elastic model and the other is the non-linear deformation gradient model, have been implemented. Due to the usage of particle modeling for both fluid and structure, this formulation is able to handle large deformation of the fluid-structure interface.**

I. Introduction

Fluid-structure interactions (FSI) are a crucial consideration of many engineering and biological systems. Generally, for numerical simulating FSI problems, there are two types of FSI approaches: one is called monolithic (fully coupling) approach [1], [2]; the other is called partitioned coupling approach [3], [4]. The monolithic approach usually works with matching meshes and with single generalized solvers, such as Arbitrary Lagrangian Eulerian (ALE) coupling with re-meshing technique, which models the fluid with re-meshed Eulerian finite-volume grid and the structure with Lagrangian finite-element mesh. This approach is usually fully conservative and quite robust, but has poor application flexibility. One common difficulty is how to handle the large deformation of the fluid-structure interface. Alternatively, the partitioned coupling approach works on non-matching meshes and generally different solvers, by which the coupling is through the implementation of the boundary conditions. Since non-matching meshes are used, though quite flexible, this coupling sacrifices the conservation across the fluid-structure interface.

In this work, we consider a type of particle method, smoothed particle hydrodynamics (SPH), for simulating FSI problems. SPH is grid-free, Lagrangian method which have been applied to a wide range of interesting problems ranging from astrophysics to fluid dynamics, see [5]. Here, we show that the monolithic frame work of SPH can achieve fully conservative coupling and have sufficient flexibility on handling large deformation of fluid-structure interface. Two material models, one is the linear elastic model and the other is the non-linear deformation gradient model, have been implemented. The present soft-tissue model is similar to the particle level-set technique of Hieber et al. [6], who simulated biological tissue with a Lagrangian particle method. Although the key aspects of the non-linear tissue model are equivalent, their method includes a re-meshing procedure and the interface is defined with level sets. In contrast, we use a standard SPH formulation without the need of remeshing, which simplifies the algorithm noticeable.

II. Governing equations

The continuity equation and the acceleration equation in its general form for a particle i are given by

$$\frac{\mathrm{d}\rho}{\mathrm{d}t} = -\rho\nabla\cdot\mathbf{u} \tag{1}$$

and

$$\frac{\mathrm{d}\mathbf{u}}{\mathrm{d}t} = \mathbf{g} + \frac{1}{\rho}\nabla\cdot\sigma = \mathbf{g} + \frac{1}{\rho}\nabla\cdot\left(-p\mathbf{I}+\mathbf{S}\right)\ , \tag{2}$$

where ρ, $\mathbf{u}$, $\mathbf{g}$ and p denote the density, the velocity, the gravitational acceleration and the pressure of a computational element. The tensor σ denotes the symmetric Cauchy stress tensor and $\mathbf{S}$ can be the deviatoric stress of a solid or tissue particle or the viscous stress of a fluid particle.

We discretize Eq. 2 in its general form because that automatically offers the ability to simulate FSI problems. After calculating the stress tensor for each particle based on the appropriate fluid and structure constitutive relations, the force calculation for fluid dynamics, structure dynamics and fluid-structure coupling are straightforward without the need of special interactions.

A. Elastic solid model

Before we introduce the non-linear soft tissue model, we want to recall the basic equations of an elastic solid model.

Assuming Hooke's law with the shear modulus μ, the rate of change of the deviatoric stress tensor $\mathbf{S}$ can be obtained from

$$\frac{\mathrm{d}S_{ij}}{\mathrm{d}t} = 2\mu\left(\dot{\epsilon_{ij}} - \frac{1}{3}\delta_{ij}\dot{\epsilon_{kk}}\right) + S_{ij}\Omega_{jk} + \Omega_{ik}S_{kj} \tag{3}$$

with the strain rate

$$\dot{\epsilon_{ij}} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{4}$$

SPH simulations of advective diffusion phenomena induced by pollutants in water

F. Aristodemo, I. Federico, P. Veltri
Dipartimento di Difesa del Suolo
Università della Calabria
Arcavacata di Rende (CS), Italy
aristodemo@dds.unical.it, federico@dds.unical.it, paolo.veltri@unical.it

A. Panizzo
Business Development
Enel Green Power
Roma, Italy
andrea.panizzo@enel.com

***Abstract*—Environmental engineering problems concerning diffusion and natural advection phenomena occur in the presence of a pollutant in water. The present work deals with a 2D SPH modelling for the evaluation of the concentration field of pollutants in still water. A SPH model is formulated to solve the fickian diffusion equation applied to pollutants with the same density as the water. A lagrangian SPH formalism of the advective diffusion equation is also developed for pollutant-water, taking into account the effects of molecular diffusion and natural advection induced by differences between the fluid densities. These equations are coupled with the fluid mechanics equations. Numerical tests referring to a strip and a bubble of contaminant in a water tank with different initial concentration laws have been also carried out.**

Numerical tests referring to a strip and a bubble of contaminant having different densities in a water tank with initial constant and exponential concentration laws have been carried out. The results obtained by the proposed SPH models are compared with other available SPH formulations for the cases of fickian diffusion, showing an overall better agreement with standard analytical solutions in terms of spatial evolution of the concentration values. Capabilities and limits of the proposed SPH models to simulate advective diffusion phenomena for pollutant-water density ratios ρ_2/ρ_1 ranging from 0.1 to 1 are discussed.

I. Introduction

Pollutant transport in still water is generally dominated by the effect of diffusion and advection processes. For pollutants having densities close to the water and initial velocity equal to zero, the evaluation of the concentration field is induced by diffusion. This phenomenon is characterized by the scattering of particles by random molecular motions, which may be described by Fick's law. If the contaminant density is quite different from the water, the phenomena of natural advection become relevant. The transport process is consequently associated to the induced kinematics. In rivers a host of processes lead to a non-uniform velocity field, which allows mixing to occur much faster than by molecular diffusion alone [1]. In this case the turbulent diffusion processes assume a fundamental role in modelling the concentration field.

The modelling of diffusion processes by the SPH technique was first investigated by Cleary and Monaghan [2] referring to heat conduction problems. Other SPH modellings have been formulated to simulate diffusion phenomena in a spatially periodic porous media [3] [4], for miscible flow in fractures [5], and coupled with reactive transport and precipitation of a solute [6]. The use of the SPH method in the analysis of two-phase flows for advective diffusion processes has not been investigated. More recently a SPH diffusion modelling in the presence of a velocity field has been addressed to the simulation of the interaction between water and non-cohesive bed sediments [7].

A Smoothed Particle Hydrodynamics modelling for evaluating the concentration field of pollutants in a water tank is developed here. The present paper proposes a lagrangian model of the diffusion equation for contaminants and water when the process is dominated by molecular diffusion. Moreover a SPH advective diffusion equation for contaminant-water in the presence of a significant velocity field is formulated. The use of this advective diffusion model allows the spatial evolution of the concentration to be determined and takes into account the coupled effects of fickian diffusion and natural advection induced by the different densities of fluids in still water. These diffusion and advective diffusion models are coupled with the fluid mechanics equations rewritten in the SPH approach [8].

In the following sections the implementation of diffusion and advective diffusion SPH modelling is illustrated. Some environmental engineering problems, referring to the evolution of a strip and a bubble of contaminant in still water, are analysed. Comparisons of the proposed SPH model with other lagrangian approaches and analytical solutions are reported, showing the evolution of the concentration field at significant time steps. Capabilities and limits of the present numerical approaches for a wide range of fluid density ratios are investigated.

II. Diffusion equation

Diffusion phenomena of pollutants are described by Fick's law. Fick extended the Fourier heat transfer equation to the analysis of diffusion processes. This law states the proportionality between the mass flux diffusion, J, and the concentration gradient, ∇C. In the absence of advective phenomena, this hypothesis coupled with the mass conservation leads to the classical diffusion equation:

Simulation of Surface Tension by SPH Method and Its Applications

Mingyu Zhang, Shudao Zhang
Institute of Applied Physics and Computational Physics, Beijing, China, 100094
Zhang_mingyu@iapcm.ac.cn

***Abstract*—The methods for simulating surface tension with smoothed particle hydrodynamics (SPH) method in two dimensions and three dimensions are developed. The SPH particle on the boundary is detected dynamically. The boundary curve in 2D is reconstructed locally with Lagrangian interpolation polynomial. The boundary surface in 3D is reconstructed locally with moving least squares (MLS) method. The norm vector and the curvature of the boundary surface are calculated according to the reconstructed boundary surface and then the surface tension force can be calculated. Numerical simulation are investigated to validate the current method.**

I. INTRODUCTION

Surface tension has a significant influence on the fluid flow with free surface or multiphase flow with a sharp interface, such as a breaking dam, capillarity, binary coalescence.

SPH method has been applied to simulate the fluid flow with free surface. Nugent[1] first developed van der Waals (vdW) model which gives rise to an attractive, central force between the particles with an interaction range. Meleán[2, 3] and López[4] applied the vdW model to simulate the oscillation of the deformed drop without tensile instability, the head-on and off-center binary collision. Colagrossi[5] present a SPH formulation for two-dimensional interfacial flows with the low density ratios. Tartakovsky[6, 7] presented a surface tension model to simulate unsaturated flow through fracture junctions, the classical two-dimensional Rayleigh–Taylor instability and three-dimensional miscible flow in fracture apertures with complex geometries. Morris[8] developed the SPH formulation for the CSF model, in which the interface is tracked implicitly by simulating the advection of the color function. All the above methods give very good results. However, the surface tension force is not set explicitly. And there are artificial coefficients in these methods and the choice of the artificial coefficient is arbitrary. And also the initial temperature of the fluid has to be chosen specifically for the vdW model. It undermines the application of the vdW model to the free surface problem related to the heat transfer. Hu and adams [9-11] presented an CSF model for the SPH method and applied it into the incompressible multi-phase flow such as droplet oscillation, contact angle, droplet deformation in shear flow, Rayleigh-Taylor instability.

In this paper, the methods for simulating surface tension with SPH method in two dimensions and three dimensions are developed. The curvature can be calculated accurately according to the reconstructed interface and then the surface tension force can be obtained explicitly.

II. METHOD

The SPH particles on the interface are detected dynamically. Based on the detected boundary particle, the interface is reconstructed locally by Lagrangian interpolation polynomial in 2D and moving least squares method in 3D. Then the curvature and the norm vector of the interface can be calculated accuratelly. The surface tension force is obtained.

Before the reconstruction of the interface, the coordinate system has to be transformed. The local coordinate origin is the average point of the neighbors of the boundary particle. Then the interface is reconstructed locally. The advantage of the transformation of the coordinate system is that the reconstructed interface in the local coordinate system is guaranteed to be one valued.

The curvature is independent on the coordinate system. Since the norm vector is dependent on the coordinate system, the norm vector has to be transformed back to the global coordinate system.

Through the above steps, the interface is reconstructed and the curvature and the norm vector of the interface can be calculated accurately. The surface tension force is the product of the surface tension coefficient, the curvature and the norm vector of the interface. The surface tension force can be obtained accurately according the above method.

III. NUMERICAL SIMULATIONS

To validate the current surface tension model, the following numerical simulations are given. The cubic kernel is used in the simulations. The smoothing length is kept constant.

A. Oscillation of 2D square liquid drop

Deformation of 2D liquid drop from initial shape of square to circle is simulated. The side length of the square is 7.5×10^{-3} and 900 SPH particles are used in the simulation. The coefficient of surface tension is set to 10. The viscosity of the liquid is 1.0×10^{-6}. Figure 1 shows the snapshots of the oscillation of 2D liquid drop with initial shape of square in sequence. Due to the surface tension force, the 2D liquid drop evolves into the flower, non-equilibrium circle, diamond, round-corner square, final circle in equilibrium. During the oscillation of 2D liquid drop, the kinetic energy dissipates due to the liquid viscosity.

A Study of the Matter of SPH Application to Saturated Soil Problems

Ha H. Bui, R. Fukagawa
Department of Civil Engineering
Ritsumeikan University
Shiga, Japan
hhbui@fc.ritsumei.ac.jp; fukagawa@se.ritsumei.ac.jp

K. Sako
Global Innovative Research Organization (GIRO)
Ritsumeikan University
Shiga, Japan
kaz-sako@fc.ritsumei.ac.jp

***Abstract*—We present an application of SPH to saturated soil problems. Herein, the standard SPH formulation was improved to model saturated soil. It is shown that the proposed formulation could yield several advantages such as: it takes into account the pore-water pressure in an accurate manner, it automatically satisfies the dynamics boundary conditions between submerged soil and water, and it reduced the computational cost. Discussions on the use of the standard and the new SPH formulations are also given through some numerical tests. Furthermore, some techniques to obtained correct SPH solution are also proposed and discussed. To the end, this paper suggests that the proposed SPH formulation should be considered as the basic formulation for further developments of SPH for soil-water couple problems.**

I. INTRODUCTION

The finite element method (FEM) has been employed as the standard numerical approach in computational geomechanics. However, most of problems in geotechnical engineering often involved the large deformation and post-failure problems such as: post-failure process of a sliding slope, debris flow in landslide, seepage failure, post-failure of a slope due to soil liquefactions, etc. In such circumstances, FEM suffers several disadvantages due to mesh tangling even when the updated Lagrangian method is adopted. Re-meshing may help to resolve this problem but the procedure is quite complicated. As an alternative for such computational purposes, it is attractive to develop mesh-free methods. So far, the most popular mesh-free method in geotechnical engineering is the discrete element method (DEM) which tracks motion of a large number of particles, with interparticle contacts modeled by spring and dashpot systems [1]. The main advantages of this approach are that it can handle large deformation and failure problems; and the concept is relatively simple and easy to implement in a computer code. However, DEM suffers from low accuracy since suitable parameters for the contact model are difficult to determine. The discontinuous deformation analysis (DDA) method [2] has also been applied in geotechnical applications, but is mainly used for rock engineering, etc. On the other hand, the method of smoothed particle hydrodynamics (SPH) [3-4] has been recently developed for solving large deformation and post-failure flows of geomaterials [5-13], and represents a powerful way to understand and quantify the failure mechanisms of soil in such challenging problems.

When solving the two-phase water saturated soil, it is the common approach in computational geomechanics to treat the two-phase system as a single phase and the interaction between soil and water was considered via the contribution of the pore-water pressure using the Terzaghi's effective stress concept. Pastor et al. [10] employed this approach in his SPH model to take into account contribution of the pore-water pressure in the landslide simulation. The gradient of pore-water pressure in his model has been approximated using the conventional SPH formulation. However, our current research on the application of SPH to saturated soil revealed that such the approximation of the gradient of the pore-water pressure will lead to numerical instability problem which may failure the SPH computational process for cases when soil is completely submerged into water. Therefore, it is necessary to overcome this limitation in order to generalize the SPH applications to computational geomechanics. In this paper, we will firstly demonstrate the numerical instability problem caused by using the conventional SPH formulation. Then, we will derive a general SPH formulation which can be applied to both dry and saturated soils. Finally, we will show some advantages of the propose SPH formulation. Technique to obtain the initial stress condition of soil in SPH is also proposed by adding a damping force into the motion equation. Several numerical testes are performed to validate the proposed formulation.

II. SMOOTHED PARTICLE HYDRODYNAMICS

A. Standard SPH Formulations

In SPH, approximations for quantities of a continuum field such as density, velocity, pressure, etc., are performed using the following interpolation function,

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

where A is any variables defined on the spatial coordinate $\mathbf{r}$, and W is smoothing kernel, which is chosen herein to be the cubic-spline function [14],

Internal mechanical response of a tethered DNA in shear flow

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

***Abstract*—Tethered DNA dynamics in shear flow is simulated using Smoothed Dissipative Particles Dynamics (SDPD): a mesoscopic meshless method with explicit solvent modeling. SDPD is a thermodynamically consistent version of Smoothed Particle Hydrodynamics (SPH) able to discretize the Navier-Stokes equations and to incorporate thermal fluctuations according to the fluctuation-dissipation theorem. The advantage of the method is applicability to the cases involving complex flow. The microscopic model of the polymer molecule is a linear chain composed of several SDPD particles interacting via a finitely expendable nonlinear elastic spring. Results of the simulations are in good agreement with experimental data (mean fractional extension, distance from the wall, polymer orientation angle). By examining the cross-correlation function and power spectral density of the chain extensions in the flow and gradient directions, the existence of periodicity of the cyclic dynamics is studied.**

I. Introduction

Tethered polymers find many uses in various biological and engineering systems, such as DNA microarrays and biocompatible surfaces. Some recent works aim to study dynamics of tethered DNA in vivo [1]. Increasing complexity of systems under consideration gives rise to computer simulations as an important investigation tool.

Several experimental and numerical studies exist on the static properties of a tethered polymer in the flow [2]–[5]. However, dynamics of tethered polymer in shear flow is not fully understood. In the following paragraphs we review the literature on tethered polymer dynamics.

Fluorescence microscopy experiments by Doyle et al. [6] reveal the anomalous dynamics of shear enhanced temporal fluctuation of chain extension. The amplitude of the fluctuations has a well-defined maximum at moderate shear rate. To understand the phenomena, Doyle et al. performed Brownian dynamics simulations of a bead-spring chain with worm-like elasticity and found the average parameters are in a good agreement with the experimental results. Simulations suggest that the polymer displays a continuous recirculating motion or *cyclic dynamics*. The authors proposed the following mechanism: typically the chain is compressed and stays close to the wall (1) thermal fluctuation drives the chain away from the surfaces in the region of stronger flow, (2) the chain stretches (3) free end rotates towards the interface (4) such a configuration leads to a compression and the chain takes the original configuration. Concerning time scale of the *cyclic dynamics* Doyle et al. report that stage (2) is faster than stage (3) and cycling time is broadly distributed.

Further evidence for existence of *cyclic dynamics* comes from the studies of the polymer brush. The phenomena of flow inversion at the surface can be explained by the effect of cyclic motion of the individual polymers [7].

Brownian dynamics simulations by Schroeder et. al. [8] reveal peaks in power special density (PSD) of orientation angle for tethered DNA in the range $1 < \mathrm{Wi} < 10$ (Wi is the Weissenberg number, ratio of the relaxation time of the flow and the polymer relaxation time). The peak indicates the presence of periodic cyclic motion. The dependence of peaks position on shear rate in simulations is in good agreement with prediction based on scaling arguments.

Using cross-grain molecular dynamics simulations Gratton and Slater [9] show that *cyclic dynamics* pronounce as a peak in extension – orientation angle cross-correlation function. However, the characteristic time of the peaks in cross-correlation functions is bigger than time reported by other authors and the polymer in the simulations exhibits a *sticking* behavior. Authors attribute such behavior to atomistic effect in the near-wall region which cannot be seen in realistic systems.

In analysis of multilayer Makrkov chains as a model for tethered polymer [10] several distinct regions of high probability of chain end was found. This suggests an explanation for large fluctuation in chain extension: chain end jumps between these regions.

Molecular and Brownian dynamics study of Delgado-Buscalioni [11] lends strong support to the presence of *cyclic dynamics*. In this work a well-defined periodic motion is seen in the cross spectra of extension – distance from the wall. The cycling time scale is found to be about 10 times that of the polymer relaxation time.

Recent experimental work [12] reports for the first time observation of DNA in the flow gradient direction and provides a first direct experimental evidence of the *cyclic dynamics* by measuring PSD of polymer geometrical quantities.

Some reports question existence of *cyclic motion*. Donev and co-workers [13], [14] used a Stochastic Event-Driven Molecular Dynamics (SEDMD), lattice Boltzmann and Brownian dynamics methods and found no evidence of the *cyclic motion* reported in previous works.

In this work we present detailed smoothed dissipative particles dynamics (SDPD) simulations of tethered DNA with implicit solvent modeling. Results are compared with recent experimental data. Quantitative evidence for the existence

SPH-FEM coupling to simulate Fluid-Structure Interactions with complex free-surface flows

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

G. Oger
HydrOcean
Nantes, France

***Abstract*— The Smoothed Particle Hydrodynamics (SPH) method presents different key assets for modelling violent Fluid-Structure Interactions (FSI). First, the meshless nature of this method drastically reduces the complexity of handling the fluid-structure interface when using SPH to model the fluid and coupling it with a Finite Element Model (FEM) for the solid. Second, the method being Lagrangian, large deformations of the fluid domain can be followed, which is especially interesting for simulating violent interactions in presence of a free surface, or which induce large deformations, rotations, and translations of the solid. Third, the SPH method being explicit, the time scale of the SPH resolution in the fluid domain is naturally adapted to the FEM resolution in the solid where small timesteps are commonly used. Free-surface FSIs can also be simulated without including the air phase when it does not play a significant role. For violent interactions where the fluid compressibility matters, it is also intrinsically modelled by the SPH method. The paper details the SPH method used and the coupling. The FEM solver is a standard open source solver for solid mechanics. A validation test case that deals with the hydrodynamic impact of an elastic wedge at high speed is then presented in detail; local pressures and wedge deformations are compared to analytical data.**

I. INTRODUCTION

Nowadays, thanks to the growth of computing power numerical simulation is widely used to study complex physical problems involving fluids or solids. The modeling of coupled phenomena such as fluid-structure interactions can be carried out as well even though their simulation needs important numerical resources. In literature different methods have been developed in order to predict the deformations of a structure due to non-stationary loads caused by fluid flow, itself modified by the structure evolution. The complexity of the coupling and its simulation is much increased when both the fluid and the structure are largely deformed due to their interaction. We concentrate here on these complex fluid-structure interactions.

In a general way Lagrangian formulations based on the Finite Element Method (FEM) are employed to study the structure behaviour. The mesh follows solid deformations. However, different methods which allow following the moving interface between fluid and solid are used to simulate the fluid evolution when dealing with fluid-structure interaction problems. We can cite mesh-based methods such as Lagrangian or Arbitrary Lagrangian Eulerian (ALE) formulations of Navier-Stokes equations, often using the Finite Volume Method (FVM) or FEM [1], where the fluid mesh is deformed to adapt to the solid domain deformation. This adaptation is costly and of complex implementation. Further, large fluid domain deformations cannot be handled without using remeshing methods which significantly increase CPU costs of simulations. Another way to solve this problem is to use Volume of Fluid method (VOF) or Level-Set methods which permit to track interfaces on fixed grids [2]. With all these methods, a contact algorithm is used in order to prevent materials interpenetration. Fluid structure simulations have also been performed using the Boundary Element Method (BEM) [3]. In the latter case, only boundaries need to be meshed. However, this is limited to simplified problems in terms of geometry (of the free surface especially) and physics (no vorticity nor viscosity).

Particle-based methods have also been used to model fluid evolution in the context of Fluid-Structure Interactions. Is for instance the case with the Material Point Method in [4], or with the Particle Finite Element Method (PFEM) in [5]. However these methods are not fully meshless.

The method proposed here to simulate fluid-structure interactions relies on the Smoothed Particle Hydrodynamics method (SPH) [7][8][9] resolution of the fluid problem. The latter method being Lagrangian and meshless, no free surface tracking techniques and contact algorithm between fluid and structure domains are needed. In addition, compressibility effects are taken into account when they matter. In this paper, we first present an introduction to the SPH formalism. Then, the way we couple this method to a standard FEM solver for the solid is described. Validation is made on the hydrodynamic impact of an elastic wedge in comparison to the analytical solution. An illustrative test case is then presented where a water column impacts an elastic plate resulting on a strong interaction with a very complex flow pattern.

II. SPH SOLVER

In 1977, the SPH method was proposed by Monaghan, Gingold and Lucy [10][11]. The aim was to simulate complex astrophysical phenomena such as star formations. Later, Monaghan used this method to simulate free surface flows [12].

Numerical predictions of ship flooding scenarios using SPH

A. Marsh, C. Khaddaj-Mallat, D. LeTouzé
Fluid Mechanics Laboratory
Ecole Centrale de Nantes
Nantes, France

G. Oger
HydrOcean
Nantes, France

Flooding of a ship's deck (greenwater) or within its internal compartments can severely restrict the operational ability of the vessel, and the safety of its cargo. In severe circumstances such as those produced by freak waves or hull damage, the vessel can become unstable causing it to sink and/or capsize. The flows produced by such events tend to be highly dynamic, with large amounts of free surface deformation. For this reason, SPH is a valuable method for predicting the physics of such flows.

In this paper, SPH is used to predict fluid behaviour for two different flooding scenarios. The first is the interaction between a vessel (represented by a rigid body) and undulating travelling waves. The predicted water heights on the deck are compared to experimental results in [1]. The second is the transient flooding behaviour that occurs during, and immediately after a side collision between two vessels. Water heights are measured close to the point of impact within the vessel. The measurements are compared to experimental results in [2].

I. INTRODUCTION

Large scale flooding of a ship's deck or internal compartments can restrict its operational ability, subjecting its passengers and cargo to risk. Such flooding can be caused by large waves in extreme oceanic conditions, vessel-vessel impacts, or grounding. The work in this paper concentrates on the numerical simulation of two such scenarios using the Smoothed Particle Hydrodynamics code SPH-Flow.

The first scenario presented, concerns the interaction between regular travelling waves and an FPSO (Floating Production Storage and Offloading) vessel, commonly used for gas and oil transportation. The interaction causes flooding of the external deck (greenwater) on the bow of the vessel. Numerical analysis is undertaken in two and three dimensions; water heights on the deck are recorded for both cases, and compared to the experimental results in [1].

The second scenario concerns floodwater entering a damaged Ro-Ro ferry mid-section during, and immediately after a vessel-vessel side impact. Such a study is of interest due to the Ro-Ro vessel's susceptibility to sinking and/or capsizing in the event of internal flooding. This is due to its design, consisting of long vehicle decks separated by few external bulkheads. In the event of a side impact, the sloshing effects caused by the floodwater produce significant forces and roll moments, resulting in instability [2-5]. Significant experimental work has been completed to quantify these [2,4]. However, the physical events responsible for producing peaks in these indicators remain difficult to identify experimentally. For this reason, a numerical method capable of accurately predicting such events would be a valuable tool for providing further insight into the physics of flooding. Numerical analysis of the transient flooding behaviour in and around the ferry mid-section is undertaken in three dimensions. Water heights are recorded within the mid-section at the point of impact. These predictions are then compared to the experimental results in [2].

II. NUMERICAL MODEL

A parellelised SPH code called SPH-Flow is used in this study. Ecole Centrale de Nantes has developed this code, in collaboration with HydrOcean (see authorship). The method of parallelisation used is MPI. The domain is split into sub-domains; the size of each sub-domain is adapted so that they have approximately the same number of particles. One core is then assigned for each sub-domain. The code implements a renormalisation of kernel method, and uses an exact Riemann solver with Gudonov numerical scheme [6]. The use of a Riemann solver eliminates the need for artificial viscosity (present in the standard SPH formulation) to reduce numerical instability and dissipation [7]. Boundaries are represented by ghost particles. SPH-Flow has been successfully applied to various hydrodynamics applications [6,8].

SPH Simulations of Fish-like Swimmers

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

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

***Abstract*—Recent results from simulations of fish-like swimming linked bodies using SPH will be presented. The linked bodies, which are typically ellipses, move through a fluid due only to specified changes in the angles between the bodies. By careful choice of the relative sizes of the bodies and their gait, one can model aquatic creatures that use carangiform locomotion (such as mackerels), or anguilliform locomotion (such as lamprey eels). Results from two-dimensional simulations are presented for: (a) swimmers with and without skin covering the gaps between the bodies; (b) swimmers with arbitrary body shape and size; (c) swimmers moving with forward and turning gaits; and (d) scaling relations and optimal speeds. The implementation of the skin, gait specifications, and boundary forces will be described.**

I. Introduction

In this paper, recent results from two-dimensional SPH simulations of rigid linked bodies moving in a weakly compressible, viscous fluid are given. The work is closely related with mathematical and computational studies of swimming fish, and with the motion of underwater vehicles, all of which are propelled by changes of shape. The simulations are most similar to that of Kanso et al. [1] and Eldredege [2]. Kanso et al. consider an infinite, two-dimensional inviscid fluid, and Eldredge considers a viscous fluid modelled using the vortex particle method. Our approach is capable of handling arbitrary body shapes and boundaries.

In the following sections, the governing SPH equations, as well as the equations for the rigid bodies, constraints, gait, and skin are given. After comparing the results of the SPH simulations to those of Kanso et al. and Eldredge, the results from recent studies are presented. Recent work has focussed on the inclusion of skin (originally the rigid bodies were connected by virtual rods), the determination of scaling relations that relate the speed and power expenditure to the gait parameters, and the determination of optimal motion.

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 [3].

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 σ is used for the skin particles. The equation of motion for fluid particle a is given by

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

where

$$\mathbf{F}_a(\mathit{fluid}) = -\sum_b m_b \left(\frac{P_a}{\rho_a^2} + \frac{P_b}{\rho_b^2} + \Pi_{ab} \right) \nabla_a W_{ab}, \tag{2}$$

$$\mathbf{F}_a(\mathit{body}) = -\sum_j m_j \left(\frac{P_a}{\rho_a^2} + \frac{P_j}{\rho_j^2} + \Pi_{aj} \right) \nabla_a W_{aj} + \sum_{k=1}^{N_b} \sum_{j \in B(k)} m_j \mathbf{r}_{aj} f(|\mathbf{r}_{aj}|), \tag{3}$$

$$\mathbf{F}_a(\mathit{skin}) = -\sum_\sigma m_j \left(\frac{P_a}{\rho_a^2} + \frac{P_\sigma}{\rho_\sigma^2} + \Pi_{a\sigma} \right) \nabla_a W_{a\sigma} + \sum_{k=1}^{N_s} \sum_{\sigma \in S(k)} m_\sigma \mathbf{r}_{a\sigma} f(|\mathbf{r}_{a\sigma}|). \tag{4}$$

$\mathbf{F}_a(\mathit{fluid})$ 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 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 second is based on a formulation of the effects of boundaries due to Sirovich [4], [5]. Our prescription for this is similar to the Immersed Boundary method. A typical boundary particle j on the surface of the rigid body exerts a repulsive force $m_a m_j \mathbf{r}_{aj} f(|\mathbf{r}_{aj}|)$ on fluid particle a along the line joining their centres, where $\mathbf{r}_{aj} = \mathbf{r}_a - \mathbf{r}_j$. Correspondingly, fluid particle a exerts an equal but opposite force $m_j m_a \mathbf{r}_{ja} f(|\mathbf{r}_{aj}|)$. The form of the function $f(|\mathbf{r}_{aj}|)$ is chosen so that it mimics a delta function and provides a force on the fluid particle which is normal to the surface of the body to a very close approximation [6]. 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.

In these equations m_b is the mass of particle b, P_b is the pressure, and ρ_b is the density at the position $\mathbf{r}_b$ of the particle. The fluid is weakly compressible, with an equation of state

Numerical Simulation of a Helicopter Ditching with Emergency Flotation Devices.

Bruce K. Cartwright and Allen Chhor
Pacific ESI
Sydney, Australia
brucec@esi.com.au, allenc@esi.com.au

Paul H.L. Groenenboom
ESI Group
Delft, The Netherlands
pgr@esi-group.com

***Abstract*— Smoothed Particle Hydrodynamics (SPH) is used to model the ditching of a helicopter. Helicopters have a high centre of gravity, are not integrally watertight so tend to roll over and sink within seconds of landing on the water surface. Modern helicopters employ Emergency Flotation Systems (EFS) that are essentially large air-bags to provide increased buoyancy and stability to the helicopter, thereby enhancing occupant survival rates. A hybridised finite element code that includes SPH elements, rigid finite elements, flexible finite elements, sliding contact interface algorithms and air-bag simulation tools from the automotive industry is used to demonstrate the behaviour of the complex fluid-structure interaction scenario of a helicopter landing on water with and without inflating air-bags.**

I. Introduction

When a helicopter ditches into the sea, the high centre of gravity, narrow base and lack of watertight integrity cause it to overturn and sink rapidly. An Emergency Flotation System (EFS) such as inflatable pontoons that can be deployed prior to, or upon, contact with the water, can keep the helicopter afloat, and prevent it from overturning. This greatly improves the survival chance of the occupants.

The behaviour of an airframe with flexible pontoons in impacting and floating on water is a complex phenomenon. The explicit finite element code PAM-CRASH with its SPH solver has already been shown [1,2] to successfully simulate various scenarios involving fluid-structure interaction (FSI). The dynamics of a helicopter ditching with a rigid EFS has previously been demonstrated [3]. The development in this paper exploits the (automotive) occupant safety airbag model within the aforementioned code to approximate the inflation of large flexible pontoons, facilitating the simulation of ditching of helicopters fitted with an EFS.

II. Numerical Approach

A. *Overview*

The PAM-CRASH general purpose FE code used is built around an explicit solver optimized for the analysis of highly non-linear structural dynamical behavior. It contains element formulations for thin shells, solid elements, membranes and beams with material models for plasticity and failure for metals, plastics, rubbers, foams and composites. The code contains robust contact algorithms, including sliding interface algorithms, to model the dynamic contact between various parts within a model. The solver allows both finite elements and smoothed particles to be used simultaneously within the same model, with optimal performance achieved by using either the Distributed Memory Parallel (DMP) or the Shared Memory Parallel (SMP) version.

Interaction between particles representing a fluid and moving or deformable structures may be modelled by one of the contact algorithms. Such algorithms, while allowing sliding at the interface, prevent penetration between selected structures. Their implementations are based on the well known penalty formulation, where geometrical interpenetrations between so-called slave nodes and adjacent master faces are penalized by counteracting forces that are essentially proportional to the penetration depth. The contact algorithm will automatically detect when a particle (slave) penetrates any segments (master) of the outer surface of the finite element model of the structure. The contact thickness indicates the distance away from a contact face where physical contact is established. For SPH particles as slaves, the contact thickness should be representative of the particle spacing. This type of contact has been validated by the vertical motion of floating bodies. It has been found that the correct position is reached when the thickness defined for the contact is in the vicinity of half the particle spacing and the artificial viscosity coefficients are significantly smaller than the values normally applied for shocks. In that case the upward force is also correct. The use of coupled SPH-FE has been used for many applications such as sloshing [4], the opening of a heart valve [5], the impact of birds onto aeronautical structures [6], and lifeboat drop [7] onto water.

In addition, it is possible to define a tied contact in which virtual spring elements are automatically defined between particles that are sufficiently close to segments in the initial configuration. Such a contact acts as a rigid connection between the two parts.

B. *Hydrostatic Pressure Initialisation*

For many hydrodynamic simulations involving a free surface, a significant amount of computer time may be expended to reach hydrostatic equilibrium under gravity in the water before the event of interest can start. An option has been implemented that allows initializing the hydrostatic pressure at the start of the simulation. This may be done for an assembly

Hybrid CPU-GPU acceleration of the 3-D parallel code SPH-Flow

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

R. Dolbeau, P.-L. Cabelguen, L. Bertaux
CAPS Entreprise
Rennes, France

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

***Abstract*— SPH-flow software developed jointly by the Fluid Mechanics Laboratory of Ecole Centrale Nantes and HydrOcean is now capable of modelling complex free surface flows with complex 3-D geometries, fluid-structure interactions, and multi-fluids [6] [7]. Today the developments of this code are continuing inside a consortium joining several academic and industrial partners. This 3-D parallel software based on MPI is clearly oriented towards massively parallel computing on machines fitted with several hundreds of CPU nodes in distributed memory architectures. Thanks to its current implementation, this code is able to process industrial applications involving millions of particles [4][5][8]. The stated aim is to push further the current limits of SPH calculations, bypassing the inherent limits of this method (small time steps, time consuming particle-to-particle interaction loops, etc). As such, the code must evolve to benefit from new hardware architectures available on the market. Today, the use of GPU devices to speed up scientific calculations is widely documented, and appears to be a real evolution in computational technologies. Different test studies related to the GPU translation of SPH codes have been conducted recently, mainly within the SPHERIC community, showing very promising results [2][3]. Generally speaking, other specialties for which scattered data need to be computed see a great interest in GPU (see for example [1] in molecular dynamics). However, GPU computing requires a specialized adaptation of the code in regards to language (CUDA or other languages) and the algorithms implemented (memory access and contiguous data in the memory bank).**

This paper presents a first attempt of CPU-GPU hybridization of SPH-Flow. The original MPI-based distributed memory parallelization is preserved, allowing massive calculations on several CPU and GPU devices.

I. INTRODUCTION

The work presented in this paper is a joint effort between Hydrocean and CAPS Entreprise. The latter company is developing the "Hybrid Multicore Parallel Programming" (HMPP) technology. This programming environment is a directive-based compiler dedicated to build parallel GPU accelerated applications. It targets NVIDIA as well as AMD/ATI GPUs. As part of the promotion and development of High-Performance Computing in France and Europe, the French national high-performance computing agency GENCI (Grand Equipement National de Calcul Intensif) joined CAPS Entreprise to carry out a call for proposals for the scientific community for porting complex applications on hybrid graphic-based accelerator systems. The main requirement on admission to the project resided in the prior existence of a parallelization dedicated to distributed memory architectures (via MPI for instance). SPH-Flow has been retained on this project call, and a first attempt of CPU-GPU hybridization has been achieved. For this first attempt of CPU-GPU hybridization, it was thus decided to retain the current parallel algorithm of SPH-Flow (based on Fortran90 + MPI) and to adapt the main time consuming procedures to GPU computing.

In this paper, the various questions regarding CPU-GPU hybridization of a code initially programmed using MPI libraries is presented and discussed. Particular attention is given to the necessary changes to be performed on the SPH algorithm originally dedicated to parallel CPU for making it fit to CPU-GPU hardware. The main adaptation techniques used are based on particle-to-particle interaction loop adaptations, and on Hilbert space filling curves dedicated to sorting particle data. The chosen strategy for the CPU-GPU translation of a code originally designed for parallel CPU architectures is then presented. Finally, the speedup obtained is presented and discussed on up to 32 CPU-GPU hybrid compute nodes.

Development of a Dual CPU-GPU SPH model

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

***Abstract*— A Dual CPU-GPU code is introduced in this manuscript. The code has been developed starting from the SPHysics model taking advantage of the potential and high performance of Graphics Processing Units (GPUs). On one hand, the code is accelerated using a powerful parallel programming model and, on the other hand, new realistic applications can be carried out using a huge number of particles due to the memory space provided by the GPU cards. All the features and possibilities of the code are described in detail.**

I. INTRODUCTION

SPH model has proven to be a suitable tool to reproduce a wide variety of astrophysical applications and hydrodynamic problems. The SPH method is capable of dealing with problems with free surface, deformable boundary, moving interface, wave propagation and solid simulation. However any of those applications requires an excessive computational time if a fully three dimensional simulation is carried out. In order to analyze some seconds of physical time, an implementation on a single CPU not only takes a large computational time but also provides results without enough physical information if millions of particles are not being simulated. Furthermore, a short period of physical time takes a large computational time of simulation when the code runs on a single CPU and the number of particles to be considered is limited if parallel computing is not implemented. Graphics Processing Units (GPUs) are designed to treat a high data flow and to render pixels (at a few tens of frames per second) and from a computational point of view they are highly efficient. Due to the unstoppable development of the market of video games and multimedia, their computing power has increased much faster than the one of CPUs. Thus, GPUs appear as an accessible alternative to accelerate SPH models using a powerful parallel programming model where the graphics cards are used as the execution devices.

A Dual CPU-GPU SPH code has been developed to deal with real-life engineering problems using SPH models. Two main advantages are obtained from using GPU computing on SPH; the achieved speedups and the possibility to deal with simulations using a huge number of particles. Thus, the dual CPU-GPU solver allows the detailed study of some problems that require massive computation.

SPHysics is a SPH numerical model developed to study free-surface flows. It is the product of a collaborative effort amongst researchers at the Johns Hopkins University (US), the University of Vigo (Spain) and the University of Manchester (UK). The code, written in FORTRAN90, is available to free download and for public use at http://www.sphysics.org. A complete description of the software is found in the SPHysics user's guide [1]. Although the method allows a fine description of the flow in the shore areas, its main drawback is its high computational cost, in such a way that the model cannot be efficiently applied over large domains. That is the reason why GPU computing is demanded, as a result the CPU-GPU code name *DualSPHysics* has been developed starting from the SPH features implemented in the FORTRAN SPHysics code. More information about the *DualSPHysics* project can be found at http://ephyslab.uvigo.es/index.php/eng/dual_sphysics/ and different applications and animations can be visualized at http://vimeo.com/user3287916/videos.

The work presented in [2] introduced the framework to implement SPH codes using the best approaches and performance optimization techniques. That manuscript was focused on identifying the best approaches for efficient parallelization of work since a properly and full use of all the capabilities of the GPU architecture is not straightforward. As a first step the implementation was focused on solving the particle interactions on GPU computing since SPH models spend most of their computational time on that. Next moves were the implementation of the neighbor list and the time integration in order to develop a totally GPU-SPH model. The manuscript [3] analyzed the best approaches for neighbor lists. Different procedures were analyzed as gridding algorithms, reordering particles techniques and two approaches to create the neighbor list were compared. Best results were considered to implement the *DualSPHysics* code and optimizations were applied to CPU and GPU versions obtaining a robust and efficient CPU-GPU solver.

INDEX