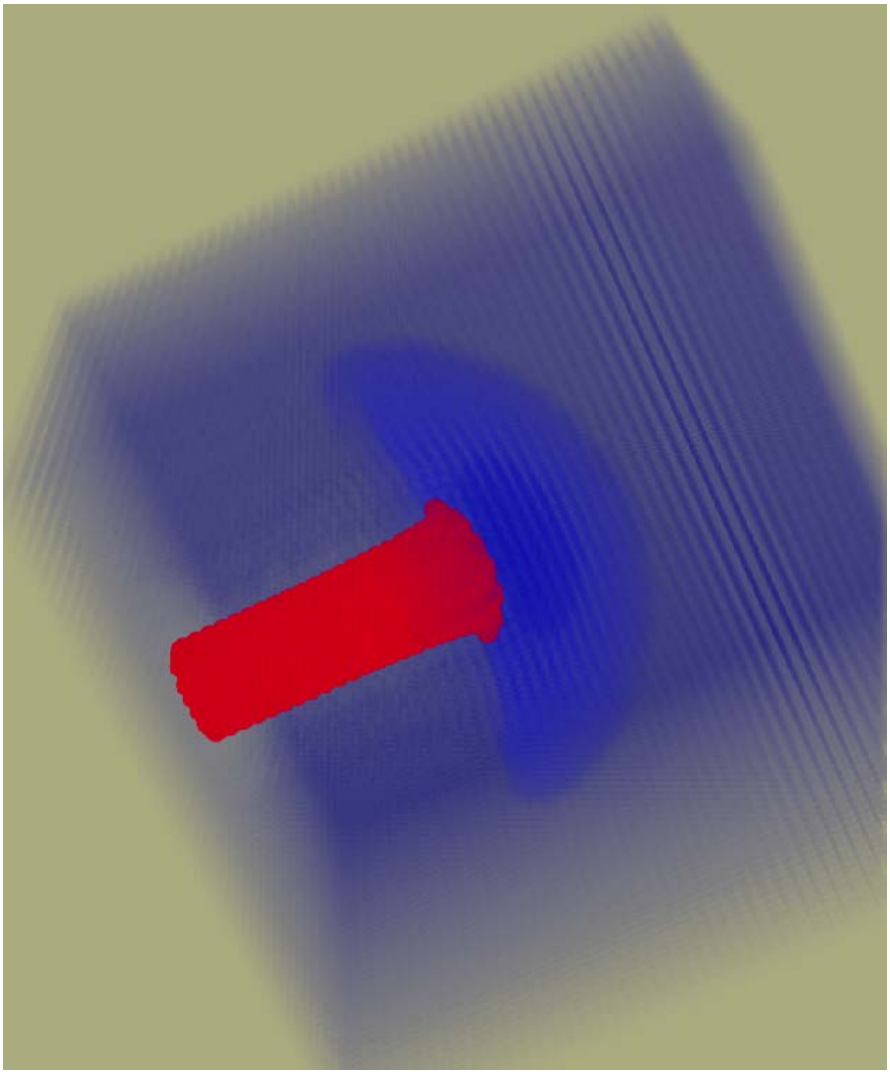
ERCOFTAC SIG SPHERIC

III[rd] International Workshop

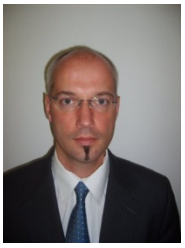
Ecole polytechnique fédérale de Lausanne

2008, June, 4[th] - 6[th]



Editor: Dr. Pierre Maruzewski

Foreword



"This volume presents an update on the state of research and development in Smoothed Particle Hydrodynamics as it includes all the papers presented at the Third Workshop on SPH held at EPFL, the Swiss Federal Institute of Technology in Lausanne, June 4-6, 2008. The present volume is the printed version of the 51 papers made available through a USB key issued at the Workshop to the participants to ensure timely dissemination of the latest results provided by the authors. The volume is thus dedicated towards specialists and professionals in the area of SPH applications, who could not attend the Workshop as well as anybody who would like assess the most active current research areas in the field of SPH methodology.

The volume is structured according to the Workshop topics which includes the following sections:

- Advances in SPH model;
- Free surface flows;
- Wave impact;
- Incompressible method;
- Turbulence;
- High performance computing;
- Non Newtonian fluids;
- Fluid-Structure;
- Multiphase flows;
- Astrophysics.

After the above few comments, which certainly miss the expected vivid and stimulating presentation and discussion held during the Workshop I would like to thank all the contributors for keeping to the tight schedule for the submission of papers and Prof. François Avellan, Director of EPFL Laboratory for Hydraulic Machines for agreeing to organize the Third SPH Workshop. I would like to acknowledge the tremendous work of Mrs. Valérie Jacquot-Descombes in touch with all EPFL services. I further thank as well as the local team, Dr. Cécile Munch-Alligné, Dr. Mohamed Fahrat, Olivier Braun, Alireza Zobeiri and Vlad Hasmatuchi of the EPFL Laboratory for Hydraulic Machines for helping in the edition of this volume. Special thank goes to Dr. Philippe Cerrutti who made possible SPHERIC IIIrd Web Site and to Mrs. Isabelle Stoudmann for her administrative support.

This Workshop will not be possible without the commitment of Dr. Etienne Parkinson and Dr. Jean-Christophe Marongiu, Dr. Jean Favre and John Biddiscombe, Co-Chairmen of the Workshop.

We are very grateful to our sponsors, who financially support the edition of this volume, EPFL, CSCS, VATECH Hydro Andritz and ERCOFTAC, Further thanks to Lausanne Tourisme for their help in organizing the tourism guide.

Lausanne, June 4, 2008"

Dr. Eng. Pierre Maruzewski
Chairman of SPHERIC Workshop IIIrd and editor
EPFL – STI - LMH
Switzerland

Preface

"Smoothed Particle Hydrodynamics (SPH) was developed to study non-axisymmetric phenomena in astrophysics. However, developments based on this approach have been launched for a far wider range of applications by various research teams in association with industry. Since the meshfree technique facilitates the simulation of highly distorted fluids/bodies, it offers great potential in fields such as free-surface flows, elasticity and fracture, where Eulerian methods can be difficult to apply. Furthermore, with the ever increasing size and cost reduction of computer clusters, parallel implementations allow large-scale simulations that were previously limited to mainframes.

Following the impulse generated by a collection of local initiatives, France, United Kingdom, Italy.., a need has been identified to foster and cluster efforts and developments. The goal of SPHERIC is indeed to foster the spread of this simulation method within Europe and abroad. It forms a framework for closer co-operation between research groups working on the subject, and serves as a platform for information exchange from science to industry. One of the most important goals is the assessment and development of this method for all its possible applications. With an emphasis on validation, it will then be possible to consider transfer of information, and thereby technology, to companies not currently engaged in the technology. Today, June 2008, 30 months after the birth of SPHERIC, the number of member organizations has reached 57, including 8 industrial partners.

These goals can best be achieved through annual workshops. These organized to serve the following goals:

To develop the basic scientific concepts, including parallelism and post-processing methods.

To communicate experience in the application of the technology,

To foster communication between industry and academia,

To discuss currently available as well as new concepts,

To give an overview of existing software and methods,

To define and run benchmark test cases.

After the first SPHERIC workshops, Rome 2006, Madrid 2007, we are now meeting in Lausanne for a promising third session. We expect that the number of attendees will exceed 100, and about 40 oral presentations will be held. From these proceedings it will be clear that SPH is becoming very popular for fluid-structure interactions, a field where this method shows all its qualities. It is also noteworthy that a training day is organized around SPH modeling and post-treatment before the workshop. We expect that these four days of presentations and discussions will benefit your current and future work."

Damien Violeau
Chairman of ERCOFTAC SIG SPHERIC
EDF - LNHE
France

Dr. Eng. Pierre Maruzewski
Chairman of SPHERIC Workshop IIIrd
EPFL – STI - LMH
Switzerland

Dr. Nathan Quinlan
Secretary of SPHERIC
National University of Ireland
Ireland

Dr. Ben Rogers
Webmaster of SPHERIC
University of Manchester
United Kingdom

Committee

I. ORGANIZING COMMITTEE

- Dr. Pierre Maruzewski, EPFL - LMH, Lausanne, chairman of the 3rd SPHERIC
- Dr. Jean Favre, CSCS, Manno
- Dr. John Biddiscombe, CSCS, Manno
- Dr. Etienne Parkinson, Andritz, VA TECH Hydro, Vevey
- Dr. Jean-Christophe Marongiu, Andritz, VA TECH Hydro, Vevey

II. SPHERIC STEERING COMMITTEE

- Damien Violeau, Electricité de France, LNHE, Chatou, France
- Dr. Nathan Quinlan, National University of Ireland, Galway, secretary
- Dr. Ben Rogers, University of Manchester, UK, webmaster
- Prof. Dominique Laurence, EDF, University of Manchester
- Prof. Moncho Gomez-Gesteira, University of Vigo, Spain
- Dr. Andrea Panizzo, University of Rome, La Sapienza, Italy
- Dr. Etienne Parkinson, Andritz VA TECH Hydro, Switzerland
- Dr. Paul Groenenboom, ESI BV, Netherlands
- Dr. David Graham, University of Plymouth, UK
- Dr. David Le Touzé, Ecole Centrale de Nantes, France
- Dr. Jean Favre, CSCS, Switzerland
- Dr. Peter Berczik, Ruprecht-Karls-Universität Heidelberg, Germany

III. LOCAL ORGANIZING COMMITTEE

- Prof. François Avellan, EPFL - LMH
- Dr.Pierre Maruzewski, EPFL - LMH
- Dr. Mohamed Fahrat, EPFL - LMH
- Dr. Philippe Cerrutti, EPFL - LMH
- Dr. Cécile Münch, EPFL - LMH
- Ing. Olivier Braun, EPFL - LMH
- Ing. Alireza Zobeiri, EPFL - LMH
- Isabelle Stoudmann, EPFL - LMH
- Valérie Jacot-Descombes, EPFL - DII-I

IV. INTERNATIONAL SESSION CHAIRMEN

- Prof. Joe Monaghan, Monash University, Australia
- Dr. Antonio Souto-Iglesias, Universidad Politecnica de Madrid, Spain
- Dr. Ben Rogers, University of Manchester, United Kingdom
- Damien Violeau, EdF –LNHE, France
- Dr. David Le Touzé, Ecole Centrale de Nantes, France
- Prof. Javier Bonet, Swansea University, United Kingdom
- Dr. Etienne Parkinson, VATECH Hydro Andritz, Switzerland
- Dr. Nathan Quinlan, National University of Ireland, Ireland
- Prof. Francis Leboeuf, Ecole Centrale de Lyon, France
- Dr. Andrea Colagrossi, INSEAN, Italy
- Prof. Jean-Paul Vila, INSA de Toulouse, France
- Dr. Peter Berczik, Astronomisches Rechen-Institut, Germany
- Dr. Pierre Maruzewski, EPFL, Switzerland

Contents

IV. POSTER SESSION

Smoothed Particle Hydrodynamics Stochastic Model for Flow and Transport in Porous Media

Alexandre M. Tartakovsky
Paci c Northwest National Laboratory
Richland, USA
alexandre.tartakovsky@pnl.gov

Daniel M. Tartakovsky
University of California San Diego
La Jolla, USA
dmt@usd.edu

Paul Meakin
Idaho National Laboratory
Idaho Falls, USA
paul.meakin@inl.gov

***Abstract*— A meso-scale stochastic Lagrangian particle model was developed and used to simulate conservative and reactive transport in porous media. In the stochastic model, the fluid flow in a porous continuum is governed by a combination of a Langevin equation and continuity equation. Pore-scale velocity fluctuations, the source of hydrodynamic dispersion, are represented by the white noise. A smoothed particle hydrodynamics method was used to solve the governing equations. Changes in the properties of the fluid particles (e.g., the solute concentration) are governed by the advection-diffusion equation. The separate treatment of advective and diffusive mixing in the stochastic transport model is more realistic than the classical advection-dispersion theory, which uses a single effective diffusion coefficient (the dispersion coefficient) to describe both types of mixing leading to over-prediction of mixing induced effective reaction rates. The stochastic model predicts much lower reaction product concentrations in mixing induced reactions. In addition, the dispersion theory predicts more stable fronts (with a higher effective fractal dimension) than the stochastic model during the growth of Rayleigh-Taylor instabilities.**

I. Introduction

Flow and transport in porous media can be alternatively described on two fundamental scales. On the pore (microscopic) scale, these phenomena are governed respectively by the Navier-Stokes and advection-diffusion equations. On the continuum scale (with a characteristic length much larger than the size of a single pore) the o w and transport are usually described by macroscopic transport equations [1], [2] that represent the volumetric or statistical averages of the Navier-Stokes and advection-diffusion equations. These equations consist of the continuity equation,

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

the momentum conservation equation,

$$\frac{d\mathbf{u}}{dt} = -\frac{\nabla p}{\rho} + \mathbf{g} - \gamma \mathbf{u}, \quad \frac{d\mathbf{x}}{dt} = \mathbf{u}, \tag{2}$$

and the advection-dispersion equation,

$$\frac{dC}{dt} = \frac{1}{\rho} \nabla \cdot (\rho D \nabla C), \tag{3}$$

where $\mathbf{x}$ is the position of a uid volume element moving through a continuous porous medium with an average pore velocity of $\mathbf{u}(\mathbf{x})$, and $\rho(\mathbf{x})$ and $\mu(\mathbf{x})$ are the density and viscosity of the uid at position $\mathbf{x}$. The friction coef cient $\gamma = \phi\mu/(\rho k)$, porosity ϕ, permeability k and hydraulic conductivity $\phi g/\gamma$ are some of the macroscopic parameters characterizing the continuum properties of porous media. In Eq. (3), C is the solute concentration (mass fraction),

$$D = (D_m/\tau + \alpha \, |\mathbf{u}|) \tag{4}$$

is the dispersion coef cient, treated for simplicity as a scalar; D_m is the molecular diffusion coef cient, and τ and α are the tortuosity and the dispersivity of the porous medium.

In a homogeneous porous medium, dispersive mixing is the result of a combination of molecular diffusion (diffusive mixing) and spreading due to variations in the uid velocity (advective mixing). In the Darcy(continuum)-scale o w equation (2) these velocity variations are disregarded and the Darcy-scale dispersion theory, Eq. (3), treats dispersion in a homogenous porous media as a Fickian diffusion process with a macro-scale effective diffusion coef cient, D, which combines the effects of advective mixing and diffusive mixing [1]. The form of the dispersion coef cient can be derived by treating the Darcy velocity eld as a stochastic process with a prescribed velocity covariance function using ensemble averaging [3] and renormalization-group analysis [4] or by using the method of volume averaging [5], which considers pore-scale velocity uctuations around the average (Darcy) velocity. However, there is ample evidence that dispersive mixing is very different from purely diffusive mixing (e.g. the fractal dimensions of the fronts (isoconcentration contours) resulting from diffusion and dispersion are different [6]), and the advection-dispersion equations also signi cantly overpredict the extent of reaction in mixing induced chemical transformations [7]–[9]).

We propose a new stochastic Lagrangian particle model for o w and transport in porous media that allows advective mixing and the diffusive mixing to be treated separately. We assume that uid o w in homogeneous porous media is governed by a stochastic Langevin o w equation that is obtained by adding white noise uctuations, ξ, to the momentum conservation equation (2). As in [3] and [4]), the noise represents random interactions between the uid and the disordered porous medium that forces uid o w paths to deviate from the smooth o w paths predicted by the Darcy scale continuum o w equations (Eqs. 1,2). The molecular diffusion of solutes carried by the uid is governed

Oblique impact of a jet on a plane surface solved by SPH: suggestions to improve the results of the pressure profiles

Diego Molteni
Dipartimento di Fisica e Tecnologie Relative,
Universita di Palermo, Viale delle Scienze, 90128,
Palermo, Italy
molteni@unipa.it

Andrea Colagrossi
INSEAN,
Italian Ship Model Basin,
Rome, Italy
a.colagrossi@insean.it

***Abstract*— Smoothed Particle Hydrodynamics method is well suited for simulations of un-compressible fluid systems with free surfaces or quickly varying boundary conditions A very convenient approach in the SPH framework is the one assuming a weak compressibility of the fluid: in this case the numerical solution procedure is simple and explicit. We study the case of a jet of water on an inclined rigid wall. This problem has an analytical solutions and therefore it is possible to make an accurate, comparison of the SPH results with the analytical ones. In the standard SPH implementations, it is well known that the pressure values exhibit a consistent noisy behavior, since small density variations are amplified by the large sound speed, therefore producing random pressure spikes. We propose a simple therapy that produces more regular and accurate pressure profiles. Essentially we add to the continuity equation a diffusive term, with a coefficient proportional to the sound speed, the kernel of the particles and to the particles size. This new term is analogous to the artificial viscosity that appears into the momentum equation. It is purely numerical and in the continuum limits it goes to zero and the general conservation properties of the algorithm are not changed. We find that the best results are obtained with $\xi\, h\, c_s\, (W_{ik}/W_{ii})^2$ where ξ is an a-dimensional coefficient of the order ≈ 0.1, *h* is the particle smoothing length, c_s is the sound speed and W_{ik} is the kernel at the *i* position evaluated at the *k* position. We have verified the improvements is excellent in many cases. Here These comparisons testify the quality of the improvements. The computational cost of the proposed therapy is quite small since the diffusion the can be calculated at the same time level of the other spatial derivatives.**

I. Introduction

The Smoothed Particle Hydrodynamics (SPH) method is very appropriate for simulating complex fluid dynamics and is frequently used when the problem shows strong free surface dynamics. Indeed in this case a mesh free method has to be preferred. However in many problems the attention is focused on the fluid kinematics instead of the prediction of the pressure field. In general the flow speeds look quite good, but checking the distribution of pressure, the situation is different: large random pressure oscillations are present due to numerical high frequencies acoustic signal. In the simulations of violent liquid-solid impacts the pressure value is very important and its evaluation turns out to be critical for practical applications. Such kind of impact problems are strongly time dependent and the analytical pressure solution, to compare with computer pressure values, are complex and limited to very simple geometries (see e.g. [2]). The result is that this problem is not deeply investigated in the SPH literature. In [1] the authors suggested a filtering of the density with a MLS integral interpolation as a way to reduce the numerical noise in the pressure evaluation. That procedure gives good results, but for long time simulation it does not properly conserves the total volume of the particle' system since the hydrostatic component has been improperly filtered [8] Following the idea of the smoothing process, it is rather natural to think to a diffusion process. Furthermore it is well known that, in the framework of SPH, the diffusion algorithm is conservative and therefore it preserves the mass conservation. Obviously the diffusion coefficient has to vanish as the numerical accuracy increases in order to avoid unphysical effects and to recover the consistency of the discrete equations. We analyzed few simple problems and found clear evidence of the discrepancy between analytical and computer pressure values and clear evidence of the improvements. Particular attention has also be paid on the time integration scheme adopted which play an important role both in pressure evaluation as well as in the computation costs.

II. Governing Equations

A. The continuum equations

In free-surface SPH, the fluid is generally considered unviscous and the flow is free to have rotational motion; the problem is thus governed by the Euler equation in the domain Ω which in the presence of a generic external force field $\vec{f}$ reads

$$\frac{D\vec{u}}{Dt} = -\frac{1}{\rho}\nabla p + \vec{f}$$

for the momentum equation, where $\frac{D}{Dt}$is the comoving derivative, $\vec{u}$ is the fluid speed, *p* the pressure;

$$\frac{D\rho}{Dt} = -\rho\, div\,(\vec{u})$$

A Hybrid Boussinesq-SPH Model for Coastal Wave Propagation

A. J. C. Crespo
Grupo de Física de la Atmósfera y del Océano
Universidade de Vigo
Ourense, Spain
alexbexe@uvigo.es

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

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

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

Abstract— **One of the important advances to be made for SPH is the development of a hybrid model that couples the computationally intensive, but highly resolved, SPH code to a more efficient, but less descriptive, model in the offshore. For example Boussinesq models are efficient at propagating waves ashore; however, special techniques are required to track multiply connected free surfaces that occur during the wave breaking process. By coupling the two models, an optimized model for coastal wave propagation can be developed. Here, a model that couples FUNWAVE (a fully nonlinear Boussinesq model for surface waves) to SPHysics is presented.**

I. INTRODUCTION

The periodic passage of storm near coastal areas gives rise to dangerous waves on the shore line during winter time. It is difficult, computationally and physically, to analyze these events by means of a single model due to the presence of multiple scales both in time and in space. The present study considers the use of different models to generate and propagate the wave field from the open sea to the coastal region. In particular, the interaction between high waves and coastal structures is considered.

A variety of models using different numerical and physical approaches has been developed to handle wave propagation and different types of wave transformations as refraction, diffraction, breaking, run-up and overtopping. Recently, research is focused on coupling models with different numerical and mathematical approaches ([1] and [2]). These models have several advantages and limitations, but the primary goal of such an approach is to combine the advantages of the individual models in a single model, thus increasing the accuracy, efficiency and regime of validity.

A hybrid method is developed starting from two existing wave propagation models. The model couples the finite difference Boussinesq model, FUNWAVE ([3]) to SPHysics ([4]) model.

FUNWAVE has been proven to be able to simulate surface waves from deep water with accuracy and with satisfactory results both in the intermediate water depths and in near shore areas.

SPHysics (http://wiki.manchester.ac.uk/sphysics.) has been used for wave impact studies on offshore structures ([5], [6] and [7]) showing good agreement between numerical and experimental results. Unfortunately, its computational cost does not permit extending the model far from the near shore area.

This hybrid approach employs FUNWAVE for wave propagation in the offshore region and uses SPHysics to handle wave breaking, run-up and overtopping in the domain close to coastal structures.

A first approach and an example of implementing a one way coupling can be seen in [8].

II. BOUSSINESQ MODELS

Boussinesq-type equations provide a general basis for studying wave propagation in two horizontal dimensions. At their core, the equations are the shallow water equations for nondispersive linear water propagation. This basic foundation is extended by the addition of terms which include the lowest order effects of nonlinearity and frequency dispersion. This formulation provides a sound and

Simulation of Interfacial and Free-Surface Flows using a new SPH formulation

A. Colagrossi, M. Antuono
INSEAN
The Italian Ship Model Basin
Via di Vallerano 139, Roma, Italy
a.colagrossi@insean.it

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

D. Molteni
Dipartimento di Fisica e Tecnologie Relative
Università di Palermo
Viale delle Scienze 18, Palermo, Italy

***Abstract*—In this work a new SPH model for simulating interface flows is presented. This new model is an extension of the formulation discussed in [4], and shows strong similarities with one proposed by Hu & Adams [6] to study multiphase flow. The main difference between these two models is that the present formulation allows for simulating multiphase flows together with the presence of a free surface.**

The new formulation is validated on test cases for which reference solutions are available in literature. A Rayleigh-Taylor instability is first studied. Then, the rise of an air bubble in a water column is investigated. Finally, the model capabilities are illustrated on the case of a drop of a heavy fluid entering a tank filled with water.

I. Introduction

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

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

In the present work a new SPH model for simulating interface flows is presented. This new model is an extension of the formulation discussed in Colagrossi & Landrini [4] and it is based on the variational approach introduced by Bonet & Lok [2]. The SPH formulation presented here shows strong similarities with one proposed by Hu & Adams [6] to study multiphase flows. The main difference between these two models is that the present formulation allows for simulating multiphase flows together with the presence of a free surface (meaning here an interface without accounting for the (air) phase above it).

Further, in the present formulation a specific attention is paid to enhance the accuracy of the scheme, especially through the use of a Shepard kernel. This leads in particular to the derivation of an original variant of renormalization of the gradient of this kernel, which differs from the one usually associated in literature to this Shepard kernel.

The formulation is validated on test cases for which reference solutions are available in literature. After classical tests as the one of a droplet oscillating without gravity, more complex validation cases are simulated such as Rayleigh-Taylor instabilities, or an air bubble rising by gravity in a water column at rest. The last problem studied in the present work is the case of a droplet of a heavy fluid entering a tank filled with water. The latter involves two different kinds of liquids and the free-surface dynamics, namely the droplet impact generates free-surface gravity waves which radiate far away.

II. Physical model

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

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

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

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

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

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

Swimming With and Without Skin

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

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

***Abstract*— In this paper we describe our SPH simulations of swimming linked bodies in two dimensions. The bodies are ellipses that pivot around links, which lie between the ellipses. In our simulations the ellipses are defined by boundary force particles around their perimeter. In the absence of a skin the fluid can pass between the linked bodies. If they have a skin the fluid is prevented from flowing between the bodies and the swimming is more like that of real fish or eels. We model the skin by SPH skin particles, which are coupled by elastic forces. The skin particles connect the bodies and join seamlessly to the boundary force particles that define the shape of the rigid bodies. The convergence of the simulation is discussed, as well as the scaling relations satisfied by the dynamics. The results may be generalized to an arbitrary number of linked bodies each of which may have a different geometry.**

I. Introduction

This paper is concerned with the investigation of swimming in two dimensions. In this particular case, the swimming object is composed of linked, rigid, elliptical bodies. Motion through a fluid is due only to changes in angles between adjacent bodies. This work follows on directly from [1] in which linked bodies without skin were considered.

The previous work was most similar to simulations by Kanso et al. [2] and Eldredge [3]. Both studied motion of three linked ellipses in a two dimensional fluid of infinite domain. Kanso et al. considered an inviscid fluid, whereas the vortex particle method used by Eldredge included viscosity.

The aim of this work is to investigate the changes to the motion when a skin is included. In the absence of a skin, the fluid could flow through the gaps between connected ellipses. If they have a skin, the fluid is prevented from flowing through the gaps. Hence, the linked bodies with skin are more like real fish or eels. It is shown that the inclusion of skin makes the swimming more efficient. For a prescribed gait, the bodies move further in a given period of time with skin than they do without. The fluid in our simulations has a periodic boundary, and the Reynolds number is $\Re = 200$.

The fluid is modeled using SPH fluid particles. The boundaries of the rigid bodies are defined by boundary force particles. The present work introduces SPH skin particles, which define a layer of elastic skin. Like body particles, the skin particles interact with the fluid particles via boundary forces. The difference, however, is that they interact with their neighbours through a spring force. The skin behaves like an elastic cover between the bodies, and seamlessly connects to the bodies at specified points.

II. Methodology

In the present work, we consider motion of three elliptical, linked bodies in a viscous fluid in two dimensions. A typical configuration of the rigid bodies in shown in Fig. 1.

The motion of the linked bodies is controlled through the angles φ_1 and φ_2 which are specified functions. In this case the angles are functions of t, although in general they may also be functions of spatial coordinates.

The main aim in this work is to compare the motion of the linked bodies with and without skin. Where the bodies do not have skin, fluid can pass through the gaps between the bodies. When the skin is attached, the gaps between the bodies are covered, preventing fluid from passing through. We expect that the bodies with skin swim a further distance in a given period of time.

A. Equations of motion

The equations of motion for the fluid are the Navier Stokes equations. In cartesian tensor from these equations are

$$\frac{d\mathbf{v}}{dt} = \frac{1}{\rho}\left(\nabla\cdot\mathbb{T} - \nabla P + \mathcal{F}\right), \tag{1}$$

where $\mathbb{T}$ is the stress tensor and P is the pressure. The formulation of the boundary force $\mathcal{F}$ is related to that of Peskin [4].

The equation of motion for a rigid body k with mass M_k is

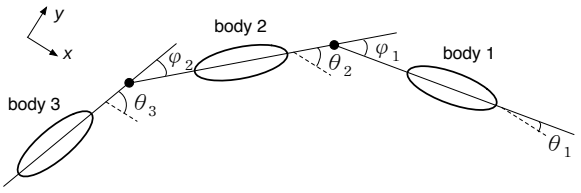


Fig. 1. Configuration of the linked bodies. The link positions are denoted by the filled circles. The prescribed angles φ_1 and φ_2 give the orientation of one body relative to the next. The angles are measured between the major axes of the ellipses, which run through the centre of mass of each body. The angles θ are defined relative to a fixed direction, that is, the x axis of the cartesian coordinate system.

A New 3D Parallel SPH Scheme for Free-Surface Flow

A. Ferrari
Department of Civil and Environmental Engineering
University of Trento
I-38050 Trento, Italy
ferraria@ing.unitn.it

M. Dumbser
Department of Civil and Environmental Engineering
University of Trento
I-38050 Trento, Italy
dumbserm@ing.unitn.it

E.F. Toro
Department of Civil and Environmental Engineering
University of Trento
I-38050 Trento, Italy
toroe@ing.unitn.it

A. Armanini
Department of Civil and Environmental Engineering
University of Trento
I-38050 Trento, Italy
armaninia@ing.unitn.it

Abstract— **In alternative to the classical approaches [1], [2], a new robust and accurate SPH scheme is proposed, able to track correctly complex three-dimensional free-surface flows and, even more important, also able to compute an accurate and little oscillatory pressure field. A new treatment of the solid boundary conditions is also proposed. To asses the accuracy of the new SPH scheme, a three-dimensional mesh-convergence study has been performed for the strongly deforming free-surface in a 3D dam-break and impact-wave test problem.**

The parallelization of our new 3D SPH scheme has been carried out using the MPI paradigm together with a dynamic load balancing strategy to improve considerably the computational efficiency of the code. Thus, simulations involving millions of particles can be run on modern massively parallel supercomputers, obtaining a very good performance, as confirmed by a speed-up analysis. The applications in 3D consist of environmental problems, such as dam-break flows and impact flows against a wall. The numerical solutions obtained with our new 3D SPH code have been compared with either experimental results or with other numerical reference solutions, obtaining in all cases a very satisfactory agreement.

I. Introduction

The SPH scheme is a technique for approximating the continuum equations [3] based on particle-particle interactions and can be interpreted as an explicit central finite difference (FD) scheme with moving points. It has been demonstrated in the literature that the scheme is not stable using explicit Euler time stepping [4]. In the literature, two different SPH versions have been used to stabilize the scheme.

The oldest and more common formulation was proposed by Gingold and Monaghan [1], where an artificial viscosity term is added in the evolution equation of the velocity. It produces a repulsive force between the particles when two points approach each other. This formulation is very accurate to compute the free-surface profile of an incompressible fluid, but the numerical pressure profile resulting from the original approach presented by Monaghan [7] is highly oscillatory and thus unacceptable. Although the density fluctuations are within a few percent only, the amplitude of the spurious oscillations in the pressure field is remarkable. The cause of these spurious pressure fluctuations lies in numerical instabilities but they are further amplified by the stiff equation of state, the so-called *Tait* equation.

An alternative and more general formulation of the SPH scheme derives from the work of Vila [2], which uses approximate Riemann solvers to evaluate a numerical flux for each couple of interacting particles. In contrast to the approach presented in [7], the Vila formulation produces a monotone pressure field and this latter formulation of the SPH method is also stable. Unfortunately, it is too diffusive and hence it can not be used to simulate violent free-surface flows, which are at the focus of our attention. Even the Godunov flux, based on the *exact* Riemann solver, does not help to remedy the excess of numerical viscosity inherent in the SPH formulation of Vila.

In conclusion, neither of the two above-mentioned SPH approaches is able to produce simultaneously accurate solutions for the pressure field and also for the free-surface. Therefore, it is necessary to construct a new stable and more accurate SPH scheme.

II. The New SPH Formulation

The new formulation proposed in this paper uses an explicit third order Runge-Kutta scheme in time to improve the linear stability of the method, following a similar time evolution procedure of Oger *et al*. [5]. Since the high order TVD Runge-Kutta time discretization initialized by Shu and Osher [6] has been extremely successful in solving nonlinear hyperbolic conservation laws with total variation diminishing (TVD) finite difference and finite volume schemes, the third order TVD Runge-Kutta scheme [6] has been implemented. However, the explicit high order time integration is not enough to stabilize the numerical pressure

SPH SIMULATION OF A FLOATING BODY FORCED BY REGULAR WAVES

Sauro Manenti, Andrea Panizzo
Dipartimento di Idraulica trasporti e Strade
University of Rome "Sapienza"
Rome, Italy
sauro.manenti@uniroma1.it

Piero Ruol[1], Luca Martinelli[2]
1 University of Padua
2 University of Bologna
Italy

***Abstract*— The dynamic analysis of a floating body response under the wave forcing involves complex aspects related with fluid-structure interaction. In this work a numerical approach to solve such a problem is proposed by means of the SPH code. Basic aspects regarding linear wave generation in a flume and free-damped oscillations of a one degree of freedom (DOF) floating box have been early investigated. Finally the fully-coupled dynamic response of a floating breakwater forced by regular waves has been analyzed and compared with experimental measurements.**

I. INTRODUCTION

The analysis of the dynamic behaviour of a floating body under wave forcing represents one of the main concerns in coastal and ocean engineering. In nearshore environments where the wave field spectrum is characterized by a limited peak period (e.g. lakes, bays or tidal lagoons) prediction of the floating body response concerns mainly with the analysis and design of floating breakwaters (FBs).

The engineering approach to this problem becomes rather complex due to the fluid-structure interaction (waves force the body motion and, in turn, its movements affect the incoming field through radiated water disturbances); thus the adoption of sophisticated numerical models is required for correct prediction of the FBs performances.

Unfortunately the numerical tools currently adopted are based upon restrictive hypotheses that seems to be inconsistent with the actual behaviour of these structures (e.g. infinitesimal height of the incident waves, small oscillation around the static floating position etc.).

In this context the use of the SPH code seems to offer promising results as its lagrangian approach along with the assumption of a meshless interpolant scheme allow to adapt to possible high strains experienced by the fluid and the floating body; even surface separation (induced by the overtopping) is allowed thus overcoming most of the critical aspects of other numerical techniques traditionally adopted for modelling the above mentioned problem [6].

In subsequent sections are illustrated the most significant results obtained by means of the SPH code [4]. In particular the following topics are discussed:

1. numerical generation of linear wave in a flume;
2. free-damped oscillation of a floating body with 1-DOF;
3. dynamic response of a 1-DOF floating breakwater under regular waves (2-D model).

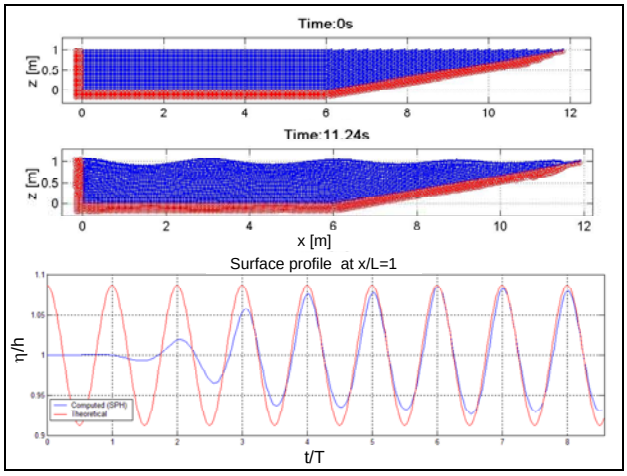


Figure 1. Wave flume and time surface elevation

II. GENERATION OF REGULAR WAVES IN A FLUME

The early analysis has been carried out by means of the SPH code for simulating free-surface linear waves generated in a flume.

The facility is made up of two parts (Fig. 1): the former, which is close to the wave-maker paddle, has a constant depth (h=1.0m) and a length of 6.0m; the latter, toward the shore boundary, has a uniform slope (1:6) and the same length of the previous one.

Wave impact simulations using incompressible and weakly-compressible SPH models

Jason P. Hughes*, David I Graham*, Dominic E. Reeve**, Philip W. James*, Andrew J. Chadwick**, John Lawrence**

*School of Mathematics and Statistics, University of Plymouth, UK
**School of Engineering, University of Plymouth, UK

1. INTRODUCTION

Coastal structures such as breakwaters, harbour walls or cliffs are often subject to wave impacts. Such structures can suffer severe damage during extreme storms (see Figure 1, for example). In laboratory experiments, wave impacts have been characterised as 'low-or high-aeration', depending upon the shape of the wave as it reaches the structure (Bullock *et al.* [1]). Low aeration impacts are characterised by a sharp peak at impact, followed by quasi-hydrostatic pressures as the wave runs up then down the vertical wall (see Figure 2). This includes the so-called 'flip-through' case where no air is entrapped as the wave breaks. When an air cushion is entrapped as the wave overturns, a high-aeration impact can occur. In this case, impact pressures can be greatly in excess of the low-aeration values and characteristic pressure oscillations due to the acoustic vibration of the entrapped air cushion follow the impact peak.

Figure 1: Alderney breakwater (photography: Ilona Soane-Sands)

Recent work by the authors [2] has shown that incompressible smoothed particle hydrodynamics (I-SPH) can provide excellent simulations of unaerated wave impacts on coastal structures. In particular, it is capable of modelling the flip-through at impact and resulting extreme vertical accelerations and jet plume up the wall. I-SPH was also shown to cope (qualitatively, at least) with the jet breakup and resulting splashing as the jet returns to the water body. However, it was less successful when applied to problems involving air entrainment. It is to be expected that at least part of the problem with the I-SPH computations is due to the absence of a gaseous phase in the computations.

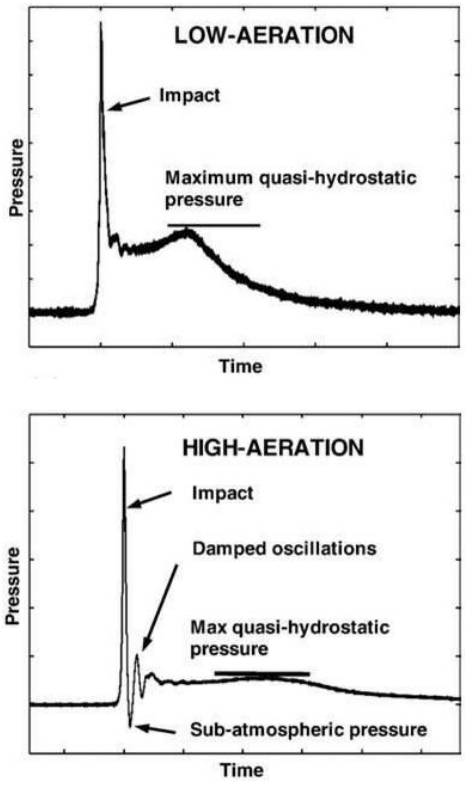


Figure 2: Pressure profiles for 'low-aeration' and 'high-aeration' wave impacts (from Bullock *et al*, [1]).

The team at Plymouth are in the process of developing such a model. As part of the development, a choice must be made between using incompressible (I-SPH) or weakly-compressible versions (WCSPH) of SPH. The present paper reports a comparison between these two model types for wave impacts. Several different formulations of both I-SPH and WCSPH are used. The calculated pressure profiles at

Coastal flow simulations using an SPH formulation modelling the Non-Linear Shallow Water equations

M. de Leffe, D. Le Touzé and B. Alessandrini
Laboratoire de Mécanique des Fluides (CNRS UMR6598)
Ecole Centrale de Nantes
1 rue de la Noë - 44300 Nantes - France
Matthieu.de-Leffe@ec-nantes.fr - David.LeTouze@ec-nantes.fr - Bertrand.Alessandrini@ec-nantes.fr

***Abstract*—In the present paper, an SPH model of the shallow water equations is presented. The objective of this model is to perform flooding simulations of complex three-dimensional bathymetries of a wet bottom and a dry land. Its implementation is described, including especially the procedures to be able to follow the fluid domain expansion during flooding simulations, namely an anisotropic kernel with variable smoothing length is used as well as a periodic redistribution of the particles.**

A number of validation tests are made. The model results are first checked on the case of a dam break on a flat dry bottom, and the propagation of a solitary wave. Then more complex three-dimensional cases are simulated and compared to other models and experiments, e.g. a dam break flooding a slope of complex shape, and a solitary wave running up an island.

I. Introduction

Flooding of dry coasts due to rough-sea breaking waves or tsunamis is a major concern in coastal engineering. Similarly, land flooding in case of hydraulic dam break is another example where a large area of dry land is invaded by water. To simulate such events, mesh-based methods are rather inadequate since they would require the use of huge Eulerian meshes distributed over the entire possibly flooded domain. The use of a lagrangian meshless method such as SPH would conversely allow for tracking the flow evolution, without requiring a spatial discretization of dry parts of the domain.

Still, standard SPH simulation of such events whose space and time scales are much larger than the ones usually simulated by SPH, would exhibit prohibitive computational costs. An alternative is to use the SPH scheme to model the non-linear shallow water equations, as proposed by Rodriguez-Paz & Bonet [5] and Ata & Soulaimani [9], and extended by Panizzo et al. [2]. Each particle then represents a water column whose height is linked to the smoothing length through its mass conservation. These particles are still followed in their Lagrangian motion, but the number of dimensions of the problem is reduced by one thanks to the shallow water modelling. Further, the flow is considered as incompressible and time steps can then be much wider than in standard SPH. Finally, to preserve the explicit nature of the SPH scheme a Newton-Raphson scheme is applied to enforce the link between interpolation kernel extension, water depth and particle mass.

The two- and three-dimensional versions of the shallow water SPH model have first been implemented following the mentioned literature. And the test cases proposed in [5] and [2] have been reproduced, namely a 2D dam break on a slope and a 3D cylindrical patch of water.

Then the model has been applied to more complex flows. However, in three dimensions the model accuracy is degraded by the formation of anisotropic repartitions of particles. It is especially the case when the flow evolves mainly in one direction, e.g. a 3D dam break between lateral walls. To tackle this problem, an elliptic kernel formulation has been derived, so as to be able to better follow the flow evolution. Nonetheless, numerical difficulties have arisen on complex 3D cases, due to the stretching of particle kernels when the water depth is decreasing, as it is the case when the flow is wetting the coast. A particle redistribution procedure has thus been implemented, allowing for periodically change the particle mass distribution so as to recover a more regular space occupation of the wetted domain. With these tools, the obtained formulation is robust and able to simulate complex situations, such as a flow invading a coast of complex 3D shape, or the flow past an island. Good comparisons are obtained against experimental and high-order Boussinesq model results.

II. Physical model

In the present paper, the shallow water equations are modelled. In conservative form they read

$$\begin{cases} \dfrac{\partial h}{\partial t} + div(h\mathbf{v}) = 0 \\ \dfrac{\partial h\mathbf{v}}{\partial t} + \boldsymbol{\nabla} \cdot (h\mathbf{v} \otimes \mathbf{v} + \dfrac{gh^2}{2} I) = -gh(\mathbf{grad}(b) + \mathbf{S}_f) \end{cases} \tag{1}$$

where $\mathbf{v}$ is the velocity, $h \geq 0$ indicates the water depth, g is the acceleration due to gravity, b is the bottom elevation, and $\mathbf{S}_f$ is the friction defined by

$$\mathbf{S}_f = \frac{n^2 \mathbf{v} \|\mathbf{v}\|}{h^{4/3}} \tag{2}$$

in which n denotes Manning's roughness coefficient.

We can rewrite the previous system of equations in the following form

$$L_v(\phi) + div(F(x,t,\phi)) = S \tag{3}$$

Simulation of Vortex Spindown and Taylor-Green Vortices with Incompressible SPH Method

Rui Xu[1], Charles Moulinec[2], Peter Stansby[1],
Benedict D. Rogers[1], Dominique Laurence[1]
[1]The School of MACE, University of Manchester, UK
[2]SFTC Daresbury Laboratory, WA4 4AD, UK

***Abstract*— Several points about wall boundary conditions in incompressible SPH(ISPH) method are discussed here. With the aid of kernel normalization, the vortex spindown case is successfully simulated. The difficulties in the simulation of Taylor-Green vortices are discussed. Remeshing is used to overcome these difficulties. However, for the remeshing method, the artificial diffusion can be a problem. And the free surface definition would not be obtained.**

I. Introduction

The weakly compressible SPH (WCSPH) method is widely used in the flow simulations. However, for WCSPH, the pressure field strongly depends on a state equation, which introduces large pressure fluctuations. Moreover, to satisfy the CFL time-step constraint, with a sound speed in the CFL number expression, the time step is limited to a very small value. Compressibility causes the sound wave reflection at the boundaries. All these drawbacks make an incompressible algorithm desirable to couple the pressure and velocity in SPH.

An approach to modeling free surface, incompressible flows using a fully Lagrangian technique was proposed by Koshizuka et al. [4]. In this method, a penalty-like formulation was employed to adjust the pressure where density variations occurred. In 1998, Koshizuka et al. [7] presented another incompressible method. A pressure Poisson equation was solved instead of a penalty method with source term proportional to density variations. In 1999, Cummins et al. [2] applied the projection method in SPH, which projects the intermediate velocity fields onto a divergence-free field and a curl-free field respectively. Shao et al. [6] used an incompressible method, similar to that in [7], to describe the free surface in dam breaking case.

An incompressible SPH (ISPH) method is examined here, based on the code SPHysics, where the pressure field is solved by a pressure Poisson equation (PPE), instead of using the state equation. Similar to other ISPH schemes, the total computing time is highly reduced because higher time steps can be handled, and smoother pressure fields are obtained. The test cases vortex spindown and Taylor-Green vortices are simulated. With the application of the normalized fifth-order kernel, the gradient and divergence calculations are discussed to satisfy the wall boundary demands.

II. Incompressible SPH

A. Projection Method

The projection presented here employs a first-order Eulerian time step – higher order schemes are easily implemented. The particle positions, r_a^n, are advected with velocity u_a^n to positions r_a^*,

$$\vec{r}_a^* = \vec{r}_a^n + \Delta t(\vec{u}_a^n). \tag{1}$$

An intermediate velocity u^* is calculated based on the momentum equation without pressure gradient term,

$$\frac{\vec{u}^* - \vec{u}^n}{\Delta t} = \nu\nabla^2\vec{u}^n + \vec{F}^n. \tag{2}$$

Solving the Pressure Poison equation(PPE) Eq.3, the pressure at time $n+1$ can be obtained,

$$\nabla^2 p^{n+1} = \frac{\rho}{\Delta t}\nabla\vec{u}^*. \tag{3}$$

The velocity at time $n+1$, $\vec{u}^{n+1}$ will be obtained from the modification of $\vec{u}^*$by the gradient of p^{n+1}. Therefore,

$$\vec{u}^{n+1} = \vec{u}^* - \frac{\Delta t}{\rho}\nabla p. \tag{4}$$

Afterwards, the particle position is advanced in time,

$$\vec{r}_a^{n+1} = \vec{r}_a^n + \Delta t\left(\frac{\vec{u}_a^{n+1} + \vec{u}_a^n}{2}\right). \tag{5}$$

B. The Solvability Constraint for Poisson Equations

In the code, the mirror particle boundary condition is used to achieve homogeneous Neumann boundary conditions for Poisson equations. The mirror particle wall boundary method is first proposed in [2]. The application in incompressible SPH is first presented by Cummins [2].

The homogeneous Neumann boundary conditions are used for the Poisson equation at the wall. Therefore, a compatible constraint should be satisfied for the solvability of Eq.3, in a fully wall-bounded case. This constraint can be deduced from the integral of the left hand side of Eq.3 in the whole space.

$$\int_\Omega \triangle P d\Omega = \int_\Omega \nabla\cdot(\nabla P)\, d\Omega = \int_{\partial\Omega} \nabla P\cdot\overrightarrow{n}\, dS = 0 \tag{6}$$

A constant-density approach for incompressible multi-phase SPH

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

***Abstract*— A constant-density approach, which corrects the density errors by adjusting the half-time-step velocity with exact projection, is proposed for the multi-phase SPH method developed in our previous work (X. Y. Hu and N. A. Adams, An incompressible multi-phase SPH method. J. Comput. Physics, 227:264-278, 2007). As no prescribed reference pressure is required, the present approach introduces smaller numerical dissipations and, for the first time, allows to simulating flows with large density ratios by a projection SPH method. Numerical examples for Taylor-Green flow, capillary waves and for Rayleigh-Taylor instability are presented and compared to theoretical solutions or references from literature.**

I. Introduction

In our previous work [6], an incompressible multi-phase SPH method is proposed for flows with sharp density and viscosity interfaces. To handle the problem of accumulating density error and particle clustering, which can occur to the projection SPH method [2] [11] [10], a constant-density constraint has been proposed by correcting the particle position with an iterative approach. Although numerical experiments demonstrated that this approach is accurate and efficient, its convergence behavior of the iterative scheme is difficult to predict due to the non-linear relation between particle position and particle density. Another issue with the method proposed in [6] is that for stable solution and good accuracy an optimal reference pressure, which is superimposed on the pressure distribution obtained from constant-density and velocity-divergence free constraints, needs to be chosen and prescribed.

II. Method

In this paper, we propose a new technique for enforcing the constant-density constraint by correcting the half-time-step velocity of a particle. Other than non-linear iteration, this velocity is obtained by an one-step correction based on an exact projection approach. Since only a linear system is involved, the convergence behavior of the iterative scheme becomes more predictable. With exact projections, no emperical reference pressure is required to be superimposed during the constant-density constraint step, and the reference pressure for the velocity-divergence-free constraint can be treated in a simple way. Numerical experiments show that the present methods to obtain the reference pressure introduces smaller error and allow to simulating flows with large density ratios. Since previous time-step criteria based on the reference pressure in [6] are not applicable, new criteria based on the predicted density error and the accumulated density error during previous steps are introduced.

III. Numerical examples

The following two-dimensional numerical examples are provided to validate the proposed incompressible multi-phase SPH method. For all cases a quintic spline kernel [9] is used as smoothing function. A constant smoothing length, which is kept equal to the initial distance between the neighboring particles, is used for all test cases. The discretizations of viscous force and surface force follow Ref. [5]. As elliptic solver a diagonally preconditioned GMRES (q) method is used. If not mentioned otherwise, no-slip wall boundary conditions are implemented following the approach of Cummins & Rudman [2].

A. Two-dimensional Taylor-Green flow

The two-dimensional viscous Taylor-Green flow is a periodic array of vortices, where the velocity

$$\begin{aligned} u(x,y,t) &= -Ue^{bt}\cos(2\pi x)\sin(2\pi y) \\ v(x,y,t) &= Ue^{bt}\sin(2\pi x)\cos(2\pi y) \end{aligned} \tag{1}$$

is an exact solution of the incompressible Navier-Stokes equation. $b = -\frac{8\pi^2}{\mathrm{Re}}$ is the decay rate of velocity field. We consider a case with $\mathrm{Re} = 100$, which has been used to test different incompressible SPH methods [1] [3] [6]. The computation setup is the same as that of [6]. The computation is performed on a domain $0 < x < 1$ and $0 < y < 1$ with periodic boundary conditions in both directions. The initial particle velocity is assigned according to Eq. (1) by setting $t = 0$ and $U = 1$. Same as in [6], the initial particle configuration is taken from previously stored particle positions (relaxed configuration). In order to study the convergence properties the calculation is carried out with 450, 900, 3600 particles, respectively.

Figure 1 shows calculated positions of particles and vorticity profile, respectively, at $t = 1$ with 3600 particles. It can be observed that the particle distribution is in quite good agreement with the results of [6] (their Fig. 1). It can be also observed that a uniform particle distribution without clustering is produced. The current SPH simulation recovers the theoretical solution quite well with somewhat larger errors in regions close to the centers of vortex cells. Fig. 2a shows the calculated decay of the maximum velocity of the flow with different resolutions. It can be observed that for low resolution results, the predicted Reynolds number is slightly smaller than

Permeable and Non-reflecting Boundary Conditions in SPH

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

***Abstract*— Inflow and outflow boundary conditons are essential for the application of computational fluid dynamics to many engineering scenarios. In this paper we present a new approach to the Smoothed Particle Hydrodynamics (SPH) simulation of flow through inlet and outlet boundaries. Each permeable boundary is associated with an inflow or outflow zone outside the domain, in which particles are created or removed as required. The analytic boundary condition is applied by prescribing the appropriate variables for particles in an inflow or outflow zone, and extrapolating other variables from within the domain. Non-reflecting boundary conditions, described in the literature for mesh-based methods, can be implemented within this framework. Results are presented for simple one-dimensional flows, quasi-one-dimensional compressible nozzle flow, and two-dimensional nearly incompressible flow around a cylinder at Reynolds numbers of 40 and 100. Computed velocity fields in all cases are in good agreement with analytical results or results from conventional computational methods. However, pressure distribution on the cylinder surface has significant errors. The different wake structures for flow about the cylinder are correctly predicted. Furthermore, the cylinder flow is successfully modelled with a non-uniform particle distribution to reduce computational cost.**

I. Introduction

In many important questions in engineering fluid dynamics, the flow field extends over a very large spatial domain, but the phenomena of interest are restricted to a relatively small region. Examples include external aerodynamics (where the local flow is governed by conditions in the notional "free stream"), turbomachinery, pipes and channels, and blood flow. These problems can be simulated successfully if the computational model is restricted to a small domain of interest, with boundary conditions that provide reasonable models for the interaction of fluid in the domain with fluid in the exterior surroundings. In particular, boundary conditions must allow fluid to enter and leave the local computational domain. This approach is well established in Eulerian mesh-based methods.

However, the use of a Lagrangian frame of reference places Smoothed Particle Hydrodynamics (SPH) at a disadvantage. The Eulerian frame of reference naturally describes a stationary spatial region with inflow and outflow of fluid, while the Lagrangian reference frame follows fluid particles which may spend only a short time traversing the spatial region of interest. Flow into and out of a domain can be modelled in SPH with spatial periodicity, but this is accurate only for very special cases, and its use even as an approximation is severely restricted. Furthermore, periodic boundaries cause particle distributions to be recycled through the domain, so that a perturbed particle distribution may degrade without limit over time. While SPH has been successful in the modelling of unbounded (e.g. astrophysical) processes and wall-bounded unsteady flows (e.g. dam-break and wavemaker flows), it has not been widely used for problems with inflow and outflow. To extend the scope of SPH to a wider range of engineering flows, it is essential to incorporate inflow and outflow boundaries.

This paper presents a method for physically accurate implementation of inflow and outflow boundaries in SPH. The method incorporates a framework for the insertion of new particles at an inflow boundary and the removal of particles at an outflow boundary, while ensuring that appropriate mathematical boundary conditions are enforced. Following some background material in sections II and III, this framework is described in section IV, and results are shown for a simple 1D wave propagation problem. In section V, a non-reflecting boundary condition due to Giles [1] is described, and its implementation in the new SPH framework is shown. In section VI, results are presented for the new boundary conditions for compressible flow through a quasi-one-dimensional nozzle and for flow around a cylinder at Reynolds numbers of 40 and 100.

II. Discretisation

Throughout this paper, the standard SPH formulations given below in (1–3) are used to discretise the Euler and Navier-Stokes equations. These discretisations are reviewed in detail by Monaghan [2] and others.

$$\frac{\mathrm{D}\rho_a}{\mathrm{D}t} = \rho_a \sum_b (\mathbf{u}_a - \mathbf{u}_b) \cdot \tilde{\nabla} \tilde{W}_{ab} \frac{m_b}{\rho_b} \quad (1)$$

$$\frac{\mathrm{D}u_a}{\mathrm{D}t} = -\sum_b m_b \left(\frac{p_a}{\rho_a^2} + \frac{p_b}{\rho_b^2} + \Pi_{ab} + R_{ab} \right) \cdot \tilde{\nabla} \tilde{W}_{ab} \quad (2)$$

$$\frac{\mathrm{D}e_a}{\mathrm{D}t} = \frac{1}{2} \sum_b m_b \left(\frac{p_a}{\rho_a^2} + \frac{p_b}{\rho_b^2} \right) (\mathbf{u}_a - \mathbf{u}_b) \cdot \tilde{\nabla} \tilde{W}_{ab} m_b \quad (3)$$

Here p is pressure, ρ is density, $\mathbf{u}$ is velocity and e is internal energy. This system is closed by the ideal gas equation of state. W_{ab} denotes a kernel function $W(r, h)$ evaluated with $r = |\mathbf{x}_a - \mathbf{x}_b|$, the distance from particle a to particle b. h is the smoothing length. Throughout this paper, the cubic spline

Experiences of SPH with the lid driven cavity problem

Andrea Panizzo, Tatiana Capone, Salvatore Marrone
DITS, Department of Hydraulic, Transportation and Roads
"Sapienza" University, Rome, Italy
andrea.panizzo@uniroma1.it, tatiana.capone@uniroma1.it, salvatore_marrone@fastwebnet.it

***Abstract*— This work sheds lights on the application of the SPH method to simulate a flow in a cavity driven by a moving top lid. The considered problem is a good benchmark to experience lots of aspects of SPH, main actors in the numerical simulations being the used boundary conditions and the numerical techniques to introduce viscous and turbulent stresses in the simulated momentum equations. The work presents simulations up to Re=1000.**

I. The lid driven cavity flow

The driven cavity problem at hand is that introduced by Ghia et al. [3], and consists of fluid in a square cavity with top lid moving at velocity U_{lid}, as sketched by Figure 1. This problem has been recently considered as test case for SPH models by SPHERIC. The Reynolds number of the driven flow is defined as Re=U_{lid}L/ν. By varying Re, different shapes and features of the generated vortex result, and additional vortexes at the bottom angles and at the top left angle may appear. We here refer to results of the work of Ghia et al. [3] as a reference to compare velocity profiles and the shapes of the driven vortex obtained with SPH. These reference numerical simulations assumed no slip conditions at the three cavity boundaries, while results for the horizontal and vertical components of the velocity vector **v** (U and V respectively) were referred to the vertical and the horizontal sections trough the geometric center C_c of the cavity. The center of the main central vortex C_v generally does not coincide with C_c.

II. Details of the used SPH code

All the performed numerical simulations have been carried out considering a square cavity with side L=1.0 m, velocity of the top lid U_{lid}=1.0 m/s, and density of the fluid in the cavity ρ=1 Kg/m^3. Thus, different Re numbers have been obtained by varying the fluid viscosity μ.
The fluid in the cavity has been considered weakly compressible, the pressure P varying as a function of density variation ρ according to the equation of state (Batchelor, [1]):

$$P = \frac{c_s^2 \rho_o}{\gamma}\left[\left(\frac{\rho}{\rho_o}\right)^{\gamma} - 1\right] \tag{1}$$

with γ=7. In this work, the speed of sound was set to c=10.0 m/s, which is ten times the maximum speed in the cavity. The fluid has been modelled using 40x40 particles initially placed in a lattice of square cells with cell size inh=0.025 m. We used a constant smoothing length approach, with h=1.3inh.

Continuity and momentum SPH equations are the classic ones introduced by the milestone work of Monaghan [4]. In particular, the momentum equation written in the SPH formalism reads:

$$\frac{d\mathbf{v}_i}{dt} = -\sum_{j=1}^{N} m_j \left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2} + \Pi_{ij}\right)\nabla_i W_{ij} \tag{2}$$

where the term Π_{ij} introduces the fluid viscosity. We considered the formulation proposed by Monaghan ([5], [6]), which reads:

$$\Pi_{ij} = -\frac{K \upsilon_{sig} \mathbf{v}_{ij} \cdot \mathbf{r}_{ij}}{\bar{\rho}_{ij} |\mathbf{r}_{ij}|} \tag{3}$$

where $\mathbf{v}_{ij} = \mathbf{v}_i - \mathbf{v}_j$, with similar notation for $\mathbf{r}_{ij}$, $\bar{\rho}_{ij}$ is the average density of particles i and j, $\upsilon_{sig} = c_i + c_j$, c_i being the speed of sound of particle i. In the continuum limit this viscous term is equivalent to a shear viscosity (in this study the fluid is weakly compressible thus the bulk viscosity is negligible) and its value, using the cubic spline kernel, is:

$$\nu = \frac{15}{112} K \upsilon_{sig} h \tag{4}$$

Therefore, given the value of the kinematic viscosity ν and the resolution length h, the constant K is determined. The advantage of (3) is that it guarantees both linear and angular

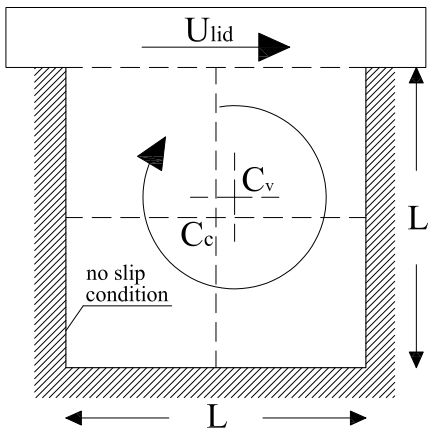


Fig. 1. Sketch of the driven cavity problem

Forced Two-Dimensional Wall-Bounded Turbulence Using SPH

Martin Robinson
School of Mathematical Sciences
Monash University
Clayton, Melbourne, Australia
martin.robinson@sci.monash.edu.au

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

***Abstract*— SPH was used to simulated forced, wall-bounded 2D turbulence in a square box. Initial results using the Cubic Spline kernel found significant particle disorder and clumping, which contributed to a large excess of kinetic energy at the smaller length scales of the simulation. This excess in turn caused a large increase in the viscous dissipation at these scales. Possibly related to this, but not proven to be, is the fact that kinetic energy does not build up at the longest wavelength, contrary to theoretical predictions and published experiments. The particle clumping was found to be caused by the tendency of particles to separate themselves at a distance equal to the location of the kernel's spline point. Using a modified Cubic Spline with the spline point set to the initial particle spacing eliminated all clumping, but had the disadvantage of decreasing the falloff of the kernel's Fourier transform. The use of the Wendland kernel gave the best results, also eliminating particle clumping while having a faster Fourier transform falloff than the Modified Cubic Spline. However, even this kernel did not eliminate the excess kinetic energy at small scales or the lack of kinetic energy buildup at larger scales. Finally, a global measure of simulation accuracy is proposed based on the homogeneous nature of the turbulence.**

I. Introduction

As a benchmarking application for SPH, two-dimensional, isotropic, homogeneous turbulence has a number of advantages and one rather large disadvantage. Thanks to the chaotic and highly non-linear nature of turbulence, it is a challenging simulation for any numerical method and for SPH in particular. SPH can be sensitive to particle disorder, something which turbulence is excellent at producing. The method also tends to suffer from excess numerical dissipation, which could disrupt the link that exists in 2D turbulence between the small dissipative scales and the large scale energetic motions of the fluid.

At the same time, while the turbulence is difficult to simulate, it provides many interesting avenues to probe what is happening in the simulation. For continually forced turbulence, where the particle dynamics are homogeneous and isotropic, spatial and temporal averages, autocorrelations and other statistical tools can be used and compared with the significant numerical, theoretical and experimental work on the subject of turbulence. But perhaps the most dominant form of analysis used in this subject would have to be the use of the Fourier transform to separate and compare the large range of turbulent length scales. This too is only useful for spatial homogeneous flows, and could provide new insights into SPH simulations.

The chief disadvantage of simulating turbulence is of course the large number of particles needed to fully resolve the wide range of important length scales. This is mitigated somewhat by only considering 2D turbulence but is still a problem given that the forcing scale has to be well removed from the domain size and the smallest dissipative scale, and that the ratio between the forcing and smallest length scales is proportional to $Re^{-0.5}$. For this paper we have used $Re \approx 1400$ and integrated over a time period of 50 secs, which has required the calculation of 250,000 timesteps using 300x300 particles.

But, thanks to the rapid rise in the use of computer clusters and multiple CPU desktop computers, a parallel SPH implementation can overcome these difficulties. Standard quasi-compressible SPH is a local method and is easily and efficiently partitioned among multiple processors. Once the parallel code is written, it scales up extremely well to an arbitrary number of processors, limited only by the number of CPUs that are available.

There have been a number of SPH turbulence models proposed in the literature, based on versions of existing turbulence models (e.g. Monaghan [1], Violeau [2]). Examples of Direct Numerical Simulations (DNS), where the full range of scales is modelled, are fewer. Mansour [3] has recently completed a PhD thesis on SPH turbulence, both modelled using the α-SPH model and DNS. Our work attempts to investigate questions raised by this work, as well as extending it to wall-bounded turbulence.

II. SPH Method

This work uses the standard quasi-compressible SPH method outlined in the 2005 review paper by Monaghan [4]. The viscosity term is calculated using the Monaghan [5] form, which is based on the dissipative terms used in Riemann Solvers and conserves linear and angular momentum. The density for each particle is found by integrating the continuity equation.

The particle's position and velocity are integrated using the Verlet second order method. In order to preserve the reversibility of the simulation (in the absence of viscosity), $d\rho/dt$ is calculated using the particle's position and velocity at the end, rather than the middle, of the timestep.

Modelling a fish passage with SPH and Eulerian codes: the influence of turbulent closure

Damien Violeau, Réza Issa, Sofiane Benhamadouche and Khaled Saleh
LNHE & MFEE departments
Electricité de France R&D
Chatou, France
damien.violeau@edf.fr

Jacques Chorda and Marie-Madeleine Maubourguet
Groupe Hydrologie de Surface et Ecohydraulique
Institut de Mécanique des Fluides de Toulouse
Toulouse, France

***Abstract*—A flow in a fish passage is modelled with three industrial codes based on different numerical approaches: SPH, Finite Volumes and Finite Elements. Various turbulent closures are used and compared. It appears that significant discrepancies exist between the models, in terms of mean velocities as well as turbulent kinetic energy predictions.**

I. INTRODUCTION

Fish passes are more and more used in the context of environmental sustainability of large river dams. They allow fish migration despite the presence of large river structures. The design of such a device requires the knowledge of the spatial distribution of the mean flow but also of turbulent kinetic energy (TKE). A particular type of fish passage, namely "vertical slot fish passes", will be considered in this paper as a benchmark for codes based on various numerical approaches. For the CFD modeller point of view, the interest of this type of flow is the complexity of Reynolds-averaged velocity distribution, subject to a high rate-of-strain and mean vorticity. It is not obvious that traditional turbulent closure models are capable of reproducing this complexity with a satisfactory agreement. The results of three codes based on various numerical techniques (SPH, Finite Volumes, Finite Elements) will be presented here, focusing on the effect of turbulent closure on the quality of predictions.

II. PHYSICS OF THE PROBLEM

A. *The vertical slot fish passage*

The fish pass considered here is visible on figure 1. It consists of series of pools separated by vertical slots allowing fish to swim against the current from one box to the next. The walls are designed in order to ensure a rest area in each pool. The exact geometry of a pool is presented on figure 2. The considered slope of the bed is $1/10^{th}$, and the flow rate is 0.736 m^3/s.

It is seen that the configuration of the walls lead to a strong turbulent jet immediately past the slot, then making large clockwise and counter-clockwise recirculation patterns. Not only the prediction of the mean flow is important for the design of such a water work, but also is capital the correct prediction of the spatial distribution of TKE, which is expected to stay under a certain limit in the rest area.

To the first approximation, the flow can be considered as steady, two-dimensional and periodic in the x–direction (see figure 2). However, the first two assumptions may be subject to discussion. Experimental observations have pointed out some possible instabilities in the motion of the jet. Besides, it has been known for a long time by turbulence modellers that the turbulent structure of such a complex flow cannot be strictly two-dimensional. Thus, the numerical models presented below will be mainly 2-D, but a 3-D case will also be investigated for a better prediction. In all 2-D approaches, the 3-D flow rate and the water depth mentioned above lead to a 2-D flow rate of 0.657 m^2/s.

Figure 1. Global view of a vertical slot fish passage.

Conventional SPH Revisited

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

Abstract - In the conventional SPH method, the time-dependent partial differential equations which describe conservation of mass, momentum and energy, and which together with a constitutive equation describe the motion of a continuum, are semi-discretised using kernel interpolation. These can be divided into two main groups namely variationally based and strong form based derivations. However, the resulting equations are not the usual form of the discretised SPH equation often found in literature [Libersky, Petschek, Carney, Hipp and Firooz (1993), Morris (1995), Richard and Papaloizou (1994)]. In order to ensure that the semi-discretised equations maintain conservation properties locally additional terms are introduced into the conservation of mass equation and the conservation of linear momentum equation. These changes of the conservation equations were performed with variable rigor.

The process of SPH discretisation of conservation equations is revisited in this paper with the intention to provide interpretation of the terms introduced to provide Galilean invariance and improve the rigor of the process. In the present study the conservation equations are derived in a moving coordinate frame (for a moving control volume) following the same steps as in the conventional SPH method. The time-dependent partial differential equations are semi-discretised to a time dependent ordinary differential equations using the following steps:

- **multiplying both sides of the equation with a kernel function;**
- **integrating all the products over the entire domain;**
- **linearising the integrals if needed;**
- **Integrating each term using the technique of integration by parts, this operation moves spatial derivatives from operating on the field variables to operating on the interpolation kernel;**
- **disregarding boundary terms , and**
- **converting all the integrals into a summation over a set of discrete particles.**

1. The Conventional SPH Method

A. *Kernel Approximation*

The SPH method is based on the kernel interpolation which in its discrete form uses randomly distributed interpolation points, with no fixed connectivity, to approximate field variables and their spatial derivatives. In SPH, spatial derivatives are calculated by analytical differentiation of kernel function. To illustrate this consider a continuum represented by a set of interacting particles. (Fig. 1)

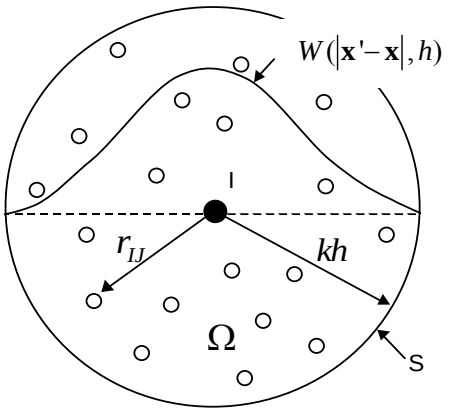


Figure 1. Kernel support and neighbouring particles in the discrete kernel interpolation.

Each particle *I* interacts with all other particles *J* that are within a given distance (equal to 2*h* in the case of a B-spline kernel) from it. Parameter *h* (the smoothing length) determines the spatial resolution of the calculations. The interaction between *I* and *J* is weighted by the smoothing (or kernel) function $W(|\mathbf{x}'-\mathbf{x}|,h)$. Using this principle, the value of a continuous function can be approximated at any

Riemann solvers and efficient boundary treatments: an hybrid SPH-finite volume numerical method.

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

Etienne Parkinson
Hydraulic research department
ANDRITZ VATECH Hydro AG
Vevey, Switzerland

***Abstract*— Despite its recent attractivity in many fields of the numerical simulation, the numerical method SPH still faces some major challenges which restrict its use in real applications. Among these drawbacks, boundary conditions are the most obvious. Considering free surface flow simulation, the point of view adopted in this paper is that most of these difficulties (stability, accuracy, boundary conditions) can be overcome by a shift in the flow description. The standard SPH method is built on the non conservative form of the Euler equations written in a pure lagrangian frame. Following previous work from Vila, an *Arbitrary Lagrange Euler* (ALE) description together with a weak form of the Euler equations are chosen. This leads to an hybrid method directly linked to the finite volumes method. It is then possible to introduce finite volumes flux schemes into the hybrid method, in particular upwind schemes like the Godunov scheme. This brings a major improvement to the stability of the method. A second order scheme similar to the well-known MUSCL scheme of Van Leer is set to reduce numerical diffusion and improve accuracy. Finally, the volumic upwind scheme is extended to boundaries treatment through the introduction of boundary fluxes and partial Riemann problems. The viability of this hybrid method is proved on several test cases representative of water impacts on solid walls.**

I. Introduction

The SPH method for the simulation of fluid flows is usually presented as a whole package, in which the physical description of the flow and the numerical technique used for discretization are given. In practice, inviscid fluid flows are described by the system of Euler equations in a pure lagrangian and non conservative form. The system is closed using a pseudo-compressible state equation for the fluid, usually the Tait equation. Spatial derivatives are then numerically approached using the classical kernel then particle approximations. This is the standard SPH method and it suffers from several weaknesses, namely the accuracy, the stability and the correct treatment of boundary conditions.

In order to remedy this, some techniques have been set up. Stability is treated through a numerical viscosity term that stabilizes the centered standard SPH scheme. Spatial accuracy can be improved by correction techniques like renormalization or MLSPH. Boundary conditions are more tricky, and avalaible treatments essentially concern wall boundary conditions. Among them, the ghost particle method appears to be the most efficient, but its application in the case of complicated shaped bodies is not satisfactory.

In this paper we propose an approach which can bring efficient answers to these questions. It is based on work from Vila ([9]). We first change the way the flow is described by adopting an ALE (Arbitrary Lagrange Euler) point of view. This facilitates the description of boundary conditions, which are inherently of eulerian nature. The conservative form of the Euler equations is then used and the particle approximation of this system of equations leads to the resolution of one dimensional Riemann problems between neighbouring points. The practical numerical discretization is achieved with Riemann solvers and higher order schemes. Finally, boundary field values are obtained by solving partial Riemann problems and boundary fluxes are directly computed and integrated on boundary surfaces (see also [6]).

II. ALE conservative formulation

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

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

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

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

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

Moving Boundary Problems in the Finite Volume Particle Method

R. Nestor, M. Basa, N. Quinlan
Department of Mechanical and Biomedical Engineering
National University of Ireland, Galway
Galway, Ireland
malachy.nestor@nuigalway.ie

***Abstract*— The Finite Volume Particle Method (FVPM) is a mesh-free method for fluid dynamics. The method is formulated from the integral form of the conservation equations, and particle interactions are described in terms of inter-particle fluxes. Boundaries can be implemented without the need for fictitious particles, and the method is conservative. The motion of the particles can be independent of the fluid velocity.**

The advantages of Lagrangian particle motion are numerous, allowing free-surface and moving boundary problems to be simulated with relative ease. However, Lagrangian particle motion in FVPM has been problematic due to the development of highly non-uniform particle distributions, which affect the accuracy and robustness of the method.

In this article, a particle velocity correction is proposed which acts to maintain the uniformity of the moving particle cloud, regardless of the flow configuration. This is facilitated by FVPM due to the independence of particle and fluid velocities, in a similar manner to Arbitrary Lagrangian-Eulerian methods for mesh-based discretisations. The scheme is assessed for incompressible lid-cavity flow at Reynolds number 1000 (SPHERIC benchmark 3). In addition, the suitability of the method for moving boundary problems is demonstrated for incompressible flow over a moving square cylinder in a fixed rectangular walled enclosure (SPHERIC benchmark 6).

I. Introduction

The Finite Volume Particle Method (FVPM) was introduced by Hietel et al. [5], and subsequently developed by Keck [6], [7], Teleaga [15], and others. As in SPH, the fluid in FVPM is represented by particles. However, FVPM differs from SPH in that interactions between particles are defined in terms of fluxes. Particle-boundary interactions can also be described using fluxes, so boundary conditions can be enforced without the use of fictitious particles. The method is conservative in a similar manner to traditional finite volume methods. An interesting feature of the method is that the particle motion may be chosen arbitrarily, and may be independent of the fluid velocity.

Moving boundary problems in fluid dynamics represent a significant challenge for Eulerian mesh-based CFD methods. Lagrangian motion of the computational nodes is a feature of mesh-free particle methods that offers the ability to handle moving boundary problems with relative ease. However, the use of Lagrangian particle motion means that the particle distribution is determined entirely by the velocity field. Highly distorted particle distributions can result in regions of high flow strain, which invariably leads to a degradation of the robustness and accuracy of any numerical discretisation. This has been demonstrated recently in SPH by Colagrossi *et al.* [3] in the context of an incompressible moving boundary problem. In this work, we argue that for incompressible flow in FVPM, this issue can be worse than the corresponding SPH case, and that some control needs to be exercised over the motion of the particles. We propose a particle motion correction, based on a previous formulation for one-dimensional problems [13], which maintains the uniformity of the particle distribution. The resulting FVPM scheme is applied to incompressible flow problems with and without moving boundaries.

II. The Finite Volume Particle Method

In FVPM, the fluid is represented by a set of N particles. A particle is defined by a normalised, compactly-supported test function ψ_i

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

where $W_i = W\left(\mathbf{x}-\mathbf{x}_i\left(t\right),h\right)$ is a compactly supported kernel function for particle i, centred at $\mathbf{x}_i$, with a compact support radius $2h$. Each particle is associated with a volume V_i, and a discrete value of any field variable ϕ_i:

$$V_i = \int_\Omega \psi_i \,\mathrm{d}\mathbf{x} \tag{2}$$

$$\phi_i = \frac{1}{V_i}\int_\Omega \phi\psi_i \,\mathrm{d}\mathbf{x}, \tag{3}$$

where denotes the region coinciding with the particle support. The FVPM is derived from the integral form of the governing equations:

$$\frac{\partial}{\partial t}\int_\Omega \mathbf{U}\,\mathrm{d}\mathbf{x}+\int_\Omega \nabla\cdot\mathbf{F}\,\mathrm{d}\mathbf{x}=0. \tag{4}$$

$\mathbf{U}$ is the vector of conserved quantities

$$\mathbf{U}=\begin{pmatrix}\rho\\ \rho\mathbf{u}\end{pmatrix}, \tag{5}$$

where ρ is the fluid density, and $\mathbf{u}$ is the fluid velocity vector. $\mathbf{F}$ is the flux vector

$$\mathbf{F}=\begin{pmatrix}\rho\mathbf{u}^T\\ \rho\mathbf{u}\otimes\mathbf{u}+p\mathbf{I}-\tau\end{pmatrix}, \tag{6}$$

High-Performance Computing SPH: Towards A Hundred Million Particle Simulation

C. Moulinec[a], R. Issa[b], D.R. Emerson[a] and X.J. Gu[a]
[a] STFC Daresbury Laboratory, WA4 4AD, UK
[b] EDF R&D, National Hydraulics and Environment Laboratory,
& Saint-Venant Laboratory for Hydraulics, 6, quai Watier, 78400 Chatou, F

***Abstract*— Few 3-D SPH simulations have been carried out in Computational Fluid Dynamics (CFD) to our knowledge, mainly due to prohibitive computational investment since the number of particles required in 3-D is usually too large to be handled by a single processor. In this paper, the new parallel version of the SPH code, Spartacus-3D, is presented. The modifications to the code, based on a localisation of previously global arrays, combined with a switch from global to local communications where possible, leads to a better efficiency of the code and an increase in the number of particles that can be handled in a simulation. We have reached 103M particles on 2,048 cores of the UK's high-end computer (Cray XT4, HECToR). The parallelisation strategy is discussed as well as results regarding the speed up of the code. Two simulations of 3-D channel flow are carried out, the former in the laminar regime and the latter in the turbulent regime.**

I. Introduction

The Smoothed Particle Hydrodynamics (SPH) numerical method has been successfully applied in fluid mechanics to simulate strongly distorted free surface flows in situations where classical Eulerian approaches would fail because of grid steepness. However, 3-D SPH simulations remain costly, because a large number of particles are necessary in order to simulate such flows and because of the nature of classical SPH which requires a homogeneous initial particle distribution for incompressible flows. In order to handle these problems while reducing computational time, a parallel 3-D SPH code, namely Spartacus-3D, was developed in 2006, based on the MPI library [7]. The fact that most arrays were global in that version implied that a limited total number of particles could be handled, the memory required by processors being over-estimated. Moreover, global communications were compulsary. As a result, performance was good up to a few tens of cores, depending on the configuration case [7], but that version of the code was not suitable for massively parallel computations. The latest version of the code, where the accent has been put on array localisation, is presented here.
After describing the methodology, some SPH principles and the numerics used in Spartacus-3D, the parallelisation technique is explored. The performance of the code is shown on the UK's new high-end computing resource (Cray XT4) [9]. The code is then applied to the test case of a 3-D channel flow at two different Reynolds numbers to simulate a laminar ($Re = 100$ based on the bulk velocity) and a turbulent ($Re_\tau = 180$ based on the friction velocity) flow.

II. Methodology

The pseudo-compressible Navier-Stokes equations written in Lagrangian form for Newtonian fluids and incompressible and laminar flows read:

$$\frac{d\mathbf{u}}{dt} = -\frac{1}{\rho}\nabla p + \nabla\cdot(\nu\nabla\mathbf{u}) + \mathbf{F^e}, \qquad \frac{d\rho}{dt} = \rho\nabla\cdot\mathbf{u} \tag{1}$$

where $\mathbf{u}$ is the velocity vector, t the time, ρ the density, p the pressure, ν the kinematic viscosity, and $\mathbf{F^e}$ represents an external force. In addition, '∇' and '$\nabla\cdot$' are the gradient and divergence operators, respectively.
This system of 4 equations in 3-D (see Eq. 1) is not closed, because $\mathbf{u}$, p and ρ are all unknowns. To close the system, the pressure is expressed as a function of the density through the state equation [3], as:

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

where ρ_0 is the reference density, c_0 a numerical speed of sound, and γ a constant coefficient. In order to simulate an incompressible flow, c_0 must be at least ten times larger than the maximal velocity of the flow. The nearly-incompressible assumption used here implies that the Mach number M has to be less than 0.1. Consequently, the relative variation of density, which scales as M^2, is less than 1%. Through Eq. 2, it follows that the pressure automatically goes to zero when the density equals the reference density, which is a necessary condition for free-surface flows.
A particle's position is updated at each temporal iteration by the following integration:

$$\frac{d\mathbf{r}}{dt} = \mathbf{u}. \tag{3}$$

Equations 1, 2 and 3 are discretised explicitly in time (first order) and the SPH approach is used to perform the spatial discretisation.

HPC For Spartacus-3D SPH code and applications to real environmental flows

Réza Issa, Damien Violeau and Guillaume Thibault
LNHE & SINETICS departments
Electricité de France R&D
Chatou, France
reza.issa@edf.fr

David Latino
IBM Global Engineering Solutions
IBM Systems & Technology Group
Dubai, United Arab Emirates

Charles Moulinec
Computational Engineering Group
Daresbury Laboratory
Warrington, United Kingdom

John Biddiscombe
Swiss National Supercomputing Centre
Manno, Switzerland

***Abstract*— This paper provides computing performance of the massively parallel SPH code Spartacus-3D in terms of speed-up on a Blue-Gene L machine as well as first validations of the code. An example of 3-D lagrangian visualization with pv-meshless is also given, opening to the near future real applications of Spartacus-3D in coastal and river engineering. A first attempt to simulate an industrial environmental case, a breaking wave impacting an off-shore wind turbine, will probably be introduced during the workshop.**

I. INTRODUCTION

At EDF R&D, SPH aims at improving the design of coastal protections of power plants or optimizing the geometry of environmental river structures such as dam spillways, fish passes, etc. For this purpose, a 3-D SPH code, Spartacus-3D, has been developed since 1999. However, real applications of 3-D SPH is still an issue, mainly because of heavy CPU time requirements. Consequently, an SPH code needs to be massively parallelized in order to be used for most industrial applications. Another issue is post-processing: on one hand, the post-processor has to efficiently handle large amount of data and, on the other hand, has to be powerful enough for visualizing complex 3-D Lagrangian simulations in a clear manner. This question is now partially answered by pv-meshless, developed by CSCS from the Paraview open-source code.

In this paper, the parallelization technique implemented in the last version of Spartacus-3D is briefly presented. It consists in an improved algorithm than the one described in [5]. First values of speed-up of Spartacus-3D on a Blue-Gene L machine will be shown as well as validations of the code and first pv-meshless outputs. The question of pre-processing for complex industrial flows will also be raised.

II. PARALLELISM

A. Description of the needs

Applications of 3-D SPH could be numerous in the context of environmental flows (see figure 1). As far as EDF (Electricité de France) is concerned, one should mention:

- Breaking waves and submersion problems in general;
- Flows over dam spillways, fish passages and other river waterworks;
- Design of sea defence for the protection of power plants against flooding;
- Design of offshore facilities such as offshore wind turbines, marine current turbines, etc.
- Release and intake waterworks of coastal power plants, and transport of material by the currents and waves in their vicinity;

Modelling such flows with Eulerian methods is rather difficult, in some cases impossible, due to the complex adaptative mesh required. On the contrary, the development of SPH for free-surface flows [1] makes it easier, due to the capability of this method to model highly distorted free-surfaces together with multiphase flow and fluid-structure interactions. However, simulating industrial flows such as those presented on figure 1 would require several million particles in order to represent most complex features of the flows. For instance, the case of a flow over the spillway of figure 1 would need between 5 and 50 million particles. Moreover, when using weakly compressible SPH, small time steps are necessary. Computing times on a single processor will hence make real applications impossible. Considering the spillway of figure 1, a simulation corresponding to one minute of physical time would require 25 years of computing

High performance computing 3D SPH model: Sphere impacting the free-surface of water

Pierre Maruzewski
Ecole Polytechnique fédérale de Lausanne
EPFL – STI - LMH
Lausanne, Switzerland
pierre.maruzewski@epfl.ch

Guillaume Oger
HydrOcean
Nantes, France
guillaume.oger@hydrocean.fr

David Le Touzé
Ecole Centrale de Nantes
ECN
Nantes, France
david.letouze@ec-nantes.fr

John Biddisccombe
Swiss National Supercomputing Centre
CSCS
Manno, Switzerland
john.biddiscombe@cscs.ch

***Abstract*—In this work, an analysis based on a three-dimensional parallelized SPH model developed by ECN and applied to free surface impact simulations is presented. The aim of this work is to show that SPH simulations can be performed on huge computer as EPFL IBM Blue Gene/L with 8'192 cores. This paper presents improvements concerning namely the memory consumption, which remains quite subtle because of the variable-H scheme constraints. These improvements have made possible the simulation of test cases involving hundreds of millions particles computed by using more than thousand cores. This is illustrated in the paper on a water entry problem, namely on a test case involving a sphere impacting the free surface at high velocity. In the first part, a scalability study using from 124'517 particles to 124'105'571 particles is realized. In the second part, a complete simulation of a sphere impacting the free surface of water is done with 1'235'279 particles. A convergence study is achieved on pressure signals recorded by probes located on the sphere surface. Furthermore, ParaView-meshless developed by CSCS, is used to show the pressure field and the effect of impact.**

I. INTRODUCTION

Smoothed Particle Hydrodynamic method is well suited to model complex free surface problems of fast dynamics such as the water entry of 3D objects of complex geometry. With respect to classical Level-Set or VOF interface tracking, the free surface remains always precisely described by the particles in their Lagrangian motion with no need for adapting a mesh. Presently, most of the models present in the free surface SPH related literature are two-dimensional, and thus do not really suffer from high computational cost difficulties. But actual engineering applications are indeed three-dimensional, which dramatically increases the computational cost, and finally limits the use of the SPH method. Millions of particles and an efficient parallelization are then required. In this context, being an explicit method, SPH presents the other asset of a rather straightforward parallelization with respect to classical Finite Difference Method, Finite Volume Method or Finite Element Method used in Computational Fluid Dynamics. Ecole Centrale de Nantes decided to develop a parallelized model. This latter is tested on the Cray XD1 cluster with 32 cores [1]. Furthermore, Sbalzarini has developed a highly efficient parallel particle-mesh library which provides a computational tool for mesh-based method [2]. Finally, Ecole Polytechnique Fédérale de Lausanne has shown that SPH parallel simulation is available on High Performance Computing machine like an IBM Blue Gene/L with 8'192 cores [3].

II. SPH SOLVER

A. *Method*

The SoPHy-N model [1], developed by Ecole Centrale de Nantes, relies on renormalization kernel method and implementation of an exact Riemann solver with Godunov numerical scheme as described by Vila et al [4]. The latter presents various advantages such as avoiding artificial viscosity required in standard SPH, decreasing numerical dissipations and increasing stability, in a way inspired from compressible finite-difference and finite-volume schemes. Boundary conditions are imposed using ghost particles.

SPH simulation of non-newtonian mud flows

Tatiana Capone
Dept. of Hydraulic, Transportation and Roads, DITS
"Sapienza" University
Rome
tatiana.capone@uniroma1.it

Andrea Panizzo
Dept. of Hydraulic, Transportation and Roads, DITS
"Sapienza" University
Rome
andrea.panizzo@uniroma1.it

***Abstract*— Non newtonian mud flows are simulated including the Bingham model in SPH. A smooth switch is used to connect the mechanic fluid behaviour of the yielded (where deformation occurs) and un–yielded (no deformation) regions.**

I. Introduction

In the present work the capability of SPH to model non-newtonian mud flows is analyzed and tested. These kind of fluids consist in solid grains immersed into a water matrix; it is possible to say that from a certain concentration of solid grains into the mixture they behave like non Newtonian fluids [5]. For these fluids the Bingham model seems to be one of the best to represent the viscoplastic behaviour [6] in which no deformation takes place until a certain stress is applied, as extensively reported in literature [1]. In the present work a particular kind of Bingham viscosity model is used. In order to avoid the discontinuity between the yielded and the unyielded regions, a smooth switch is introduced, giving a higher viscosity for low shear stresses values. In literature these kinds of constitutive equations are known as biviscosity models [2]. A two dimensional annular viscometer test is used in order to carry on the steady flow analysis and to set the parameters necessary to give a correct evaluation of the viscosity field, following the analytical solutions calculated for this particular model implemented and comparing it with other non-newtonian and newtonian models. Then an unsteady flow analysis is presented consisting in a dam break flow of a mixture of kaoline and water, with different percentage of solid grains following the experimental results of Komatina D. and Jovanovic M. [5].

II. Governing Equations

In order to model a Bingham fluid flow we have to refer to the constitutive equation of a generalized newtonian fluid, it differs from a newtonian one in the way that the shear stress τ depends upon the shear rate tensor (i.e. the gradient of the velocity field) at a particular time:

$$\mathbf{D} = \frac{1}{2}\left(\nabla\mathbf{u} + \nabla^{\mathbf{t}}\mathbf{u}\right) \tag{1}$$

where $\mathbf{D}$ has the form shown below:

$$\mathbf{D} = \begin{bmatrix} \frac{\partial u}{\partial x} & \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \\ \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) & \frac{\partial v}{\partial y} \end{bmatrix} \tag{2}$$

Usually this tensor is used in the formulation of the viscosity field in the form of its second invariant:

$$|\mathbf{D}| = \sqrt{\mathbf{D} : \mathbf{D}} = \sqrt{\sum_{ij} D_{ij} D_{ij}} \tag{3}$$

The constitutive law for the generalized newtonian fluids is thus of the following form:

$$\tau = \mu\left(|\mathbf{D}|\right)\mathbf{D} \tag{4}$$

in which the relation between viscosity and the second invariant of the shear rate tensor is specified .
In the presence of classical newtonian fluids this relation becomes:

$$\tau = 2\mu\mathbf{D} \tag{5}$$

The Bingham plastic behavior is represented by the presence of a critical yield stress under which no deformation takes place. Above this level viscosity depends upon the second invariant of the shear rate tensor, as usual for the generalized newtonian fluids, and the yield stress. The constitutive equations are the following:

$$|\tau| \leq \tau_y \Leftrightarrow \mathbf{D} = 0 \tag{6}$$

$$|\tau| > \tau_y \Leftrightarrow \tau = \left(\frac{\tau_y}{|\mathbf{D}|} + 2\mu\right)\mathbf{D} \tag{7}$$

In this work we propose a slightly different Bingham model that is present in literature with the name of "bi-viscosity model" [2]. The main difference between this kind of model and the classical Bingham one is that in the un-yielded region a greater viscosity is given to the fluid, instead of having no deformation at all.

$$|\mathbf{D}| \leq \frac{\tau_y}{2\alpha\mu} \Leftrightarrow \tau = 2\alpha\mu\mathbf{D} \tag{8}$$

$$|\mathbf{D}| > \frac{\tau_y}{2\alpha\mu} \Leftrightarrow \tau = \left(\frac{\tau_y}{|\mathbf{D}|} + 2\mu\right)\mathbf{D} \tag{9}$$

In (8) and (9) are reported the expression of the stress tensor in the un-yielded and in the yielded regions, where μ is the classical dynamic viscosity and α is a coefficient that gives

SPH Molecules - a model of granular materials

Tatiana Capone
DITS, Univeristy of Roma (la Sapienza)
Roma, Italy

Jules Kajtar
School of Mathematical Sciences
Monash University
Vic. 3800, Australia

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

***Abstract*— An SPH model of granular material is described. The model replaces standard SPH particles by small molecules each of which moves as a rigid body. We simulate dry granular materials forming piles and show that they have an angle of repose similar to that in nature. We simulate an avalanche by placing the molecules on a steep hillside and show that the avalanche runs onto a horizontal plane and eventually comes to rest. Finally, we simulate dambreaks using a mixture of granular and liquid SPH particles. The effective viscosity of the mixture increases as the concentration of the granular materials increases. The increase is small for a concentration less than 0.5 but increases very rapidly for concentrations above this. These results are similar to those for a Bingham rheology.**

I. Introduction

This paper is concerned with the simulation of problems involving granular material or powders combined with fluids. They include geophysical phenomena such as mud flows and submarine landslides, and industrial phenomena such as the flow of chocolate or bitumen. The combination of a granular material and a fluid results in a material with a non-linear rheology. The rheology may be time dependent when the granular material breaks up, or when it becomes heated, or when the random motion of the grains alters.

In some cases the rheology can be approximated by that of Bingham, Close, and Herschel-Bulkley. The simulation problem then reduces to integrating the Navier-Stokes equations with a non linear viscous term. In the case of the Bingham and Herschel-Bulkley rheologies this has been carried out by Vola et al. (2004) using a Characteristic Galerkin method with a finite element decomposition, and in the case of SPH by Shalo and Lo (2003) and Hosseini et al. (2006).

The situation with many geophysical materials is much less satisfactory because the rheology is not known. In this paper we consider a more general approach where the effects of the grains or powder is modelled by a much smaller number of SPH particles which behave like rigid bodies which we call molecules. The molecules consist of a small number of particles, typically 5. The interaction between each of the molecules, and between the molecules and the fluid must be modelled and we do this by using simple forces. This aspect of our method is similar to molecular dynamics or the discrete element method (DEM). The advantage of our method is that we can explore very complicated systems involving many different types of grains and fluids and, because we use SPH we can, in principle, find the equivalent continuum equations by taking the continuum limit of the SPH equations.

II. The types of Molecules

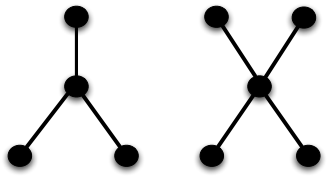


Fig. 1. Two examples of the molecules we use. Each filled circle marks the site of a particle of the molecule.

In Fig.1 we show two examples of the molecules we used for our two dimensional simulations. Each filled black circle in figure 1 indicates a particle of the molecule, and is the site of a force which acts on particles whether from another molecule, a liquid SPH particle or a boundary force particle. The black lines in figure 1 (called legs in the following) emphasize the geometry. A molecule moves as a rigid body according to the total force and torque exerted on the particles of the molecule. The force/mass $\mathbf{f}_{jk}$ exerted by a particle k of a molecule on a particle j of another molecule, boundary particle or fluid particle, is radial and has the form

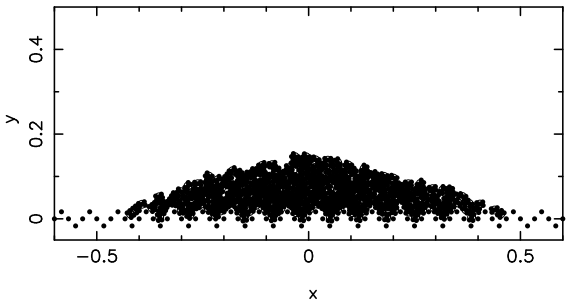


Fig. 2. A pile of granular material modelled by rectangles. Note the corrugated base and the fact that some molecules have fallen into the corrugations.

A SPH thermal model for the cooling of a lava lake

A. Herault
Université Paris-Est
S3IS
Paris, France
herault@ct.ingv.it

A. Vicari, C. Del Negro
INGV
Sezione di Catania
Catania, Italy
vicari@ct.ingv.it

***Abstract*— During the solidification of lava, heat is released by radiation from the free surface as well as by conduction into the ground from the base, leading to non-uniform solidification. The upper solidified layer grows at a faster rate than the lower solidified layer. There is no available analytical solution for this kind of problem. For this reason a SPH thermal model for the cooling of lava was implemented. The model was used for the simulation of the cooling of a lava lake and a lava flow, for which we propose an evolution law for the crust thickness and the surface temperature. We have explored the possibility of implementing the SPH model on a graphics processing units.**

I. INTRODUCTION

The numerical simulation of lava cooling raises the need to address the phase change problem in arbitrary geometry with many fronts of solidification and nonlinear Neuman condition (radiative losses). The simulation of such systems can sometimes present difficulties for finite element methods (FEM): the simplest way to deal with the phase change phase with FEM is to introduce an effective specific heat that shows a peak around the temperature of phase change; when the different solidification fronts connect themselves, as in the case of lava, this method is unable to provide the position of the front and a proper temperature profile around the front. Following the work of Monaghan et al. [1], a SPH model was implemented for the resolution of this type of problems. The SPH method presents the important advantage to take into account implicitly the evolution of solidification fronts and therefore they not require any specific treatment, whatever their shape and their number. We applied our model on two problems: the cooling of a lava lake and the cooling of a lava flow. The results obtained were compared to those obtained with finite elements and, in the case of the lava lake, to a semi analytical solution given by Manglick [2], demonstrating the superiority both in terms of resolution and ease of implementation of the SPH method. The next step will be to introduce the motion equation coupled to the thermal model. However, in this case the capability of calculation becomes a determinant factor. For this reason, we have explored the possibility of implementing the SPH model on a graphics processing units (GPUs), which is a cheap non-specialized GRAPE [6]. SPH applications are well suited to GPUs, due to their extensive computational requirements, and because they lend themselves to parallel processing implementations. Modern GPUs contain hundreds of arithmetic units and can be harnessed to provide tremendous acceleration for many numerically intensive scientific applications. The increased flexibility of the most recent generation of GPU hardware combined with high level GPU programming languages have unlocked this computational power and made it much more accessible to computational scientists. NVidia released on 2007 the first software development kit (SDK) for genéeral purpose graphics processing units in order to facilitate the scientific calculation on GPU. We implemented our SPH model on a NVidia GeoForce 8800. The results obtained on a 3D test case will be shown in terms of speed up performance.

II. SPH COOLING MODEL OF A LAVA FLOW

A. Notations

Throughout this paper we will use the following notations:

- T absolute temperature, T_g ground temperature, T_{ext} external temperature, T_s phase change temperature;

- m mass, ρ density, c_p heat capacity per unit mass at constant pressure, κ coefficient of thermal conductivity, L latent heat of solidification, T_s temperature of solidification, k_b the Stefan-Boltzmann constant and ε emissivity;

- the properties of the liquid phase will be noted with the subscript '*l*' , while those of the solid phase with the subscript '*s*';

- the vectors are noted in **bold**;

- the particles are identified by Greek subscripts;

- $\mathbf{r}$ distance from origin, $\mathbf{r}_{\alpha\beta} = \mathbf{r}_\alpha - \mathbf{r}_\beta$

- h smoothing length, $W(\mathbf{r},h)$ smoothing kernel and F scalar function defined by $\mathbf{r}_{\alpha\beta} F(\mathbf{r}_{\alpha\beta}, h) = \nabla_\alpha W(\mathbf{r}_{\alpha\beta}, h)$;

Modelling 3D fracture and fragmentation in a thin plate under high velocity projectile impact using SPH

Raj Das* and Paul W. Cleary
CSIRO Mathematical and Information Sciences
Private bag 33
Melbourne, Australia
E.mail: raj.das@csiro.au

***Abstract*— Damage by fracture under high-speed impact is a dominant mode of failure in many industries. Numerical modelling can play a crucial role in understanding and predicting complex fracture processes. The commonly used mesh-based Finite Element Method has difficulties in accurately modelling the high deformation and disintegration associated with fracture, as this often distorts the mesh. Here we use Smoothed Particle Hydrodynamics (SPH) to study the brittle fracture of a 3D thin plate during impact by a high-speed projectile. The transient stress field and the resulting damage evolution are investigated. SPH predictions of the fracture patterns and the distribution of fragments are found to be physically sensible.**

I. INTRODUCTION

Fracture and fragmentation are governed by multiple physical processes involving different time and length scales. The sources of fracture are micro and macro voids, micro-structural defects, and initial flaws. Numerical fracture models can potentially predict both small and large scale fracturing events, ranging from breakage in mills [1] to the formation of impact craters [2]. Modelling breakage in granular flows is useful in optimising the design and setting operating parameters in mills to enhance mill efficiency and achieve desired product quality [1].

In the past, approaches for studying fracture have been heavily dependent on empirical relations based on laboratory scale experimental data [3]. For large scale impact problems, the extrapolation of this data using a scaling law is not always reliable [4]. As a result, the empirical models do not often perform well outside the regimes of the original experiments. Numerical fracture models, based on well-reasoned physical models, can potentially provide effective solutions for predicting large scale fracturing events, such as explosions in mines [5], the formation of impact craters [2]. Furthermore, the assessment of damage initiation and progression can be rapidly and easily performed for a wide range of problem parameters, such as geometry, loading, initial defect characteristics and distribution, which will extend beyond the scope of experiments. Effective damage modelling techniques in conjunction with efficient numerical methods have strong potential for increasing our understanding of fundamentals of breakage and for providing practical tools that can be used for equipment and structure design.

Here we use Smoothed Particle Hydrodynamics (SPH) to model brittle failure of structures under impact load. SPH was first proposed by Gingold and Monaghan [6] and Lucy [7] for astrophysical problems. It has been traditionally used for modelling fluid flows [8, 9]. In recent years, there has been a growing interest in applying SPH to a wide variety of solid mechanics problems [10-12]. SPH is a mesh-free numerical method, which is used to obtain solutions to systems of partial differential equations, by discretising materials into 'particles'.

Traditional Eulerian methods have difficulties in tracking fracture on a node by node basis and in predicting the evolution of the damage. The history tracking ability of SPH provides a natural framework enabling the prediction of damage initiation, evolution and crack propagation. Wingate *et al.* [13] applied SPH for modelling crater produced by hypervelocity impact and compared the results against other numerical methods. Randles *et al.* [14] proposed a stress based fracture model in SPH and applied it to simulate oblique impact of tungsten cubes on multi-layered targets. Johnson *et al.* [15] used 2D SPH models to study the penetration of a long rod into a thick target. Libersky *et al.* [16] used the SPH code MAGI to simulate fracture of an aluminium target under the impact of steel cubes. Medina and Chen implemented a visco-plastic material model into their SPH code and studied high velocity impact of steel projectiles on composite plates [17]. Recently, Das and Cleary used SPH to analyse the effect of different rock shapes on fracture patterns [18], and to study the impact fracture of 2D slender thick columns [19]. Here we use SPH for three-dimensional modelling of impact fracture of a thin shell structure.

SPH Interaction of Fluids and Solids

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

***Abstract*— The study is concerned about the application of the smoothed particle hydrodynamics (SPH) method within the computational fluid dynamics and the elastodynamics. A brief description of the SPH model for an incompressible fluid and for an isotropic elastic solid is presented. The implemented model of incompressible fluid is tested for a laminar flow in a duct and the implemented elastic material model is examined during the simulation of the mechanical response of rubber rings. The main scope of the work is focused on the SPH implementation for problems featuring fluid-structure interaction. The coupling between the fluid and the solid model is described and its feasibility to the fluid-structure interaction modelling is addressed. Results of the fluid-structure interaction simulation are compared to data published in the literature and the performance of the implemented SPH code is discussed.**

I. Introduction

Since the invention of the smoothed particle hydrodynamics (SPH) in late 1970s, [4] and [8], the SPH method went through a substantial development. The flexibility of its meshless Lagrangian nature and ease of implementation may be well employed within numerous branches of computational physics. Within this contribution, an attention is paid to the SPH application to the incompressible fluid dynamics and to the elastodynamics. Well known SPH models for a laminar flow of incompressible fluid, e.g. [9], [3] or [7], and for an isotropic elastic material, e.g. [6] or [5], are adopted and their brief description is presented. A special attention is paid to the SPH ability to couple the above described models and solve the fluid-structure interaction problem. A number of various studies on fluid to structure or structure to structure interaction were published so far, e.g. [2], [14] or [1].

There are two major SPH approaches to the material interaction representation applied in practice. The first approach might be called "the summation over all particles" where the governing equations are solved for all particles together, while the second approach treats the particles of every material separately and let them interact through a special interaction algorithm (that might be represented by an application of contact forces at the material interface). The latter approach is applied within this study. An implementation of fast and flexible contact algorithm for solution of fluid-structure interaction problems (based on a study of Campbell et al. [2]) is presented.

In addition, an implementation of rigid boundary conditions and commonly used standard SPH stabilising terms enforcing the stability of the numerical simulations are described. The implemented SPH code is applied to test problems taken from the literature.

II. SPH model

In the following, the standard SPH method is applied, i.e. a continuum is discretised by a finite set of interpolating points (particles) with invariant coordinates in the material frame and representing a finite mass of the continuum. The function values and their derivatives at a specific particle are interpolated from the function values at surrounding particles using the interpolating (smoothing) function and its derivatives, respectively. More information on deriving the general SPH equations and smoothing function properties can be found in e.g. [9] or [7]. Within this study, the generally used cubic B-spline smoothing function is applied, [9].

A. Incompressible fluid

The symetric form of the SPH governing equations describing the dynamics of the incompressible fluid is considered. The conservation of mass is assured by the continuity equation

$$\frac{\mathrm{d}\rho_i}{\mathrm{d}t} = \sum_j m_j(\mathbf{v}_i - \mathbf{v}_j)\cdot\nabla_i W_{ij}, \tag{1}$$

where m is the mass, ρ is the density, $\mathbf{v}$ is the velocity vector and $W_{ij} = W(|\mathbf{r}_i - \mathbf{r}_j|, h)$ is the smoothing function with a continuous derivative $\nabla_i W_{ij}$. The index i, j respectively, denotes the variables at the particle i, j respectively, and ∇_i denotes a derivative according to $\mathbf{r}_i$ which is the position vector.

The conservation of momentum is described by the Navier-Stokes equation in the form

$$\frac{\mathrm{d}\mathbf{v}_i}{\mathrm{d}t} = \mathcal{P}_i + \mathcal{V}_i + \mathbf{f}_i, \tag{2}$$

where $\mathcal{P}_i$ is the pressure term, $\mathcal{V}_i$ represents the viscous forces and $\mathbf{f}_i$ is the body force. The pressure term is derived so that

$$\mathcal{P}_i = -\sum_j m_j\left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2}\right)\nabla_i W_{ij}, \tag{3}$$

where p is the pressure. The viscous forces are modelled by a term derived by Morris et al. [13]

$$\mathcal{V}_i = \sum_j 2\mu\frac{m_j}{\rho_i\rho_j}\frac{(\mathbf{r}_i - \mathbf{r}_j)\cdot\nabla_i W_{ij}}{|\mathbf{r}_i - \mathbf{r}_j|^2 + \eta^2}(\mathbf{v}_i - \mathbf{v}_j), \tag{4}$$

SPH framework to model fluid shell interactions

Serguei Potapov
LaMSID - UMR EDF/CNRS 2832
EDF-R&D
Clamart, France
e-mail: serguei.potapov@edf.fr

Bertrand Maurel, Alain Combescure
LaMCoS - UMR CNRS 5259
INSA-Lyon
Villeurbanne, France
e-mail: alain.combescure@insa-lyon.fr

***Abstract*—This paper presents a unified numerical framework based on SPH formulation and devoted to the modelling of fluid structure interaction problems involving large motion of the fluid and large deformation with a possible failure of the structure. For the fluid domain we use the balance equations written in a current configuration according to an updated Lagrangian formulation. For the solid domain (3D and shell models) we base the SPH framework on the total Lagrangian formulation. That allows avoiding the so-called tension instability inherent to the updated Lagrangian formulation, and thus very large solid deformations become easy to handle. Because both fluid and structure are modelled with SPH, the interaction between the fluid and the structure is treated via a unilateral contact algorithm. To detect contacts we use the Pinball concept designed for solid contact mechanics. The presented SPH framework is implemented in the EUROPLEXUS fast dynamic software.**

I. INTRODUCTION

To study the dynamic response of complex mechanical systems containing fluid and structural parts, it is of common practice to use distinct fluid and structure solvers that communicate through a well-defined interface between the two media. Such an approach is convenient to study the so-called permanent Fluid-Structure Interaction (FSI) problems where the contact between the structure and the fluid takes place during the whole transient of interest even if fluid and/or structure undergo large internal deformations [1]. However, such a classical FSI approach fails when the topology of 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. The mesh-based methods are no more suitable. To deal with medium discontinuity, several Lagrangian-type meshless (or particle-based) methods have been introduced (see [2] for a complete review) and applied to model fluid flows, solid deformations and high velocity impact and fracture. Among several methods, we selected the SPH technique because of its ability to treat both fluid and solid dynamic problems. Here we present an entirely meshless approach to study accidental-type transient FSI problems by modelling both fluid and structure with the SPH method. The whole algorithm is implemented in the EUROPLEXUS fast dynamics software [3].

II. SPH FOR FLUID AND SOLID MEDIA

A. Fluid modelling with SPH

The evolution of the fluid continuum is described, in isothermal conditions, by the continuity and momentum conservation equations, which, in SPH approximation, take the following discrete form:

$$\frac{d\rho_i}{dt} = \rho_i \sum_j m_j \frac{1}{\rho_j} (\underline{v}_i - \underline{v}_j).\underline{\nabla} W_{ij} \tag{1}$$

$$\frac{d\underline{v}_i}{dt} = -\sum_j m_j \left(\frac{\underline{\underline{\sigma}}_i}{\rho_i^2} + \frac{\underline{\underline{\sigma}}_j}{\rho_j^2} + \Pi_{ij} \underline{\underline{I}} \right).\underline{\nabla} W_{ij} \tag{2}$$

For the fluid domain, we write the balance equations in a current configuration according to an Updated Lagrangian (UL) formulation. Velocity is chosen as a primary variable and the constitutive equations relate Cauchy stresses to strain rates. In (1) and (2) W_{ij} is a chosen kernel (cubic in our case) and Π_{ij} is an artificial viscosity term suggested in [4].

B. Solid modelling with SPH

For the solid continuum, the SPH formulation is similar to that of the fluid but the stress-strain equations relate now the stresses to the strains. The principal variable for the solid SPH is the displacement.

There is a significant difference between the fluid and the solid formulations: the fluid cannot withstand any tensile hydrostatic pressure and hence all particles experiencing positive volume change are no longer connected to their neighbours. This is not the case for solid particles. The first computations with solid formulation using updated Lagrangian formulation have shown numerical instabilities in tension as well as in shear even if the strains remain small (about 1%). It has been shown in [5] that this is due to numerical instabilities of the UL formulation and that the

Simulating Dynamic Surface Tension of Lung Surfactant Using SPH

S. Adami[a], I. Mahle[a], X.Y. Hu[a], N.A. Adams[a], M. Wendel[b],
[a]Lehrstuhl für Aerodynamik, TU München, Germany
[b] Medizinische Fakultät Carl Gustav Carus, Institut für Physiologie, TU Dresden, Germany
stefan.adami@aer.mw.tum.de

***Abstract*—A two-dimensional smoothed particle hydrodynamics method to simulate the dynamic surface tension of oscillating air bubbles with surfactants at the phase interface is presented. The modelled interfacial transport is based on a well-known one-dimensional description of the surfactant transport phenomenon to air-liquid interfaces. A main advantage of this Lagrangian formulation is the adaptive interface respresentation. A parameter variation shows, that this model can predict the adsorption of surfactant to an interface for constant-area bubbles as well as for oscillating interfaces. Even the "pseudo-collapse" of surfactant films at high adsorption rates was observed.**

I. Introduction

The presence of surfactant in the liquid-lining layer of pulmonary alveolar structures has a major impact on the surface tension at the air-liquid interface. With increasing surfactant concentration at the interface, surface tension decreases and consequently the breathing process is influenced, i.e., the work required for respiration is reduced [1]. Furthermore, many engineering and scientific applications use the effect of surfactants on interfaces, e.g., to manipulate the deformation of drops and bubbles in shear flows [2], [3].

Experimental investigations on the behaviour of lung surfactant in vitro frequently use a bubble surfactometer [4], [5]. With this instrument, a nearly spherical bubble of air is formed at the end of a narrow tube and exposed to a liquid suspension containing surfactant. Oscillating the bubble volume with time, the dynamic behaviour of the surface tension due to the surfactant at the air-liquid interface can be measured. The surface tension is not measured directly but rather the pressure drop across the bubble surface (pulsating bubble surfactometer [6]) or the changing shape of the bubble (captive bubble surfactometer [7]) is estimated.

To the knowledge of the authors, this is the first time a multiphase SPH method [8] is proposed to simulate the dynamic surface tension of surfactant. This fully Lagrangian, grid free method handles discontinuities across phase interfaces naturally and is therefore able to describe the transportation phenomenon of the surfactant at the interface and in the bulk solution. Due to a localy conservative formulation of the surfactant transport even complex interface geometries can be calculated with this model.

Transport of surfactant from the liquid suspension to the air-liquid interface follows an adsorption-limited model taking into account adsorption and desorption [5]. The coupling of interfacial transport and diffusion in the bulk solution is straightforward [9]. As constitutive equation to determine the surface tension in dependence of the interfacial surfactant concentration two linearly isothermal regions are used. So far, only capillary forces normal to the interface are considered, Marangoni forces and tangential transportation of surfactant on the interface are neglected. The extension of the model representing these effects is an ongoing task.

II. Computational Model

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

$$\frac{d\mathbf{v}}{dt} = \mathbf{g} - \frac{1}{\rho}\left[\nabla p + \nabla \cdot \Pi^{(\nu)} + \sigma\left(\nabla \cdot \hat{n}\right)\hat{n}\right] \ , \qquad (1)$$

where ρ, $\mathbf{v}$, and $\mathbf{g}$ are material density, velocity and body force, respectively. The last two terms on the RHS denote the viscous and the surface tension force with the surface-tension coefficient σ, see [8]. Instead of solving the density evolution equation, the particle density ρ_i is computed at each timestep by

$$\rho_i = m_i \sum_j W_{ij} \ , \qquad (2)$$

with $W_{ij} = W\left(r_i - r_j\right)$ and the ,smoothing function $W\left(\mathbf{r}\right)$ with smoothing length h. This form conserves mass exactly and since neighboring particles contribute to the density only by affecting the specific volume of particle i, (2) allows for density discontinuities. Using a weakly-compressible formulation of the particle hydrodynamics, the pressure p and the density ρ can be related by a stiff equation of state

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

with $\gamma = 7$. The parameters p_0, ρ_0 and b determine the threshold of the admissible density variation.

A. Interfacial surfactant concentration

With the SPH method the interface has no sharp representation, rather an interface-layer with the width of the influence domain of the smoothing function exists. As a consequence, also the interfacial surfactant concentration Γ exists not only exactly at the interface but is distributed over the particles adjacent to the interface. Since the exact instantaneous conservation of surfactant in the interface region is an important feature, not the change of concentration but the change of the

Two-phase flow simulations using a volume fraction SPH scheme with a Riemann solver

N. Grenier, D. Le Touzé, P. Ferrant
Laboratoire de Mécanique des Fluides (CNRS UMR6598)
École Centrale de Nantes
1 rue de la Noë - 44300 Nantes - France
nicolas.grenier@ec-nantes.fr

J.-P. Vila
Institut de Mathématiques de Toulouse (CNRS UMR5219)
INSA Toulouse
135 avenue de Rangueil - 31000 Toulouse - France
vila@insa-toulouse.fr

***Abstract*—In the present paper a two-phase SPH model is developed, based on the approach proposed by Cueille and Vila [1]. A volume fraction is used to model the presence of the two immiscible fluids, and a mechanical equilibrium relation closes the system. The latter is relaxed to obtain an effective numerical model. An adequate Riemann solver is also used. The developed model and its implementation in the SPH context are described in the first part of the paper. In a second part, simplified test cases are simulated to validate the model. Then, the rise of an air bubble in water is treated. The effect of using a MUSCL scheme in the Riemann solver is especially discussed.**

I. Introduction

Modelling two phases is required to accurately describe interfacial flow evolutions and in some cases, free-surface ones. Interfacial flows are involved, for instance, in the process of separation of oil from water ahead oil wells. In free-surface flows, the presence of a second phase is crucial in case of air entrapment, sprays, or cushion effects. Interfacial flows are characterized by density ratios often rather low, whereas free surface flows usually involve air and water, that is a high density ratio of 1000.

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

To overcome this key difficulty and to be able to simulate two-phase flows, a new approach of SPH has been proposed by Cueille and Vila [1]. It is based on the main features of the original SPH formulation: it is Lagrangian, compressible and interpolation relies on a smoothing kernel. But the main idea is to suppose that the two phases locally coexist everywhere in the domain and comply with a unique velocity field. Thus, a so-called 'volume fraction' is introduced.

In two dimensions, the system to resolve is composed of two mass conservation equations for each phase and two equations for the momentum, plus a mechanical equilibrium relation (for the mixture pressure) to close this equilibrium model. Relaxing the latter relation one obtains a new model (the relaxation model) which is more adapted for numerical treatment. Following Cueille and Vila [1], this relaxation model is resolved in two steps. In a first step, the hyperbolic step, the system is solved with a numerical scheme based on an exact Riemann solver for each interaction in the compact support. In a second step, pressure relaxation is applied and the equilibrium volume fraction is thus updated without solving a specific transport equation.

A specific attention is paid to enhance the accuracy, especially using a M.U.S.C.L. scheme in the Riemann solver. Actually, the combination of the minmod limiter associated to this scheme with the resolution of the hyperbolic system leads to overshoots of volume fraction at the relaxation step (it should be bounded by 0 and 1), especially when large density ratios are considered. To avoid this, a correction of the slope of conservative variable gradients is proposed.

The formulation is validated on one-dimensional shock tube tests (non-stationary contact discontinuity and rarefaction/shock) and two-dimensional linear sloshing. Finally, to explore the potential of the present formulation a simulation is performed of an air bubble rising by gravity in a water column at rest.

II. Governing equations

Two insviscid immiscible fluids are considered; in the cases studied, a gas of density ρ_g and a liquid of density ρ_l. We suppose that no phase change occurs. An hypothesis of low compressibility is assumed: velocities in each phase stay small compared to the sound speed, disabling all thermal effects. Thus each phase follows an isothermal linearized equation of state:

$$p(\rho) = p_0 + c^2(\rho - \rho_0)$$

where the sound speed c is defined by $c^2 = \frac{\partial p}{\partial \rho}$.

In the present model the two fluids are supposed to be present everywhere. A volume fraction is thus introduced to characterize the presence of each phase, α_g for the gas and α_l for the liquid, with a condition of saturation $\alpha_g + \alpha_l = 1$. To alleviate notations we will note α the gas volume fraction, $(1-\alpha)$ becoming the liquid volume fraction. Moreover, since we deal with immiscible fluids, only one fluid is physically

Lifeboat water entry simulation by the hybrid SPH-FE method

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

***Abstract*—This paper presents numerical simulation results for water entry of a free-fall lifeboat by the hybrid SPH-FE method. Using models in two dimensions, the influence of numerical parameters has been studied. Advanced features in the software allow effective simulation in three dimensions. The presence of air in the model has also been investigated. The results will be compared with experimental data from model scale tests.**

I. INTRODUCTION

For evacuation of the crew from an off-shore platform during emergency situations, free-fall lifeboats are essential. The free-fall lifeboat under consideration is capable of evacuating up to 64 people from a height of up to 32 meter. It is vital to be able to predict the deceleration of the lifeboat during water entry and the trajectory after the impact. In addition to model tests, numerical simulation allows to assess the phenomena of the water impact and the trajectory of the lifeboat after it has hit the water surface. A previous study [1] has already demonstrated that the coupled approach of finite elements and Smoothed Particle Hydrodynamics (SPH) in the PAM-CRASH software allows to simulate most of the phenomena involved. Important issues for the study reported here is to check the influence of numerical parameters and other model assumptions on the results and whether incorporation of air in the models might explain the remaining discrepancies with the model test data identified in the original study[1].

Several new features have been introduced in the software to reduce the CPU required and improve the simulation results. Special attention has been paid to the possible occurrence of air cushioning on the impact on the free surface and on air being dragged beneath the free surface.

II. DESCRIPTION OF THE NUMERICAL SOFTWARE

A. *Overview of the standard PAM-CRASH software and its SPH solver*

The commercial finite element code PAM-CRASH with its explicit solution algorithm may be used to solve dynamic, strongly non-linear structural mechanics. Material models are available for metals and plastics, including plasticity and failure, for rubber, foams and composites. It is also possible to model a liquid by finite elements using either a polynomial equation-of-state or the Murnaghan model. Hence, in regions of limited deformation or flow, the finite element method may also be used to solve the kinematics of a fluid. The SPH method is fully integrated in PAM-CRASH[2]. Unless mentioned otherwise, the SPH discussed here is basically similar to that of Monaghan[3], including the anti-crossing option (XSPH), variable smoothing length and artificial viscosity. Interaction between structures and parts of structures that are not permanently connected by finite element connectivity may be modeled by sliding interface algorithms available in PAM-CRASH, most of them based on the penalty algorithm. This type of contact has been validated for numerous applications[4].

B. *Relevant new features in the v2008 version*

The previously implemented nearest-neighbor search, which was based on a line-sort algorithm, has been replaced by a new algorithm which is based on a 3D-bucket sort. It is the same 3D- algorithm, which is already in use for other purposes in PAM-CRASH. As a further important improvement, an adaptive search frequency computation has been implemented. The major modification in the SPH of v2008 is, however, the implementation of a DMP parallel method, using domain decomposition and technology developed for the particle method used for airbag inflation [2]. The new SMP version is twice as fast as the previous one, and the DMP version scales well up to 16 processors.

Splitting for highly dissipative smoothed particle dynamics

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

***Abstract*— The method of smoothed particle dynamics (SDPD) is novel coarse grained method for simulation of complex fluids. It has some advantages over more traditional particles based methods [1]. But one of the problems common for particle based simulations of microfluid system takes place also for SDPD: it fails to realize Schmidt number of $O(10^3)$ typical of liquids.**

In present paper we apply the implicit numerical scheme that allows significantly increase time step in SDPD and perform simulation for larger Schmidt number. Simulations using this methods show close agreement with serial solutions for Couette and Poiseuille flows. The results of benchmarks based on temperature control are presented. The dependence of self-diffusion coefficient D on kinematic viscosity is examined and found to be in agreement with empirical observations [2].

I. Introduction

A few years ago a new generalization of Smoothed Particle Hydrodynamics methods (SPH) [3] was introduced [4]. This method is called Smoothed Dissipative Particle Hydrodynamics (SDPD) by the authors and claimed to be "improved version" of Dissipative Particle Dynamics (DPD). SDPD has the following features:

- the method is based on a second-order discretization of the Navier-Stokes equations
- transport coefficients can be used as input parameters
- hydrodynamics behavior is obtained at length scales of the same order of particle dimension and no coarse-grained assumption is needed

But beside mentioned advantages we found that SDPD shares some problems known for DPD. In present paper we address one of such issuer known as a *Schmidt number problem.*

Schmidt number is defined as the ratio of momentum diffusivity (viscosity) and mass diffusivity.

$$Sc = \frac{\nu}{D} \tag{1}$$

where ν is a kinematic viscosity and D is a diffusion coefficient.

And it was augured [5] that it is essential to guarantee acceptable liquid behavior to realize Sc of $O(10^3)$ in the simulations. However in varies hydrodynamic simulation done with DPD Sc number was identified as a typical for gases $O(1)$.

We would like to mentioned that there is another opinion in the literature that the self-diffusion coefficient of the particles cannot be equated with those of individual molecules. For example Jiang and coauthors [6] found that for low Schmidt number the hydrodynamic interactions are developed and reveals the correct polymer dynamics.

However as it was shown in the [7] and in [8] that the agreement with experiment is better if Sc approaches liquid values.

As far as in SDPD transport coefficient is a parameter of the model the higher Schmidt number can be achieved by considering smaller Reynolds number. And the problem of Schmidt number is mostly a technical one: the properties of the standard integration schemes make it computational very costly to run simulation for such a high viscosity.

The aim of this paper is to show that the implicit schemes known from the literature [9], [10] allows to increase significantly time step in comparison with velocity-Verlet algorithms without any further assumption and modification of the physical model.

The structure of the report is the following one: in section II we briefly describe the SDPD method, in section III we describe the time step limitation typical for velocity-Verlet and Predictor-Corrector schemes, in section IV the proposed integration scheme is described, sections V and VI dedicated to application of the scheme to macroscopic and microscopic flow respectively.

II. Model

A. Mesoscopic modeling of the liquids

Assuming an isothermal Newtonian solvent described by the following Navier-Stokes equations written in a Lagrangian framework

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

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

A Comparative Study of ANSYS AUTODYN and RSPH Simulations of Blast Waves

Børve, S. and Bjerke, A.
Norwegian Defence Research Establishment (FFI)
Kjeller, Norway
e-mail: Steinar.Borve@ffi.no

Omang, M.
Norwegian Defence Estates Agency
Oslo, Norway

Trulsen J., Børve, S. and Omang, M.
Institute of Theoretical Astrophysics
University of Oslo (UiO)
Oslo, Norway

***Abstract*— In this study, we compare the overall performance of the academically developed code RSPH with the commercially available software AUTODYN on one- and two-dimensional simulations of blast waves, both in Cartesian and cylindrical coordinates. Interactions with solid boundaries are considered. The former code couples a standard SPH description with unique techniques for reducing the degree of disorder in the particle distribution and varying the smoothing length both in time and space. The latter code comprises several solvers, among which are the Godunov and FCT solvers are both suitable tools for studying shock waves. Our study shows the RSPH code to perform well when compared to the FCT solver and distinctly better when compared to the Godunov solver, both in terms of accuracy and efficiency.**

I. Introduction

It is important, both to the military and civilian community, to be able to predict the strength and propagation pattern of shock waves generated from the detonation of high explosives. An example of this is the need to estimate the necessary safety distance near ammunition storage facilities. The nonlinear character of such problems makes numerical simulations essential tools in understanding these types of problems. We currently focus on the importance of using different numerical methods. This is done by comparing two widely different numerical codes on problems which involve the detonation of high explosives (HE). The HE is modelled using the constant volume description [12], [13]. In this description, which is a single-fluid model, the HE is assumed to become instantaneously transformed into hot, dense ideal gas. The adiabatic constant, γ, is equal to 1.4. The two codes compared in this study are the commercially available software AUTODYN and the academically developed method Regularized Smoothed Particle Hydrodynamics (RSPH).

In section II we briefly describe the three solvers to be compared. Then in section III we present results from one- and two-dimensional blast wave tests, before we make a few concluding remarks in section IV.

II. The Numerical Methods

AUTODYN was developed by Century Dynamics but is now part of the ANSYS Workbench Platform and has been used at FFI since 1997. It is an explicit hydrocode that includes several different solvers, including Euler (both Godunov and FCT), Lagrange, Arbitrary Lagrange Euler, Smoothed Particle Hydrodynamics, Shell, and Beam. Each solver has its strengths and weaknesses and is suitable for solving a certain range of problems. When modelling shock waves one would use an Euler solver, either Godunov or Flux-Corrected-Transport (FCT). The Godunov method reaches a solution by solving the local Riemann problems at all cell interfaces [5]. The FCT method combines a convective step with an anti-diffusive step. The latter step, as the name implies, corrects for the numerical diffusion introduced by the former step [1].

RSPH was developed, both as a method and a code, as an extension to SPH [2], [3]. 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 what is the case with standard SPH, resolution no longer needs to be 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 structure, and low resolution elsewhere. Combined with the capability of handling free surfaces, inherited from standard SPH, the adaptive nature of RSPH makes the code ideal for studying multi-dimensional shock wave propagation in an ambient medium. Comparisons with experiments, e. g. [12], [13], have shown good accuracy in simulating high explosives.

III. Simulation Results

In this section we present results from 5 different tests. All the tests are solved in RSPH as two-dimensional tests, although the first test only has spatial variation in one dimension. This test is indeed solved as a one-dimensional problem in AUTODYN. The last test is cylindrically symmetric. Except for the first test, all tests have been formulated as examples of realistic set-ups for experiments with HE.

A. Two Interacting Blast Waves

A much used test problem describing the interaction of two blast wave was proposed by Woodward and Colella in 1984 [17]. It involves multiple interactions of strong shocks and rarefaction waves with each other and with discontinuities. The gas is initially at rest in the interval $0 \leq x \leq 1$ with reflecting

Analysis of SPH and mesh based simulations using point based post processing tool

Yun Jang
Swiss National Supercomputing Center
Data Analysis & Visualization
Manno, Switzerland

Jean-Christophe Marongiu
Laboratory of Fluid Mechanics and Acoustics
Ecole Centrale de Lyon, University of Lyon
Ecully, France

Etienne Parkinson,
Nicolas Gervais, Helene Garcin
Andritz VA TECH HYDRO
Vevey, Switzerland

***Abstract*— In this research, we have been developing a post processing tool for SPH simulation and in use of the tool, we present comparisons of conventional mesh based simulations and new SPH simulations. Our SPH post processing is based not on the SPH kernels, but on points information. Therefore, this tool can be used for both SPH and mesh based simulations. In order to provide proper comparisons, we set the same conditions in both SPH and mesh based simulations and use our post processing tool for SPH simulation. For the mesh based simulations, we only take point information and insert the point information into the SPH post processing tool.**

I. Introduction

Mesh based simulation is the major technique in CFD research for the last few decades and is dominant due to its accuracy and easy analysis of results. In the mesh based simulation, well-defined mesh is a key aspect for accurate results and mesh size may tend to be very large. However, analysis and visualization of large mesh based results are not straightforward because of the massive size of mesh rather than the number of data points in the results. Recently, many researchers have shown interest in simulating CFD problem without meshes, applying mesh-less techniques such as SPH for instance. In SPH, simulation bases are points and one point is interacting with others within its influence range. In the results, only point information is saved and used for analysis and visualization in the post processing.

The post processing of CFD simulation is a way to analyze results and to provide a feedback to analyse the simulated configuration. There are many analysis tools for the mesh based simulations due to the easy access to connectivities between points. Therefore, the analysis and visualization highly depend on the meshes. However, the SPH simulation does not provide the meshes in the results and it is still challenging to provide proper analysis and visualization of SPH results. Especially, when assessing the quality of SPH simulations with respect to their similar mesh based simulations, the comparison can be meaningful only when both SPH and mesh based simulations are analysed with the same analysis and visualization process.

In the following, comparisons are based on various representative cases for internal and free surface flows. To analyse these cases, we probe data with points, lines, discs, and rectangles and compute scalar quantities (discharge, kinetic energy, etc.) and vector quantities (kinetic momentum, force, torque, etc.). Simulation results are compared with measurements when available.

II. Post Processing

Many commercial simulation packages have their own post processing tools. The quantitative post processing is most commonly performed on the basis of probes defined in the simulation domain. It includes visualizations and value integrations in areas in which end users are interested. There is, however, no such tool for SPH results yet because of difficulty in defining standard kernels, and finding neighbors. It is even more problematic when we want to compare SPH results with conventional mesh based simulation results.

In this work, we study a unified tool for the post processing of SPH and mesh data using probe techniques. As mentioned above, we use four different probe primitives including point, line, disc and rectangle. Besides the probing data, integrating flow quantities on disc and rectangle probes is performed so that the integrated quantities can be used as feedback to the analysis process and the simulations.

A. Probes

Point probe is the most primitive probe and it acts as a sensor in the simulation domain. For example, in the case of a static Pelton bucket [1], the point probe emulates a wall pressure sensor. Moreover, the simple point probe can be used to navigate data values in the domain, so that end users can have idea of flow values in the interesting region. The value at a point is calculated using an interpolation method with N neighbors. As extensions of the point probe, line, disc, and rectangular probes can be defined with points with proper setting of resolutions. Figure 1 shows the line, disc, and rectangular probes consisting of sample points.

The probe primitives can be defined as files so that multiple probes can be handled at the same time. For the annotations in the file, please refer to Figure 1. The probe files are defined as follows.

```
Number_PointProbes(n)
Label1 Type X Y Z SolidNumber
Label2 Type X Y Z SolidNumber
.........................
Labeln Type X Y Z SolidNumber
```

The line probe file is defined as

```
Num_LineProbe(n)
```

Gas Accretion from the Elliptic Gas Disk to the Binary System

Yusuke IMAEDA
Kobe University,
Department of Earth and Planetary Science,
Kobe, Japan
imaeda@kobe-u.ac.jp
y.imaeda@gmail.com

Toru TSURIBE
Osaka University,
Department of Earth and Space Science,
Osaka,Japan
tsuribe@vega.ess.sci.osaka-u.ac.jp

Shu-ichiro INUTSUKA
Kyoto University,
Department of Physics,
Kyoto,Japan
inutsuka@tap.scphys.kyoto-u.ac.jp

***Abstract*— We calculate the mass accretion from the eccentric circumbinary disk to the binary stars. Various disk eccentricities are considered. Godunov SPH, which is needed to capture strong shocks around the disk inner edge, is used to calculate the evolution of the eccentric circumbinary disk. The total number of the particle is 65536. Since the cold shear flow is the dominant flow in the system, we also perform the Godunov SPH calculation with Particle Rezoning Method and compare these results. The results of Godunov SPH with PRM calculation give smoother density profile and slightly higher mass accretion rate. However, the tendency does not change that the mass accretion rate drastically increases if the disk eccentricity is high enough. This indicates the rapid disk dissipation when the disk becomes eccentric by some mechanism, e.g. the secular resonance around the eccentric binary.**

I. Introduction

Binary system is widely seen in many astronomical fields such as interacting galaxies, cataclysmic variable stars, black holes, Earth-Moon system, asteroids, and so on. Our interest is a star formation process and the planetary systems. Therefore, we only consider the star-star binary, the star-brown dwarf binary, and the star-planet systems in this paper. These objects are born with a rotationally supported gas disk. Since the molecular cloud, in which stars are born, is slightly rotating initially and the angular momentum is conserved during the cloud collapse, the centrifugal force and the gravity force is balanced at some length scale. As a result of this balance, rotationally supported gas disk is naturally formed during the star formation process. Young binary stars or planets revolve in this gas disk and gradually evolve by tidal interaction.

At the beginning of the tidal evolution, the gas around the binary orbit is quickly removed from the orbital region [1], [2]. The gas accretes onto the stars or is pushed away from the binary orbit. The low density region is formed around the binary orbit, which is called a gap region or an inner hole. The formation time scale of the inner hole is several dynamical time, which is much faster than the other process. The location of the inner edge of the gas disk is determined by the balance of angular momentum transfer. While the tidal torque exerted on the gas disk by the binary pushes away the inner edge of the disk from the binary, viscous mass accretion in the gas disk pulls it back. After the inner hole is formed and the location of the inner edge is determined, mass accretion from the disk to the binary is slowed down and evolution timescale becomes quite large, which is determined by the viscous process in the disk.

On the other hand, if the binary stars revolve on the elliptic orbit, the tidal force, exerted on the surrounded gas disk, excites the eccentricity waves in the disk. When one of the pattern speeds of the wave is close to the precession rate of the binary, the resonance occurs and the disk can be highly eccentric. If the disk becomes eccentric, the pericenter of the inner edge approaches the binary orbit and the apocenter recedes from the binary. The deformation timescale is a precessional timescale, which is much larger than the dynamical timescale but shorter than the viscous timescale ($t_{\rm dyn} \ll t_{\rm pre} < t_{\rm vis}$). When the disk edge approaches the binary orbit, the dynamical accretion process can be reactivated and may change the evolution time scale of the binary. Therefore, it is important to study the gas accretion process from the eccentric gas disk to the binary to understand the long-term evolution of the binary.

Direct numerical calculation is a powerful tool to investigate such dynamical fluid interactions between the binary and the gas disk. However, we should always keep in mind that numerical calculation always includes some numerical errors in the results, and when we consider the long-term evolution, these errors accumulate. In order to resolve the gas rotation around the binary, the time step of the calculation should be much smaller than the dynamical timescale, thus we need enormous time steps for the long-term evolution.

Therefore, we take two-step approach to investigate the long-term evolution of the disk-binary interaction in this paper. We divide the evolution into secular part and the nonsecular part. In §2, we briefly summarize how to calculate the secular evolution of the gas disk, and shows that the gas disk around the eccentric binary can becomes highly eccentric, without changing the semimajor axis. Then, in §3, we calculate the short dynamical evolution. SPH is used to calculate the dynamical evolution of this system. We start the calculation when the elliptic deformation of the disk has already occurred as a consequence of the secular evolution. Since the rotationally supported gas disk is a cold and strong shearing system, we should be careful about the density errors in SPH [3], [4], and

Smoothed Particle Hydrodynamics in Thermal Phases of a One Dimensional Molecular Cloud

Mohsen Nejad-Asghar
Department of Physics,
Damghan University of Basic Sciences,
Damghan, Iran
nasghar@dubs.ac.ir

Diego Molteni
Dipartimento di Fisica e Tecnologie Relative,
Universita di Palermo, Viale delle Scienze, 90128,
Palermo, Italy
molteni@unipa.it

***Abstract*— We present an investigation on effect of the ion-neutral (or ambipolar) diffusion heating rate on thermal phases of a molecular cloud. We use the modeling of ambipolar diffusion with two-fluid smoothed particle hydrodynamics, as discussed by Nejad-Asghar & Molteni. We take into account the ambipolar drift heating rate on the net cooling function of the molecular clouds, and we investigate the thermal phases in a self-gravitating magnetized one dimensional slab. The results show that the isobaric thermal instability criterion is satisfied in the outer parts of the cloud, thus, these regions are thermally unstable while the inner part is stable. This feature may be responsible for the planet formation in the outer parts of a collapsing molecular cloud and/or may also be relevant for the formation of star forming dense cores in the clumps.**

I. Introduction

In most plasmas, the forces acting on the ions are different from those acting on the electrons, so naively one would expect one species to be transported faster than the other, by diffusion or convection or some other process. If such differential transport has a divergence, then it will result in a change of the charge density, which will in turn create an electric field that will alter the transport of one or both species in such a way that they become equal. In astrophysics, *ambipolar diffusion* refers specifically to the decoupling of neutral particles from plasma. The neutral particles in this case are mostly hydrogen molecules in a cloud, and the plasma is composed of ions (mostly protons) and electrons, which are tied to the interstellar magnetic field.

Nejad-Asghar (2007) has recently made the assumption that the molecular cloud is initially an uniform ensemble which then fragments due to thermal instability. He finds that ambipolar drift heating is inversely proportional to density and its value, in outer parts of the cloud, can be significantly larger than the average heating rates of cosmic rays and turbulent motions. His results show that the isobaric thermal instability can occur in outer regions of the cloud. The rapid growth of thermal instability results a strong density imbalance between the cloud and the low-density surroundings; therefore it may produce the cloud fragmentation and formation of the condensations.

Many authors have developed computer codes that attempt to model ambipolar diffusion. Black & Scott (1982) used a two-dimensional, deformable-grid algorithm to follow the collapse of isothermal, non-rotating magnetized cloud. The three-dimensional work of MacLow et al. (1995) treats the two-fluid model in a version of the ZEUS magnetohydrodynamic code. An algorithm capable of using the smoothed particle hydrodynamics (SPH) to implement the ambipolar diffusion in a fully three-dimensional, self-gravitating system was developed by Hosking & Whitworth (2004). They described the SPH implementation of two-fluid technique that was tested by modeling the evolution of a dense core, which is initially thermally supercritical but magnetically subcritical. Nejad-Asghar & Molteni (2007) have recently optimized the two-fluid SPH implementation to test the pioneer works on the behavior of the ambipolar diffusion in an isothermal self-gravitating molecular layer.

In this paper, we include the SPH equivalent of the energy equation, which follows from the first law of thermodynamics. Here, we include the ambipolar drift heating rate in the net cooling function, and we investigate the thermal phases in the self-gravitating magnetized molecular layer. For this purpose, the two-fluid SPH technique and its algorithm are given in section 2. Section 3 devotes to the basic equations of the one dimensional molecular layer and its evolution by SPH. The summary and conclusion are presented in section 4.

II. Numerical method

In two-fluid SPH technique of Hosking & Whitworth (2004), the initial SPH particles represent by two sets of *molecular particles*: magnetized ion SPH particles and non-magnetized neutral SPH particles. In this method, for each SPH particle we must create two separate neighbor lists: one for neighbors of the same species and another for those of different species. Consequently, each particle has two different smoothing lengths. In the following sections we refer to neutral particles as α and β, and ion particles as a and b; the subscripts 1 and 2 refer to both ions and neutral particles. We adopt the usual smoothing spline-based kernel (Monaghan & Lattanzio 1985) and apply the symmetrized form proposed by Hernquist & Katz (1989)

$$W_{12} = W(\mid \mathbf{r}_1 - \mathbf{r}_2 \mid, h_{12}) \tag{1}$$

where $\mathbf{r}_1, \mathbf{r}_2$ are positions of the particles 1 and 2, respectively. h_{12} is the smoothing length of particle 1 when considering

Accelerating Smoothed Particle Hydrodynamics for Astrophysical Simulations: A comparison of FPGAs and GPUs

G. Marcus, A. Kugel, R. Männer
Dept. of Computer Science V
ZITI, University of Heidelberg
Mannheim, Germany

P. Berczik, I. Berentzen, R. Spurzem
Astronomisches Rechen-Institut
ZAH, University of Heidelberg
Heidelberg, Germany

T. Naab, M. Hilz, A. Burkert
University Observatory Munich
University of Munich
München, Germany

***Abstract*— A comparison of two hardware accelerated platforms for computational fluid dynamics is presented. Both platforms are compared using the same application, the same software interfaces, and differ almost only in the accelerator. This allows a very direct comparison of their capabilities in terms of performance. Compared to a workstation, an FPGA provides 6-11x speedup, while a GPU provides 17x speedup under the same conditions.**

I. Introduction

Particle-based astrophysical simulation determine the motion of individual particles according to a model for the interaction forces. Long-range gravity and short-range hydrodynamical forces are amongst the most important ones for many systems. The most time consuming part of these algorithms has always been the gravity interactions, and its acceleration has been widely explored, first with ASICs like the GRAPE6 board [3], Field Programmable Gate Arrays (FPGAs) [4] and more recently with Graphics Processing Units (GPUs) [5] [9] [1]. Current implementations achieve over 300 GFlops from a single board. This is quite logical, as it represents $> 90\%$ of time for pure-CPU algorithms, it is an $O(n^2)$ in its more basic form, and highly parallelizable.

With particle numbers $> 10^4$, the latter Smoothed Particle Hydrodynamic (SPH) part is computationally very demanding, once the gravity has been accelerated. It is also more complex, as it is based on interaction lists between particles. We support such calculations, which can be applied to a wide variety of practical problems besides astrophysics, by programmable hardware. Related work include the developments of Hamada and Nakasato [8] and similar work has been done for Molecular Dynamics (MD) [10], as the algorithms are comparable.

In the following sections, we present the mathematical formulation implemented in the accelerator platforms, a description of our implementations both in FPGA and GPUs and the related hardware architectures, the software tools and interfaces used, closing with the results obtained.

II. FPGA implementation

In this section, we describe the implementation of an FPGA-based coprocessor for SPH computations. Field Programmable Gate Arrays are specialized chips that allow dynamic reconfiguration of its building blocks. Giving this capability, they are able to implement electronic circuits, or hardware designs (as we call them), by means of programming the interconnections between the blocks. This requires the design of floating point operators to support the required computations, to assemble them into a processing pipeline, and building the required control and communication logic.

As our platform of choice, we use the MPRACE-1 coprocessor boards developed in-house, which are equipped with a Xilinx XC2V3000 FPGA, 8MB of ZBT-RAM and an external PLX9656 for PCI communications with the host at up to 264MB/s. The migration to our new MPRACE-2 boards is currently in progress (see Conclusions).

A. Equations implemented

We use the standard SPH formulation as the base and modify it in order to reduce the actual number of computations done in hardware. As kernel functions, we use:

$$W'(x) = \begin{cases} 1 - \frac{3}{2}x^2 + \frac{3}{4}x^3 & \text{if } 0 <= x < 1 \\ \frac{1}{4}(2-x)^3 & \text{if } 1 <= x < 2 \\ 0 & \text{otherwise} \end{cases}$$

And the scalar part of the gradient of W as:

$$\Omega(x) = \begin{cases} -1 + \frac{3}{4}x & \text{if } 0 <= x < 1 \\ -\frac{1}{4}x + 1 - \frac{1}{x} & \text{if } 1 <= x < 2 \\ 0 & \text{otherwise} \end{cases}$$

In Step 1 we compute the density, curl and divergence of the velocity as:

$$\pi\rho_i = \sum_j \frac{m_j}{h_{ij}^3} W'(|\vec{r}_{ij}|/h_{ij})$$

$$-\frac{\pi}{3}\rho_i(\nabla \cdot \vec{\nu}) = \sum_j \frac{m_j}{h_{ij}^5}(\vec{\nu}_{ij} \cdot \vec{r}_{ij})\Omega(|\vec{r}_{ij}|/h_{ij})$$

$$-\frac{\pi}{3}\rho_i(\nabla \times \vec{\nu}) = \sum_j \frac{m_j}{h_{ij}^5}(\vec{\nu}_{ij} \times \vec{r}_{ij})\Omega(|\vec{r}_{ij}|/h_{ij})$$

Reynolds number and Shallow Depth Sloshing

A. Colagrossi
G. Colicchio
INSEAN
00128 Rome, Italy.
a.colagrossi@insean.it

L. Delorme
Eurocopter, ETDAO
13700 Marignane, France
louis.delorme@eurocopter.com

A. Souto-Iglesias
J.L. Cercós-Pita
Naval Architecture Department (ETSIN),
Technical University of Madrid (UPM)
28040 Madrid, Spain.
antonio.souto@upm.es

***Abstract*— The dependence on the Reynolds number of shallow depth sloshing flows inside rectangular tanks subjected to forced harmonic motion is studied in this paper with weakly-compressible SPH. We are interested in assessing the influence of viscous effects on the dynamics of shallow depth sloshing flows by using an SPH solver and by comparing with a Navier-Stokes level-set solver results. The goal of trying to model these viscous flows is compromised by the resolution requested due to their Reynolds number, if boundary layer effects are to be modeled. The convenience and feasibility of the implementation of free-slip and no-slip boundary conditions is also discussed.**

I. Introduction

Sloshing flows are those occurring when free surface waves are generated inside tanks, usually creating significant torque and pressure peaks on the tank due to the impact of traveling waves. These phenomena are of interest for several branches of engineering including marine, aerospace and civil engineering. An abundant literature on sloshing can be found, reviewed in the indispensable book of Prof. Raouf Ibrahim [1], one of the most prominent researchers in the field.

Among the sloshing flows, shallow depth ones are specially attractive due to the structure of the wave systems that are generated under these depth conditions. When dealing with fluids of small viscosity like water, for low frequencies a set of small traveling waves appear. With a small rise in frequency, the train of waves is suddenly transformed into a bore, or hydraulic jump, a distinct step in the water surface [2].

We are interested in assessing the effect on these shallow depth flows of modifications in the Reynolds number due to changes in the physical viscosity. For shallow water flows, previous attempts with SPH codes [3], [4] have been successful in modeling both the bulk flow in terms of the torque [4] and the wave height, even for resonant conditions [3]. Nevertheless concerns on the role of the SPH viscous term [5] on the simulations, when focusing on the time evolution of the bores, have suggested to pursuing the study by checking the flow dependence on the Reynolds number Re, keeping constant all the parameters apart from the physical viscosity. The initial results of such study are presented in this paper. Also, the convenience and feasibility of the implementation of free-slip and no-slip conditions is discussed, paying attention to the characteristic dimensions of the boundary layers (BL) to model, to the computational effort required and to the quality of the results in terms of a correct approximation of the viscous stress term in the SPH momentum equation.

Although comparing experimental data is on our agenda, it has not been possible for this article. Nevertheless, having information from a better established numerical technique, at least for the pre-splashing part of the motion, is also necessary to assess the quality and limitations of the SPH approaches. This reference data has been specifically produced for this study with a Navier-Stokes solver combined with a level-set technique (NSLS) for the tracking of the free surface evolution [6].

The paper is organized as suggested in this introduction: first, the case studies are exposed, documenting the specific flows to resolve as well as the fluid physical properties. Second, the SPH treatment of the viscosity and the BL numerical treatment are discussed. Finally, comparisons with the level set technique data will be provided.

II. Case studies

A. Shallow water flows

The simplest mean to characterize shallow depth sloshing flows is to resort to the dispersion relation for gravity waves in limited depth areas [7].

$$\omega^2 = g\,k\ \tanh\left(k\,H\right) \tag{1}$$

In this expression, g is the gravitational acceleration, d the liquid level, ω is often called the sloshing frequency and k its corresponding wave number. The free surface height function in a two dimensional rectangular container partially filled with liquid can be decomposed in a Fourier series with infinite wave numbers $k_n = n\pi/L$ where L is the tank breadth. If we focus on the first mode, the hyperbolic tangent argument of equation 1 becomes a factor of the ratio H/L. If the depth is great, then the tangent tends to one. If the ratio is for instance 1, the tangent value is 0.996, which means that the effect of the depth on the waves will be very small. In this study a ratio of 0.06 has been chosen, for which the tangent factor is 0.19, which means that this is clearly a shallow water case. The first sloshing frequency, the one corresponding to the first mode will be noted w_1.

The tank is subjected to sway motion with a sinusoidal type excitation. The amplitude A of this motion will be set constant for this study with a value of $A/L = 0.03$. The frequency range for the excitation is $0.5 \leq \omega/\omega_1 \leq 2.0$. The tank length L is 1 meter.

SPH conservation of circulation in breaking wave processes

M. Antuono, A. Colagrossi,
INSEAN (The Italian Model Basin),
Rome, Italy;
e-mail: matteoantuono@gmail.com
a.colagrossi@insean.it

J.J. Monaghan,
Monash University,
Monash, Australia;
joe.monaghan@sci.monash.edu.au

D. Le Touzé,
Ecole Centrale de Nantes (ECN),
Nantes, France;
e-mail: david.letouze@ec-nantes.fr

***Abstract*— In the present work we check the ability of the Smoothed Particle Hydrodynamics solvers to correctly account for both the conservation and generation of circulation during breaking wave processes. This is made by considering different examples of increasing complexity and testing the numerical results through the Stokes and Kelvin theorems. In all the cases the SPH seems to correctly predict both the conservation and the evolution of the vorticity and circulation.**

I. Introduction

During plunging breaking wave processes large vorticity generation is observed as a consequence of energetic splashing events and complex free-surface behavior. Indeed, during the pre-breaking evolution waves steepen, become asymmetric and eventually overturn in the form of a forward-plunging jet hitting the underlying free-surface. This leads to the formation of a cavity with large circulation generation due to topological reasons [1]. However, after the generation process ends, as a consequence of the Kelvin theorem for an inviscid barotropic flow with conservative body forces, the global circulation has to be preserved. Then, the aim of the present work is to check the ability of the SPH scheme to satisfy such a condition. However, to stabilize numerical schemes an artificial viscosity and/or other corrections are generally introduced and, consequently, the conservation of circulation is not guaranteed anymore. Anyway, increasing the spatial resolution, the consistency of the numerical corrections cited above implies the recovery of Kelvin statement [2]. To check this assumption, we heuristically estimate the accuracy of the circulation preservation in SPH simulations. The circulation invariant has a dynamic part (depending on how well the velocity is calculated), and a topological part (depending on how complex is the integration contour). The latter can often require much greater resolution than the former [2]. Thus, before proceeding to the analysis of the breaking waves, we briefly investigate some prototype cases evaluating the circulation along different close contours.

As a first case we analyze a fluid cylinder rotating with constant angular velocity. In this case, the circulation predicted by the SPH model is conserved in accordance with the theoretical solution. Then, we consider two separate rectangles of fluids colliding with shear in different ways. In all the cases the initial vorticity within each fluid patch is zero. However, after the impact, a vortex layer is generated and the global circulation is no more zero. This process is similar to what happens when a wave breaks and re-connects. Finally the problem of a two dimensional plunging breaking wave is analyzed. In the simulations, the flow starts with a long layer of fluid initially at rest and forced into motion by a vertical piston moving with constant speed. With respect to the previous prototype cases, in the breaking wave event we both face fluid/fluid interaction and the topological problem connected with the cavity generation. The temporal evolution of circulation is followed during three stages: a) pre-breaking b) impact and ricochet of the plunging jet c) post-breaking evolution. During the latter stages (b,c) the artificial viscosity acts largely since the mechanical energy is numerically dissipated during impact phenomena where the velocity field is characterized by a discontinuity along the impact line. The effect of the spatial resolution is studied showing that increasing the number of particles, the net circulation becomes closer to the vorticity volume integration. However, the rate of this convergence it is not constant in time since the complexity of the integrating loop changes dynamically.

II. Governing equations

We briefly summarize the theoretical scheme used by the SPH solvers. Since the fluid is assumed to be weakly compressible, the continuity equation for compressible fluids is used together with the Euler equation:

$$\frac{D\rho}{Dt} + \rho \nabla \cdot \mathbf{u} = 0 \tag{1}$$

$$\rho \frac{D\mathbf{u}}{Dt} = -\nabla p + \rho \mathbf{f} \,. \tag{2}$$

Moreover, the fluid is assumed to be barotropic and, therefore, the following functional dependence between ρ and p is adopted:

$$p(\rho) = \frac{\cancel{c_0^2 \rho_0}}{\gamma} \left[\left(\frac{\cancel{\rho}}{\rho_0} \right)^{\gamma} - 1 \right], \tag{3}$$

where ρ_0 is constant, c_0 is the sound velocity and γ is a dimensionless parameter greater than 1 (in all the following examples $\gamma = 7$ is used).

Simulation of Wave Impact Pressure on Vertical Structutes with the SPH Method

Giacomo Viccione, Fabio Dentale, Vittorio Bovolin
Dept. of Civil Engineering
University of Salerno
Via Ponte Don Melillo, 84084, Fisciano (Italy)
gviccion@unisa.it, f.dentale@unisa.it, v.bovolin@unisa.it

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

***Abstract*— Such is the complexity of wave-structure interaction phenomena that most available theories and procedures for the design of maritime structures – both vertical and rubble mound breakwaters - are still derived by fitting simple conceptual models to the results of physical experiments, either in the field or in the laboratories. Recent developments of CFD techniques are now supplying a better understanding of the hydrodynamic phenomena involved. In particular, Eulerian Navier Stokes codes with special free surface treatment have reached a mature stage, as in [1], [2] so that operational design tools based on such techniques seem to be not too far away in the future, at least for low slope structures. Vertical and near vertical structures, however, seem to pose a bigger challenge since simulating wave impact on a steep wall, with its related sudden changes of the free surface, is a difficult task to achieve with classical Eulerian techniques. This is a sector where Lagrangian hydrodynamic methods may offer useful solutions. The Authors' past experience in SPH code efficiency and reliability analysis [3], [4], together with their long standing experience in maritime wave problems [5], [6], provide the background for the work to be presented at the workshop. Their own SPH parallel code is applied to investigate time and space distribution of wave pressure on a vertical marine structure. Results are then compared with the ones obtained from the Eulerian based program Flow 3D, showing a good agreement.**

I. INTRODUCTION

Coastal structures may be object of severe pressures induced by breaking waves. Typically, such dynamic actions consist of short, high impulses, reaching up to hundreds of kN/m^2, followed by longer, smaller hydrodynamic forces [7]. Prediction of such evolving pressures is not straight: estimates are given by several semi-empirical formulas, such as in [8], [9]. Their main disadvantage is due to their extremely sensibility to the particular situation they are referred to. In addition, wave impact pressures are influenced by the amount of entrapped air as shown in [8].

Recently, thanks to both the increasing power of computers and the reliability of numerical techniques, Computational Fluid Dynamics yields a better understanding of the hydrodynamic processes involved into breaking waves over seawall structures. Eulerian based schemes, discretizing the well known Shallow Water equations (SWE), give satisfactory results, at least for low seabed slops. On the other hand, impact on a vertical or structures of travelling waves over steep bottoms involves the propagation of not gradually varied flows. In such cases, depth averaged Navier Stokes (N-S) equations, that is SWEs, seem to be improper to accurately reproduce such phenomena.

Lagrangian description of N-S equations by means of the Smoothed Particle Hydrodynamics (SPH) technique [10], [11], overcomes the above mentioned difficulties. Its capability to well reproduce rapidly varied free surface flows has been proved over the past few decades [12]-[14]. A brief overview of the method is presented in the following. A comprehensive review of its theoretical aspects has recently been published [15], so they need not be fully discussed here.

II. THE SPH METHOD

With this method, the original physical domain is discretized by a finite number of particles which represent small volumes of the system. They move in response to forces such as gravity and pressure, carrying at the same time physical properties. A generic scalar or vector field f_i, at a given position $\underline{r}_i$, taken by the fluid particle "i", is given by a smooth interpolation from the values f_k carried by those N_n particles within the interaction sphere of radius $r_c = 2h$ [16]:

$$f_i = \sum_{k=1}^{N_n} f_k \cdot W\left(\left|\underline{r}_i - \underline{r}_k\right|, h\right) \qquad (1)$$

where "W" is the so called kernel or weighting function and "h" is known as the smoothing length. Here the cubic spline devised by Monaghan and Lattanzio [17] is adopted:

SPH Study of High Speed Ship Slamming

D. J. Veen and T. P. Gourlay
Centre for Marine Science and Technology
Curtin University of Technology
Perth, Western Australia

***Abstract* — A preliminary investigation of wet-deck slamming of high speed multi-hull ships using Smoothed Particle Hydrodynamics (SPH) is discussed. The SPH algorithm is applied to the two-dimensional dam beak test case for the purposes of validation and the results compared with published experimental and SPH data. Validation of water sitting in a Tank is also presented, along with preliminary results for a two-dimensional wedge entering a free surface.**

I. INTRODUCTION

Numerical modelling of large floating objects in rough seas has met with limited success using available techniques in Computational Fluid Dynamics (CFD) particularly when wave impacts are involved. An understanding of the sea loads on high speed multi-hull ships is essential to ensuring the structural design criteria are met and assessing the fatigue life of the structure [1]. In particular, high speed catamarans and trimarans are prone to wet deck slam events, where the relative vertical motion of the water surface and ship bow causes water to impact the wet deck. Such events subject the wet deck to high localised pressure in the region of impact and also induce hull vibrations, commonly known as whipping, which can place significant global loads on the structure.

The current research aims to use Smoothed Particle Hydrodynamics (SPH) to model the impact of sea water on various hull shapes examined experimentally in [2] using an SPH algorithm written in Matlab®.

II. SPH METHODOLOGY

The following SPH methods have been used during the validation and preliminary slamming investigation:

- The momentum equation due to [3].
- The differential equation for continuity as quoted by [3].
- The Cubic Spline Kernel described by [3].
- The equation of state according to [4].
- The artificial viscosity described by [3] with $\alpha = 0.01$ and $\beta = 0$.
- The particles were moved using the XSPH variant according to [3] with $\varepsilon = 0.5$ and a modified Kernel.
- Reinitialisation of the density field described by [5].
- The Lennard-Jones boundary method quoted by [3].
- The predictor-corrector time stepping method [6].
- A fixed time step at 80% of the Courant Frederichs-Levy condition limit and a fixed smoothing length based on the initial particle separation [7].

III. MODEL VALIDATION

The initial stages of the project required the validation of the SPH method in two dimensions. The two-dimensional dam break was chosen as the most suitable validation case and the results compared against the experimental data of [8] and the SPH results produced by [5] and [6].

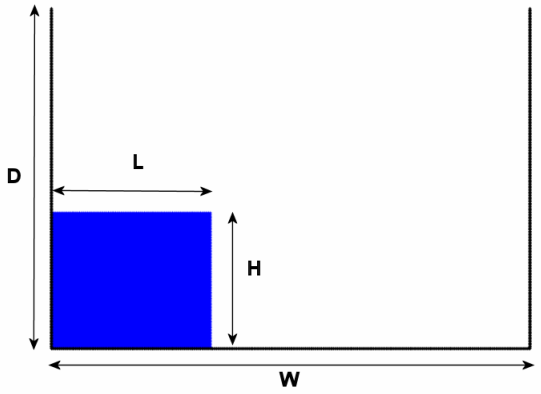


Figure 1. Schematic of the two-dimensional dam break.

Investigation of Wave Loading on a Partially Submerged Cylinder Using SPH

Pourya Omidvar,
School of Mechanical, Aerospace and Civil Engineering,
The University of Manchester,
Manchester, UK.
Pourya.Omidvar@postgrad.manchster.ac.uk

Benedict D Rogers,
School of Mechanical, Aerospace and Civil Engineering,
The University of Manchester,
Manchester, UK.
Benedict.Rogers@manchester.ac.uk

Peter K Stansby,
School of Mechanical, Aerospace and Civil Engineering,
The University of Manchester,
Manchester, UK.
P.K.Stansby@manchester.ac.uk

Abstract—**In this work, Smoothed Particle Hydrodynamics (SPH) is used to investigate the loading on a partially submerged fixed cylinder in the presence of regular waves in a two-dimensional deep water channel by using the code SPHysics. The results are compared with the experimental data of Dixon *et al.* (1979) and agreement is shown to be promising.**

I. Introduction

The work in this PhD is a part of a joint project between the Universities of Plymouth, Manchester, Oxford and Manchester Metropolitan funded by EPSRC looking at the response of wave energy devices under extreme wave loading. At the University of Manchester, we are using Smoothed Particle Hydrodynamics SPH to simulate wave propagation and loading on the wave energy device in a water channel.

As an initial step, the SPH method is used to study the loading on a partially submerged fixed cylinder in the presence of regular waves in a two-dimensional deep water channel using the code SPHysics. The results will be compared with the data of Dixon *et al*. (1979) and discussed.

II. Experiment

The experiment data of Dixon *et al*. (1979) are used in order to validate the SPH results. In their paper, first, the theory of wave forces was presented followed by the modification of vertical force theoretical formulation. Then, the force on a circular cylinder was calculated using linear wave theory for different water levels and compared with the experimental data. In this work, we use their experimental data to validate our SPH results for constant wave frequency and various wave amplitudes and axis depths.

III. Numerical Investigation

One of the main advantages of the SPH method is the ease of implementation of such a complex phenomenon like free surface flow. After the successful investigation of wave propagation in a long and intermediate depth water channel, we are currently studying wave propagation over a fixed partially submerged cylinder for different submergences and wave amplitudes. FIG. 1 shows the schematic of the problem where waves are generated by a paddle at the left boundary and absorbed by a so-called sponge layer at the right boundary. The length and height of the channel are l and h, respectively. The fixed body is located in the middle of the channel with a diameter of D and axis depth below mean water level, d'.

A. Wave forces

The dimensionless force F' is determined as the ratio of wave force to the weight of water displaced by a totally submerged cylinder in still water to compare with experimental data.

B. Boundary condition

The repulsive boundary condition, developed by Monaghan (1999), is used which prevents a water particle crossing a solid boundary.

New features and applications of the hybrid SPH/FE approach in PAM-CRASH

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

Abstract **The capabilities of the SPH processor in PAM-CRASH version 2008 have been significantly enhanced by a review of the numerical efficiency, and by making an DMP parallel version of the method available. This allows to conduct simulations with many more particles than in the past within acceptable turn-around time. Other options introduced are to initialize water at hydrostatic equilibrium under gravity load for particles and finite elements, optional neglect of particles outside regions of interest, and gauge points for pressure and free surface elevation. An option to switch from finite elements to particles during the simulation has also been made available for liquid material models. The advantages of these options will be presented for a few applications. Other applications that will be presented are that of air flow about a moving obstacle, air cushioning during water entry and waves for the Wigley hull.**

I. INTRODUCTION

New developments and selected applications of PAM-CRASH with the integrated SPH solver are discussed.

II. UPDATED SPH VERSION IN PAM-CRASH

The nearest-neighbor (NN) search in the previous version, which was based on a line-sort algorithm, has been replaced by a new algorithm, which is based on a 3D-bucket sort. It is the same 3D-bucket sort algorithm, which is already used for other purposes in PAM-CRASH..

As a further important improvement, an adaptive search frequency computation has been implemented. It is a proven technique, which is well known from the contact treatment and another particle method. The NN-search is triggered by the displacements of the particles, which are checked against threshold values at each cycle.

The major modification in the SPH of v2008 is, however, the implementation of a DMP parallel method, using domain decomposition. Figure 1 shows the relative performance for a lifeboat impact case with 86.400 particles. The new SMP version is twice as fast as the previous one, and the DMP version scales well up to 16 processors.

In order to minimize the communication between the processors for the DMP version, the components of the SPH solver have been rearranged slightly. The computation of the anti-crossing forces (or XPSH version) is now done based on the velocities of the previous cycle, but on the nearest-neighbour configuration of the current cycle and not of the previous one, as done in the original version Although the application of anti-crossing forces is a numerical method to prevent the particles from clustering together under certain loading conditions, and the change in the algorithm should not have any impact on the quality of the solution, the results may slightly differ in comparison to the original version.

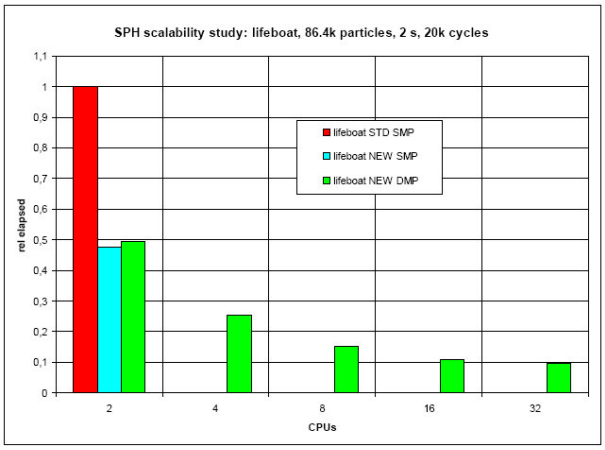


Figure 1. *Relative CPU performance versus the number of CPU's for the SMP and DMP version of PAM-CRASH compared to the original version.*

III. GAUGE POINTS

For hydrodynamic simulations with SPH it is very useful to define the concept of gauge nodes at which output is obtained for the pressure and, if relevant, for the free surface location. These gauges may be defined as moving along with the structure (as FE nodes), given a fixed motion (as free nodes), or as particles. The main advantages are that information about the pressure may be obtained at any

A regularized Lagrangian finite point method for incompressible viscous flows

Jiannong Fang and Aurèle Parriaux
Engineering and Environmental Geology Laboratory
Ecole Polytechnique Fédérale de Lausanne (EPFL)
Station 18, 1015 Lausanne, Switzerland
Email: jiannong.fang@epfl.ch

***Abstract*—In this paper we present a regularized Lagrangian finite point method (RLFPM) for the numerical simulation of incompressible viscous flows. The pressure Poisson equation with Neumann boundary condition is solved by a stabilized finite point method to satisfy incompressible constraint. A key aspect of the present approach is the periodic redistribution of the particle locations, which are being distorted by the flow. Weighted least squares approximation is implemented to interpolate the properties of the old particles onto the new particle locations. With the proposed regularization technique, problems associated with the flow-induced irregularity of particle distribution in the Lagrangian finite point scheme are circumvented. Three numerical examples are presented to validate the proposed methodology. The results demonstrate that RLFPM is able to perform accurate and stable simulations while maintaining the meshless feature.**

I. Introduction

Among the various truly meshless methods, Smoothed Particle Hydrodynamics (SPH) is the longest established grid free Lagrangian method. SPH was originally developed for astrophysical applications [1]. Since its invention, it has been extensively studied, extended and applied in many areas such as the dynamic response of elasto-plastic materials [2], [3], free surface flows [4], viscous flows [5], incompressible fluids [6], and viscoelastic flows [7]. Yet, for a long time, SPH was suffering from several problems: inconsistency; tensile instability; and difficulty in the treatment of boundary conditions. The latter two problems are somehow related to the first one. Various methods have been developed to improve SPH. Most of them have focused on replacing directly the existing SPH particle approximations to functions and their derivatives by corrected approximations with polynomial consistency enforced. As a consequence, angular momentum and/or linear momentum are usually not strictly conserved by the discrete equations and the long-time simulations are potentially unstable. Differently, Bonet and Lok [8] presented a corrected SPH formulation using a variational framework with the advantage of preserving both linear and angular momentum. Similarly, using an energy-based approach, Fang *et al*. [9] derived a general set of discrete hydrodynamics equations with conservation properties to which any corrected (high-order) or coupled particle approximation scheme can be applied. From either a variational or an energy-based approach, a close link is found between discrete velocity derivatives and momentum equation, which can be named as the scheme coherence principle. The scheme coherence principle ensures the conservation of momentum. However, it can not be satisfied together with the consistency requirement for the discrete momentum equation. Recently, Oger *et al*. [10] has shown that the scheme coherence principle could be flouted, allowing the introduction of a more accurate SPH formulation.

Finite point method (FPM) is another truly meshless method proposed by Oñate *et al*. [11]. FPM uses a weighted least square interpolation scheme within each point cloud, which can be easily constructed to have consistency of a desired order, and adopts the point collocation method to obtain the discrete equations. Therefore, it is easy for numerical implementation and boundary conditions can be implemented in a natural way by just prescribing boundary conditions on points placed on boundaries. Moreover, not like in the classical SPH method, particles in the finite point method do not have a mass associated and they are merely interpolation points for the field information, including density. This makes the method more flexible in terms of an easier particle management and an easier treatment of boundaries. The price to pay for these advantages is that the method is not strictly conservative anymore, as opposed to SPH. The efficiency of FPM for solving a range of problems has been reported in many applications. In the same spirit of the finite point method, but using a Lagrangian description, Tiwari and Kuhnert have developed the Lagrangian finite point method (LFPM) [12]. Be a strong-form method, stabilization of FPM is often necessary. Several approaches are available for that purpose.

Generally speaking, Lagrangian particle methods can handle convection dominated flows and large deformation problems very well due to their Lagrangian and adaptive nature. However, it is also known that, with a fixed support domain (or an Eulerian kernel) for each particle, it is difficult to maintain accurate interpolation based on the Lagrangian particle set, which is usually distorted by the flow map and hence loses its regularity. Sometimes particles undergo so massive amounts of strain that they do not overlap anymore in some directions. SPH solutions obtained from increasingly more irregular particle distributions will exhibit an increasing amount of numerical errors. This is the reason for the accuracy degradation of some SPH simulations. For LFPM, it is even possible to

SPH Simulation of the Flow in a Spring Safety Valve

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

***Abstract*—The simulation of the transient flow during the opening of a spring safety valve is obtained by Smoothed Particle Hydrodynamics. A Weakly Compressible SPH formulation with an explicit time integration scheme is adopted. The method is applied to the analysis of an industrial safety valve, showing the different valve behaviours and the different flow patterns which can occur when varying the total pressure at the valve inlet. In particular, it is shown that partial opening may occur when the inlet total pressure is almost equal to the threshold pressure of the valve: the plug reaches an intermediate travel and then closes, leading to an intermittent jet in the orifice between the nozzle and the plug itself. At slightly higher inlet total pressures, the safety valve shows its correct behaviour: the plug reaches its maximum travel in a finite time and a steady radial jet flows along the plug surface to the outlet valve section.**

I. INTRODUCTION

Safety valves are used to protect pressurized systems in industrial installations from sudden unwanted pressure surges which exceed the limits of safe operation. When the pressure in a flow line or in a pressure vessel raises over a fixed threshold value, the safety valve discharges part of the flow rate, limiting either the intensity of the pressure wave transmitted downstream or the unwanted excess pressure in the vessel [1].

A safety valve usually consists of a short nozzle or pipe, closed by a plug which is controlled by a spring. The elastic force on the plug is calibrated in order to balance the hydrostatic force exerted by the fluid at the limiting pressure condition. When the pressure exceeds the threshold, the plug opens and the fluid flows in the orifice between the plug and the external wall of the nozzle. This flow is a violent, unsteady liquid flow which can frequently become a flashing flow, where the pressure drop leads to a sudden vaporization of the liquid flow [2]. In any case, the flow dynamics modify the pressure distribution on the plug surface: while the pressure in the inner part of the plug above the nozzle remains approximately constant, the high velocity of the flow in the orifice leads to low pressures on the external portion of the plug surface which can eventually reduce the total hydrodynamic force applied on the plug itself. In this case, the whole the plug dynamics during the opening transient can be modified and a delay in the valve opening can occur.

The behaviour of a safety valve is regulated by specific standards which fix the maximum allowable opening and closing times under given pressure climbs and drops. Numerical flow prediction can be therefore extremely useful in the design process of the valve and in the evaluation of its characteristics. However, the simulation of such a rapid free-front flow with moving boundaries can pose problems to conventional Eulerian description.

Smoothed Particle Hydrodynamics (SPH), owing to its Lagrangian nature and to the automatic treatment of free fronts, appears to be a natural choice for the solution of this kind of problem. Recent applications of the method include the analysis of compressible flows through orifices, where steep pressure gradients occur owing to the strong contraction [3], and of pressure relief valves with moving elastic elements, where the interaction between the liquid flow and an elastic rubber membrane plays an essential role [4].

The present paper deals with the application of SPH to the simulation of the flow during the opening transient of an industrial spring safety valve. A weakly compressible SPH method to simulate the incompressible water flow is adopted and transition to the vapour phase is not considered. The effect of increasing inlet pressures on the plug dynamics is briefly discussed.

II. SPH METHOD

The incompressible flow is here analysed, both as a plane 2D and as an axisymmetric problem, by adopting a Weakly Compressible SPH (WCSPH) formulation, where the incompressible fluid is approximated by a slightly

Sponsors


LMH Laboratory for Hydraulic Machines
http://lmhwww.epfl.ch
EPFL
ÉCOLE POLYTECHNIQUE
FÉDÉRALE DE LAUSANNE
SPHERIC Workshop IIIrd
Ecole Polytechnique Fédérale de Lausanne
spheric
ERCOFTAC
2008, June, (3rd)4th-6th,
VATECH HYDRO
ANDRITZ
CSCS
Swiss National Supercomputing Centre