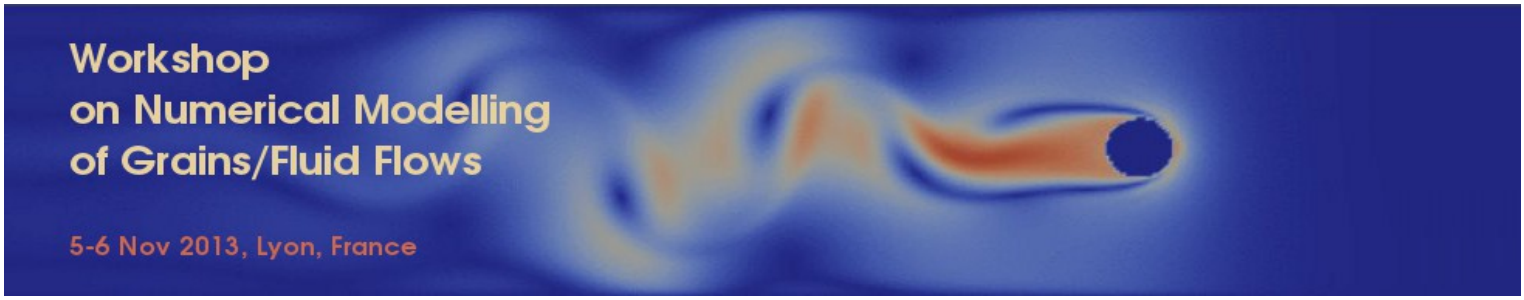


Workshop on Numerical Modelling of Grains/Fluid Flows

November 5-6, 2013
Centre Blaise Pascal, ENS de Lyon, France

November 5			
09:00 – 09:30		Welcome coffee	
09:30 – 10:15	45'	Vincent RICHEFEU	*Simulation of wetting regimes in a 2D granular packing*
10:15 – 10:35	20'	Manuel BERNARD	*DEM-CFD modeling of fluidized beds*
10:35 – 10:55	20'	Talib DBOUK	*A Suspension Balance Model for the flows Of non-Brownian Suspensions of hard spheres*
10:55 – 11:30	35'	Coffee break	
11:30 – 11:50	20'	Mona RAHMANI	*Free falling ad rising of spherical and angular particles*
11:50 – 12:35	45'	Elisabeth GUAZZELLI	*The motion of spherical particles falling in a cellular flow field*

12:35 – 14:00		Lunch	

14:00 – 14:45	45'	Bertand MAURY	*Numerical and modeling issues in fluid-grain simulations*
14:45 – 15:05	20'	Sebastian SCHMIDT	*On continuum dilute and dense granular-air flow modeling With GRAIN - Current state and challenges*
15:05 – 15:25	20'	B. S. Mahmood EBNA HAI	*Numerical Simulation of Fluid Structure Interaction (FSI) on a double wedge Airfoil based on Arbitrary Lagrangian-Eulerian Frameworks*
15:25 – 15:45	20'	Lhassan AMARSID	*Surface waves generation by immersed granular avalanches*
15:45 – 16:20	35'	Coffee break	
16:20 – 17:05	45'	Bruno CHAREYRE	*Pore-scale modelling of grain-fluid mixtures applied to dense suspensions*
17:05 – 17:25	20'	Luc SCHOLTES	*Modeling wave induced pore pressure and effective stress in an inelastic seabed*
17:25 – 17:45	20'	Nagi KHALIL RODRIGUEZ	*Molecular dynamics simulations of driven granular mixtures*
17:45 – 18:05	20'	Patrick MUTABARUKA	*Numerical modeling of the onset of immersed granular avalanches*

19:30 – 22:00		Dinner at Carmelina's (249, rue Marcel Mérieux)	

November 6			
09:00 – 09:45	45'	Cyrus AIDUN	*The mixing and migration of soft and hard particles: Application to cellular blood flow*
09:45 – 10:05	20'	Guilhem AUBERT	*Numerical study of river bedrock incision by bed load sediment transport*
10:05 – 10:25	20'	Philippe CLAUDIN	*Grains in a fluid: sediment transport and rheology*
10:25 – 11:00	35'	Coffee break	
11:00 – 11:45	45'	Yuntian FENG	*The Coupled Lattice Boltzmann and Discrete Element Method and Applications*
11:45 – 12:05	20'	Orencio DURAN VINENT	*From grains to bedforms: direct numerical simulations of aeolian ripples*
12:05 – 12:25	20'	Laurence GIROLAMI	*Sand-layer evolution in a partially filled-pipe*

12:25 – 14:00		Lunch	

14:00 – 14:45	45'	Stephane VINCENT	*A Lagrangian VOF tensorial penalty method for the DNS Of resolved particle-laden flows. Application to fluidized beds*
14:45 – 15:05	20'	Guillaume DUMAZER	*Diphasic granular transport in confined geometries: a numerical challenge?*
15:05 – 15:25	20'	Stany GALLIER	*Simulations of sheared suspensions using a fictitious domain method*
15:25 – 16:00	35'	Coffee break	
16:00 – 16:45	45'	Roberto BENZI	*Lattice Boltzmann Simulation of soft glass*
16:45 – 17:05	20'	Jeff NGOMA	*A 2D DEM-LBM numerical simulation of localized fluidization in an immersed granular medium*

Table of Contents

The mixing and migration of soft and hard particles: application to cellular blood flow

Cyrus K. Aidun
Professor
G.W. Woodruff School of Mechanical Engineering, and
Parker H. Petit Institute for Bioengineering and Bioscience
Georgia Institute of Technology; Atlanta, GA 30332 USA
e-mail: cyrus.aidun@me.gatech.edu
http://www.me.gatech.edu/faculty/aidun

The concentration of platelets in blood is a small fraction of the red blood cells (RBC); therefore, the presence of platelets plays an insignificant role in the rheology of blood flow. However, the transport and radial migration of platelets play a significant role in the initiation and rate of growth in arterial and heart valve thrombosis and thromboembolism. The dynamical coupling between the highly flexible RBCs and rigid platelets plays a significant role in margination – the physics of this coupling is the focus of this presentation. It is shown that to various degree of significance, the margination rate depends on hematocrit, platelet shape, and viscosity ratio of plasma to cytoplasm [1]. Whole blood is modeled as a suspension of deformable red blood cells (RBCs) and rigid platelets in a viscous liquid. The fluid phase is simulated using the lattice-Boltzmann method, the RBC membranes are modeled with a coarse-grained spectrin-link method, and the dynamics of rigid particles are updated using Newton's equations of motion for axisymmetric shapes [2]. The results emphasize that an increase in hematocrit increases the rate of margination. The viscosity ratio between the interior cytoplasm and suspending fluid can considerably alter the rate of margination. The aspect ratio of surrogate platelet particles influences the rate of margination as well. Spherical particles tend to migrate more quickly than disks. Highly viscous or rigid RBCs slow down margination. The coupling and scaling for rate of margination in various size vessels depending on the relevant parameters will be presented along with ideas for global application to complex geometries.

REFERENCES

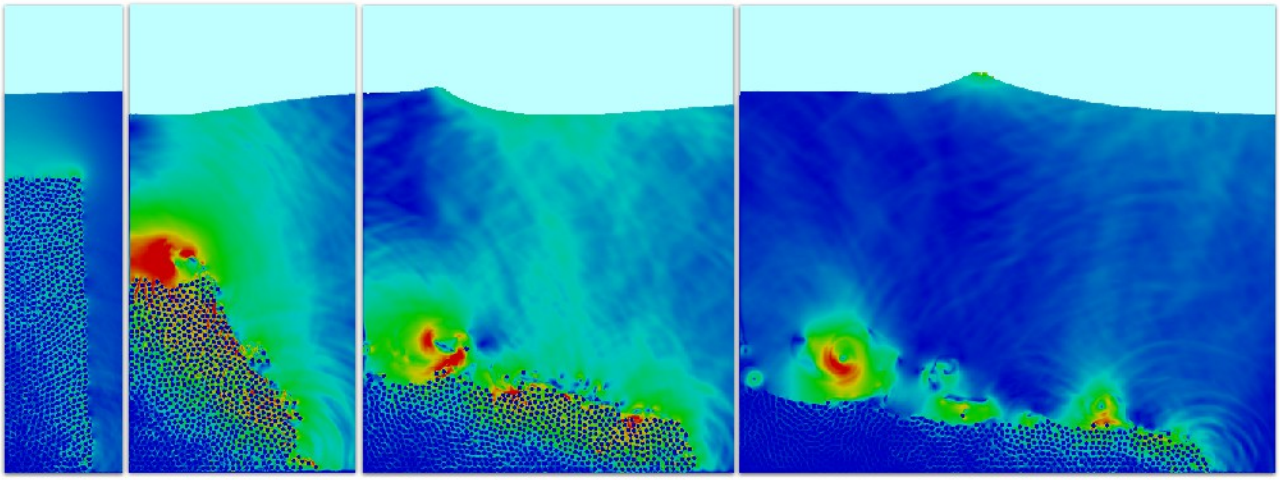
[1] Reasor, D.A., M. Mehrabadi, D.N. Ku, and C.K. Aidun. Determination of critical parameters in platelet margination. *Annals of Biomedical Eng.* doi: 10.1007/s10439-012-0648-7 (2012).

[2] Reasor, D.A., J.R. Clausen, and C.K. Aidun. Coupling the lattice-Boltzmann and spectrin-link methods for the direct numerical simulation of cellular blood flow. *International J. Num. Methods in Fluids*. Vol. 68, Issue 6, pp. 767–781, (2012).

Surface waves generation by immersed granular avalanches

Lhassan Amarsid [1], Carole Delenne [1], Jean-Yves Delenne [1], Vincent Guino [1], Franck Radjai [1]
[1] Laboratoire de mécanique et génie civil (LMGC)

We present a coupled DEM/LBM model for the investigation of surface waves generated by immersed granular avalanches. The evolution of the free surface is modeled by the mass tracking method. We investigate the evolution of surface waves and the influence of the initial configuration (water depth, height of granular column) on their amplitude.



Numerical study of river bedrock incision by bed load sediment transport

Guilhem Aubert
Laboratoire de Géologie de Lyon - Terre, Planètes, Environnement , ENS de Lyon, France

Reptation and saltation has been shown to be the only motion modes of the bed load that contribute to bedrock river incision. Field measurements revealed that the incision rate of a river fluctuates along the year with respect to the water discharge value. Numerical model and experimental results demonstrate that the number of grains per unit surface is a crucial parameter for the incision rate efficiency. We use a numerical model based on molecular dynamics to study the motion of a bed load over a river bedrock and quantify the energy transfer that is responsible of incision. We show that theenergy transfer with respect to the cover fraction present a maximum, what is the result of a competition between two antagonistic effects, the tool effect and the cover effect.

Lattice Boltzmann simulation of soft glass

Roberto BENZI
INFN Roma Tor Vergata, Italy

In this talk I will describe recent advances in the simulation of Soft Glass (emulsion) using Lattice Boltzmann Equation with frustrated interactions. In the first part of the talk, I provide a short review on the numerical approach and its validation. In particular I describe how the frustration interactions can be rephrased in terms of clear physical quantities. In the second part, I discuss the results obtained in the last few years. For suitable initial conditions and parameter choice, it is possible to simulate an emulsion like system with yield stress, non linear rheology and cooperative effects.

DEM-CFD modeling of fluidized beds

Manuel Bernard[1] – Eric Climent[2] – Anthony Wachs[1]

[1] *Fluid Mechanics Department, IFP Energies Nouvelles, 69360 Solaize*
[2] *Institut de Mécanique des Fluides de Toulouse, University of Toulouse - UMR CNRS 5502, 31400 Toulouse*

Fluid-particle flows are frequently encountered in industrial facilities and especially in chemical engineering processes. In this work, we focus on fluidized beds, which involve a fluid flow passing upward through a pack of particles with such a velocity that the fluid force acting on particles is larger than their weight. This technology is commonly used for its high heat and mass transfer rates in Circulating Fluidized Beds or Fluid Catalytic Cracking processes which are composed of millions of particles from a few micrometers to some millimeters, fluidized by a liquid or a gas. In order to optimize performances of those engineering processes, numerical simulations of multiphase flows became indispensable, especially over the last 20 years with the considerable rise of computational power and the progress in multiphase computational fluid dynamics. Depending on the length scale, there are three main approaches to simulate the dense particulate flows in fluidized beds (see fig 1).

At the micro-scale, the fluid motion equations are solved directly on a small mesh compared to the particle diameter. Direct numerical simulation (DNS) provides precise solutions but the number of fluid cells does not permit to simulate systems containing more than a few thousands particles in a reasonable computing time.

At the macro-scale, by using Eulerian methods, the fluid and solid phases are considered as two inter-penetrating media. The mesh is then coarser than a particle diameter and simulation of large domains, up the real size reactor, are doable. However, this length scale requires the introduction of numerous assumptions in the model to describe the evolution of the solid phase and its coupling with the surrounding fluid. Moreover, since the solid phase is considered as a continuous media, particle trajectories are not individually treated, which is a crucial lack of information for engineering processes.

At an intermediate scale between the DNS and Euler-Euler methods, the fluid is solved on a larger grid than the particle diameter, as it is in the Euler-Euler methods, but as in DNS methods, the particle trajectories, including collisions, are tracked with a discrete element method (DEM). This approach, commonly called discrete element method / computationnal fluid dynamics (DEM-CFD) or Euler-Lagrange method, was first introduced by Tsuji *et al.* [Tsuji et al., 1993] and Hoomans *et al.* [Hoomans et al., 1996]. This method has been widely developped and used since the beginning of the XXIst century (see *e.g.* [Kafui et al., 2002; Xu et al., 2000]).

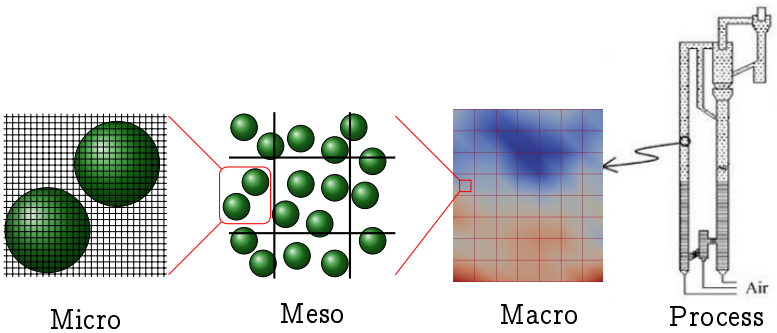


Figure 1: Different length scales used for simulating fluid-particle flows

Pore-scale modelling of grain-fluid mixtures applied to dense suspensions

Bruno Chareyre [1], Donia Marzougui [1], Julien Chauchat [2]
[1] 3SR Grenoble, France
[2] LEGI Grenoble, France

The plane shear flow of a dense fluid-grain mixture is simulated using a new hydromechanical coupled model combining the Discrete Element Method (DEM) for the solid phase with a Pore-scale Finite Volume (PFV) formulation for the fluid phase. This model, based on a regular triangulation of the particles and its dual Voronoi graph, couples the isotropic part of the strain field with the fluid flow [2,3]. In order to study sheared suspensions, we further improved the previous model: including the deviatoric part of the stress tensor on the basis of the lubrication theory[4], and extending the solver to periodic boundary conditions. Simulations of a granular media saturated by an incompressible fluid and submitted to a plane shear at imposed vertical stress are presented. The shear stress is decomposed in different contributions which can be examined separately: contact forces, lubrication forces, and drag forces associated to the poromechanical couplings [2]. The proposed numerical model is able to describe the behavior of the dense suspension. The friction and dilatancy laws predicted by the present model is compared with the experiments and rheological models found in previous works (e.g. Boyer et al. [5]).

REFERENCES

[1] V. Šmilauer and B. Chareyre, Yade Dem Formulation, in Yade Documentation, The Yade project, 1st ed. (2010). http://yade-dem.org/doc/formulation.html

[2] E. Catalano, A pore-scale coupled hydromechanical model for biphasic granular media. PhD thesis. Grenoble Institute of Technology, Grenoble (2012).

[3] B. Chareyre, A. Cortis, E. Catalano,and E. Barthélemy, Pore-scale modeling of viscous flow and induced forces in dense sphere packings. Transport in porous media, 92(2), 473-493 (2012).

[4] B. Van den Brule and R.J.J. Jongschaap, Modeling of concentrated suspensions. Journal of statistical physics, 62(5), 1225–1237 (1991).

[5] F. Boyer, E. Guazzelli and O. Pouliquen, Unifying suspension and granular rheology, Physical Review Letters, 107(18), 188301 (2011).

Grains in a fluid: sediment transport and rheology

O. Durán [1, 2], M. Trulsson [1], B. Andreotti [1], and P. Claudin [1]
[1] Laboratoire de Physique et Mécanique des Milieux Hétérogènes (PMMH),
UMR 7636 CNRS – ESPCI – Univ. Paris Diderot – Univ. P.M. Curie,
10 Rue Vauquelin, 75005 Paris, France.
email: philippe.claudin@espci.fr

[2] Department of Geological Sciences, Univ. North Carolina,
104 South Rd. Mitchell Hall, Campus Box 3315, Chapel Hill,
North Carolina 27515, USA

By means of two phase numerical simulations based on a discrete element method (DEM) for particles coupled to a continuum Reynolds averaged description of hydrodynamics, we have studied the properties of sediment transport by a flow [1] and the rheology of dense suspensions [2]. In the case of transport, we have investigated the transition from bedload to saltation when the particle to fluid density ratio is increased. In both cases, we have identified the mechanisms leading to the scaling laws relating the particle flux to the shear velocity of the flow. As for the suspensions, we have shown how the rheological laws of Bagnold and Newtonian regimes can be unified and expressed as a function of a single dimensionless number, by adding their contribution to the dissipation.

References

[1] O. Durn, B. Andreotti and P. Claudin, *Numerical simulation of turbulent sediment transport, from bed load to saltation*, Phys. Fluids **24**, 103306 (2012).

[2] M. Trulsson, B. Andreotti and P. Claudin, *Transition from viscous to inertial regime in dense suspensions*, Phys. Rev. Lett. **109**, 118305 (2012).

A Suspension Balance Model for the flows of non-Brownian Suspensions of hard spheres

Talib Dbouk [1], Laurent Lobry, Elisabeth Lemaire, and Fadl Moukalled
[1] IRSN - Institut de Radioprotection et de Sûreté Nucléaire, IRSN/PSN-RES/SEMIA/LPTM,
CE Cadarache, 13115 St Paul-Lez-Durance cedex, FRANCE. (talib.dbouk@irsn.fr)

The Suspension Balance Model (Eulerian) (SBM) of Nott and Brady (1994), Morris and Boulay (1999) for the flows of non-Brownian suspensions of hard spheres was implemented in the "OpenFOAM®" open source library.
It captures well the physical features involved in the shear-induced particle migration in simple shear flows. In fact, numerical results obtained with our modified coefficients in Dbouk et al. (2013) suggest that the simplification introduced by Morris and Boulay (1999) in the SBM, is appropriate.

The two-dimensional model was extended into a frame-invariant environment and was extended to include buoyancy effects too (see Dbouk et al. (2012)). The latest solver was implemented also in "OpenFOAM®" and is capable of dealing with general geometries.

Results predicted by the model using "OpenFOAM®" indicated that it can capture well the physical features involved in the shear-induced particle migration even in geometries where the flow of the suspension is general, and where buoyancy is present. This was demonstrated by solving numerically the two-dimensional viscous resuspension and mixing in a horizontal Couette cell problem and comparing simulation results with the measured data of Rhao et al. (2002).

Finally, it can be stated safely that the two-dimensional model described well the physical behaviour of the different suspension flows simulated (in channels & Couette Cells), and could be tested further by simulating additional general flows in more complex geometries.

References:
[Dbouk et al. 2013] T. Dbouk, E. Lemaire, L. Lobry, F. Moukalled, *Shear-induced particle migration: Predictions from experimental evaluation of the particle stress tensor*, Journal of Non-Newtonian Fluid Mechanics, Vol. 198, August 2013, Pages 78-95.http://dx.doi.org/10.1016/j.jnnfm.2013.03.006

[Dbouk et al. 2013] T. Dbouk, L. Lobry, E. Lemaire, *Normal stresses in concentrated non-Brownian suspensions*, J. Fluid Mech. , Vol. 715, pp 239-272, January (2013). http://dx.doi.org/10.1017/jfm.2012.516

[Dbouk et al. 2012] T. Dbouk, *Rheology Of Concentrated Suspensions & Shear-Induced Migration*, Ph.D. thesis, University Of Nice Sophia Antipolis, (2012).

[Rao et al. 2002] R. Rao, L. Mondy, A. Sun, and S. Altobelli, *A numerical and experimental study of batch sedimentation and viscous resuspension*, Int. J. Numer. Meth. Fluids 39, 465-483, (2002).

[Morris and Brady 1998] J. F. Morris and J. F. Brady, *Pressure-driven flow of a suspension: Buoyancy effects*, International Journal of Multiphase Flow, 24, 1, 105-130, (1998).

[Morris and Boulay 1999] J. F. Morris and F. Boulay, *Curvilinear flows of non-colloidal suspensions: the role of normal stresses*, J. Rheol. 43 (5), 1213-1237, (1999).

[Nott and Brady 1994] P. R. Nott and J. F. Brady, *Pressure-driven flow of suspensions: simulation and theory*, J. Fluid Mech. 275, 157-199, (1994).

Diphasic granular transport in confined geometries: a numerical challenge ?

Guillaume Dumazer [1], Eirik Flekkøy [1]
[1] Advanced Material and Complex Systems group, Physics Dept., University of Oslo (AMCS)

Confined mixtures of granular material and fluid exhibits properties little studied until recently. The injection of a third phase into such system leads to a rich interface dynamics. The interaction between the solid granular phase, the immersing fluid and the injected one, in small confinement, involves the competition between capillarity and friction. Stick-slip instability regimes can be obtained.

Beyond its interest for physical sciences such granular transport are challenging the geosciences engineering as well as micro-chemistry. It also stimulates our aesthetical sensitivity since the instability can develop interesting patterns.

An attempt for modelling one-dimensional confined geometries is in development and its confrontation with experiments starts to show preliminary results. The numerical modelling of such system is however at its very beginning. It provides stimulating opportunities and perspectives to explore.

From grains to bedforms: direct numerical simulations of aeolian ripples

Orencio Durán [1], Philippe Claudin [2], and Bruno Andreotti [2]
[1] MARUM – Center for Marine Environmental Research, Bremen University, Germany
[2]Laboratoire de Physique et Mécanique des Milieux Hétéroènes,
PMMH UMR 7636 ESPCI – CNRS – Univ. Paris-Diderot – Univ. P.M. Curie,
10 rue Vauquelin, 75005 Paris, France

In contrast to sand dunes which arise from the coupling of hydrodynamics and topography, aeolian ripples result from the interaction of sediment transport and topography. Current theories of their formation have been unable to explain recent field and laboratory measurements. In order to understand the dynamical mechanisms responsible for this instability, we performed direct numerical simulations of ripples using a two phase model coupling a discrete element method (DEM) for particles to a Reynolds averaged description of hydrodynamics. We find that ripples result from the competition between the resonance of grain trajectories with the modulated surface, which destabilize the granular bed, and gravity-induced downslope transport which stabilize small wavelengths. Crucially, the details of a gas-like collisional boundary layer at the interphase between the transport layer and the static bed select the scaling of ripples wavelengths and migration rates with the wind speed.

The workshop on Numerical Modelling of Grains/Fluid Flows
5–6 November 2013, Lyon, France

Numerical Simulation of Fluid Structure Interaction (FSI) on a double wedge Airfoil based on Arbitrary Lagrangian-Eulerian Frameworks

Bhuiyan Shameem Mahmood Ebna Hai [a,*] and Markus Bause [a]

[a] Department of Mechanical Engineering, Helmut Schmidt University,
University of the Federal Armed Forces Hamburg,
Hostenhofweg 85, 22043 Hamburg, Germany.

*shameem.ebna.hai@hsu-hh.de

Abstract. *Nowadays, advanced composite materials have shown remarkable resilience for lightweight structures, construction in ballistics protection, engineering, and other similar applications. Carbon Fiber Reinforced Plastics (CFRP) are being applied to many aircraft structures in order to improve performance and reduce the weight. But there is a possibility of structural damages due to Fluid-Structure Interaction (FSI) oscillations. Simulation of the FSI, where the dynamics of these currents dominate, poses a formidable challenge for even the most advanced numerical techniques and it is currently at the forefront of an ongoing work in Computational Fluid Dynamics (CFD). Since analytical solutions are only available in special cases, the equation needs to be solved by numerical methods. This paper focuses on the analysis of a non-linear fluid-structure interaction problem and its solution through the Finite Element Method (FEM). Here we briefly describe the analysis of incompressible Navier-Stokes and Elastodynamic equations in the arbitrary Lagrangian-Eulerian (ALE) frameworks in order to numerically simulate the FSI effect on an aircraft wing, which shall describe the underlying physics such as structural vibration. The principal aim of this research is to explore and understand the behaviour of the fluid-structure interaction during the impact of a deformable material (e.g. an aircraft wing) on air. This coupled problem is defined in a monolithic framework and different types time stepping schemes are implemented. Spatial discretization is based on a Galerkin finite element scheme. The non-linear system is solved by a Newton-like method. The implementation using the software library package DOpElib and deal.II serves best for the computation of different FSI configurations.*

Keywords: CFRP, Navier-Stokes, Elastodynamic, Lift, Drag, FSI, ALE, Vibration, FEM, `DOpE lib`, `deal.II`.

1 INTRODUCTION

This paper is intended to provide the motivation that will act as driving force for the execution of this research work. With the scope and objectives together, we will give the outline of this research to help the reader understand the context of the present proposal.

1.1 Motivation

Currently, composite materials have shown remarkable resilience for lightweight structures, construction in ballistics protection, engineering, and other similar applications. Composite materials are formed by combining two or more materials in such a way that the constituents are still distinguishable and not fully blended. The objective is usually to make a component which is strong and stiff, often with a low density. But there is a possibility of structural damage due to Fluid Structure Interaction (FSI). FSI is the interaction of some movable or deformable structure with an internal or surrounding fluid flow, which describes the coupled dynamics of fluid mechanics and structure mechanics. These types of problems are known as Classical Multi-Physics problems. The problems can be stable or oscillatory. Simulation of the FSI, where the dynamics of these currents dominate, poses a formidable challenge for even the most advanced numerical techniques and it is currently at the forefront of an ongoing work in Computational Fluid Dynamics (CFD). In this research work, we will focus on the analysis of the incompressible

Navier-Stokes and elastodynamic equations in the arbitrary Lagrangian-Eulerian (ALE) framework and present a numerical simulation of the FSI effect on an aircraft wing, in which these equations shall describe the underlying physics. For the implementation, we chose `deal.II` [2] based software package `DOpE lib` [3]. While widely used commercial codes, e.g. NASTRAN, FLUENT, ANSYS or COMSOL Multiphysics, can only solve particular problems of aeroelasticity and hydroelasticity and are mainly limited to linearized models, `DOpE lib` is a flexible toolbox providing modularized high-level algorithms that can be used to solve stationary and non-stationary PDE problems as well as optimal control problems constrained by PDEs as well as optimal control problems constrained by PDEs and Dual-Weighted-Resiudal approach for goal-oriented error estimation. However, FSI problems introduce new complications and complexities to be considered, such as coupling techniques, dynamic interaction, different length and time scales of subsystems, therefore making these problems much more difficult than the seperate computation of the fluid and structure. But there is not enough reference papers for study the FSI effect on an aircraft wing. With this in mind, and due to the challenge that it represents, the objective of this work is to contribute to the expansion of knowledge of this specific area.

2 MATHEMATICAL MODELS

2.1 The Fluid-Structure Interaction (FSI) problem

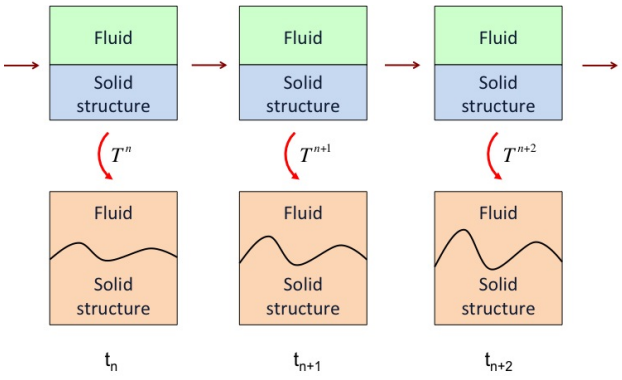


Figure 1: Typical FSI Problem in the ALE framework [7, 8, 9, 10]

Let us assume that $\widehat{\Omega} \subset \mathbb{R}^d, d = 2, 3$, be a bounded domain of the fluid-structure interaction problem in reference configuration at time $t = 0$ with the Lipschitzian boundary [13]. The outer unit normal vector at the boundary is denoted by n. Assume that $\Omega := \Omega(t)$ is split into two time-dependent subdomains $\Omega_f(t)$ (for an incompressible fluid flow) and $\Omega_s(t)$ (for an elastic structure), which is slicked to at the boundary of the domain $\widehat{u}_s = 0$. Here, the boundaries of Ω, Ω_f and Ω_s are denoted by $\partial\Omega$, $\partial\Omega_f$ and $\partial\Omega_s$, respectively. The variational ALE formulation of the fluid part is transformed from its Eulerian description into an arbitrary Lagrangian framework [8, 9, 10] and stated on the (arbitrary) reference domain $\widehat{\Omega}_f$, while the structure part is formulated in Lagrangian coordinates on the domain $\widehat{\Omega}_s$, where $\widehat{\Omega} = \widehat{\Omega}_f \cup \widehat{\Gamma}_i \cup \widehat{\Omega}_s$. Moreover, here we solve the Laplace equation for the definition of the ALE mapping. Here, the continuity of velocity $\widehat{v}_f = \widehat{v}_s$ and $\widehat{u}_f = \widehat{u}_s$ across the common fluid-structure interface on $\widehat{\Gamma}_i = \widehat{\Omega}_f \cup \widehat{\Omega}_s$. We search for $\widehat{u} \in H^1(\widehat{\Omega}_f, \widehat{\Gamma}_D)^d$ and $\widehat{v} \in H^1(\widehat{\Omega}_f, \widehat{\Gamma}_D)^d$, where the local quantities are defined by restrictions: $\widehat{v}_f := \widehat{v}|_{\widehat{\Omega}_f}$, $\widehat{v}_s := \widehat{v}|_{\widehat{\Omega}_s}$ and $\widehat{u}_s := \widehat{u}|_{\widehat{\Omega}_s}$. It is noted that this extension of the pressure is an inconsistency. While the fluid's pressure is of low regularity $\widehat{p}_f \in L^2_0(\widehat{\Omega}_f)$, the Laplace equation yields $\widehat{p}_s \in H^1(\widehat{\Omega}_s)$. This additional regularity will be fed back into the fluid domain if the extension is not properly decoupled. Since the ALE mapping is defined in accordance to the Lagrange-Euler structure mapping via $\widehat{T} := \widehat{x} + \widehat{u}_f$, we can define the following on all Ω: $\widehat{T} := \widehat{x} + \widehat{u}$, $\widehat{F} := I + \widehat{\nabla}\widehat{u}$, and $\widehat{J} := det(\widehat{F})$. In the structure domain, $\widehat{T}$ takes the place of the Lagrangian-Eulerian coordinate transformation, while in the fluid domain, $\widehat{T}$ has no physical meaning but serves as ALE mapping.

Let us consider for a given set X, the Lebesque space $L_X := L^2(X)$ and $L^0_X := L^2(X)/\mathbb{R}$. The functions in L_X with first-order distributional derivatives in L_X make up the Sobolev space $H^1(X)$. Furthermore, we can use

the function spaces $V_X := H^1(X)^d$, $V_X^0 := H_0^1(X)^d$, and for time-dependent functions:

$$\mathcal{L}_X := L^2[0,T;L_X], \qquad \mathcal{V}_X := L^2[0,T;V_X] \cap H^1[0,T;V_X^*],$$
$$\mathcal{L}_X^0 := L^2[0,T;L_X^0], \qquad \mathcal{V}_X^0 := L^2[0,T;V_X^0] \cap H^1[0,T;V_X^*],$$

The fluid-structure interaction problem in ALE framework

Find $\widehat{v} \in \widehat{v}^D + \widehat{\mathcal{V}}_{\widehat{\Omega}}^0$, $\widehat{u} \in \widehat{u}^D + \widehat{\mathcal{V}}_{\widehat{\Omega}}^0$ and $\widehat{p} \in \widehat{\mathcal{L}}_{\widehat{\Omega}}$, such that $\widehat{u}(0) = \widehat{u}^0$ and $\widehat{v}(0) = \widehat{v}^0$, for almost all time steps $t \in I$ holds:

$$\begin{aligned}
&\left(\widehat{J}_f\widehat{\rho}_f\partial_t\widehat{v}_f, \widehat{\phi}^v\right)_{\widehat{\Omega}_f} + \left(\widehat{J}_f\widehat{\rho}_f(\widehat{F}_f^{-1}(\widehat{v}_f - \partial_t\widehat{T}_f).\widehat{\nabla})\widehat{v}_f), \widehat{\phi}^v\right)_{\widehat{\Omega}_f} \\
&\quad + \left(\widehat{J}\widehat{\sigma}_f\widehat{F}^{-T}, \widehat{\nabla}\widehat{\phi}^v\right)_{\widehat{\Omega}_f} - \langle\widehat{h}, \widehat{\phi}^v\rangle_{\widehat{\Gamma}_N} - \left(\widehat{J}\widehat{\rho}_f\widehat{f}_f, \widehat{\phi}^v\right)_{\widehat{\Omega}_f} \\
&\quad + \left(\widehat{\rho}_s\partial_t\widehat{v}, \widehat{\phi}^v\right)_{\widehat{\Omega}_s} - \left(\widehat{\rho}_s\widehat{f}_s, \widehat{\phi}^v\right)_{\widehat{\Omega}_s} - \left(\widehat{J}\widehat{\sigma}_s\widehat{F}^{-T}, \widehat{\nabla}\widehat{\phi}^v\right)_{\widehat{\Omega}_s} = 0 \qquad \forall\widehat{\phi}^v \in \widehat{V}_{\widehat{\Omega}}^0 \\
&\qquad \left(\widehat{div}(\widehat{J}\widehat{F}^{-1}\widehat{v}), \widehat{\phi}^p\right)_{\widehat{\Omega}_f} + \left(p_s, \widehat{\phi}^p\right)_{\widehat{\Omega}_s} = 0 \qquad \forall\widehat{\phi}^p \in \widehat{L}_{\widehat{\Omega}} \\
&\left(\partial_t\widehat{u} - v, \widehat{\phi}^u\right)_{\widehat{\Omega}_s} + \left(\widehat{\sigma}_g, \widehat{\nabla}\widehat{\phi}^u\right)_{\widehat{\Omega}_f} - \langle\widehat{\sigma}_g, \widehat{n}^f, \widehat{\phi}^u\rangle_{\widehat{\Gamma}_i} = 0 \qquad \forall\widehat{\phi}^u \in \widehat{V}_{\widehat{\Omega}}^0
\end{aligned} \tag{1}$$

The stress tensors for the fluid and structure are implemented in $\widehat{\sigma}_f$, $\widehat{\sigma}_s$, and $\widehat{\sigma}_g$, where the stress tensors are given by $\widehat{\sigma}_f(\widehat{x}) = -p_f I + \widehat{\rho}_f\nu_f\left(\widehat{\nabla}\widehat{u}_f\widehat{F}_f^{-1} + \widehat{F}_f^{-T}\widehat{\nabla}\widehat{u}_f^T\right)$, and $\widehat{\sigma}_s = \widehat{J}^{-1}\widehat{F}\left(2\mu_s\widehat{E} + \lambda_s tr(\widehat{E})I\right)\widehat{F}^T$. In this formulation, for momentum equations, integration by parts in both subdomains yields the boundary term on $\widehat{\Gamma}_i$ as: $\left(\widehat{n}_f.(\widehat{J}\widehat{\sigma}_s\widehat{F}^{-T}), \widehat{\phi}^v\right)_{\widehat{\Gamma}_i} + \left(\widehat{n}_s.(\widehat{J}\widehat{\sigma}_f\widehat{F}^{-T}), \widehat{\phi}^v\right)_{\widehat{\Gamma}_i} = 0$. We refer to [5, 7, 8, 9, 10] for more details about the functional spaces and FSI formulation in ALE formulation.

3 NUMERICAL EXAMPLE

The aim of this research is to explore and understand the behaviour of engineering artefacts in extreme environments. To achieve the main ambition of this work, we split this research into two parts. The first part will consider to determine the effect of fluid flow over a sample airfoil (2D) and study the displacement of a control point $A(t)$ under incompressible fluid flow. The second part of this research focused on FSI effect on 3D aircraft wing to identify the list of critical design points to implementing a Damage Identification Strategy (DIS) [14], where we will design an integrated SHM system for an aircraft. But numerical simulations FSI on a sample aircraft wing (3D) still in progress.

3.1 Configuration test model

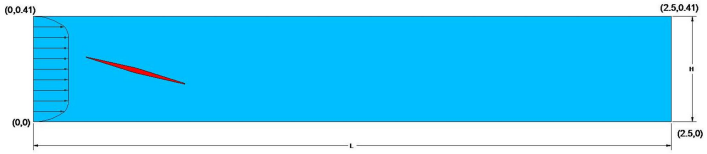


Figure 2: Computational domain

The computational domain is designed based on the $2D$ FSI benchmark as shown in Figure-2 and it is determined by following characteristics:

- The computational domain has the length $L = 2.5$ and height $H = 0.41$.
- We will examine a double wedge airfoil as our test model. The chord length of the airfoil $c = 0.41$ and maximum thickness $t = 0.07$ with a 15 degree angle of attack (AOA).
- Left end and right lower end of the airfoil is positioned at $(0.2, 0.253)$ and $(0.6, 0.147)$, respectively.
- The control points $A(t)$ are fixed at the trailing edge of the structure with $A(t)|_{t=0} = (0.6, 0.147)$, measuring x and y- deflections of the airfoil.

3.2 Material properties

This work is concerned with numerical approximation of FSI effect on a St. Venant-Kirchhoff (STVK) compressible elastic material model. This model is suitable for large displacements with moderate strains.

The elasticity of material structures is characterized by the Poisson ratio ν_s and the Young modulus E_{Y_s}. The relationship of two material parameters μ_s and λ_s is given by:

$$\nu_s = \frac{\lambda_s}{2(\lambda_s + \mu_s)}, \qquad E_{Y_s} = \mu_s \frac{3\lambda_s + 2\mu_s}{\lambda_s + \mu_s},$$
$$\mu_s = \frac{E_{Y_s}}{2(1+\nu_s)}, \qquad \lambda_s = \frac{\nu_s E_{Y_s}}{(1+\nu_s)(1-2\nu_s)},$$

where for compressible material $\nu_s < \frac{1}{2}$ and incompressible material $\nu_s = \frac{1}{2}$. And the fluid is assumed to be incompressible and Newtonian.

3.3 Boundary Conditions

The boundary conditions are as follows:

- A constant parabolic inflow profile is prescribed at the left inlet as

$$v_f(0, y) = 1.5U_m \frac{4y(H-y)}{H^2}, \tag{2}$$

  where U_m is the mean inflow velocity and the maximum inflow velocity in $1.5U_m$

- At outlet, zero-stress $\sigma.n = 0$ is realized by using the 'do-nothing' approach in the variational formulation.
- Along the upper and lower boundary, the usual 'no-slip' condition is used for the velocity.
- Left end of airfoil is considered rigid.

3.4 Initial Conditions

The initial conditions are as follows:

$$v_f(t; 0, y) = \begin{cases} v_f(0, y)\frac{1-cos\left(\frac{\pi}{2}t\right)}{2}, & t < 2.0 \\ v_f(0, y), & t \geq 2.0 \end{cases} \tag{3}$$

4 NUMERICAL RESULTS

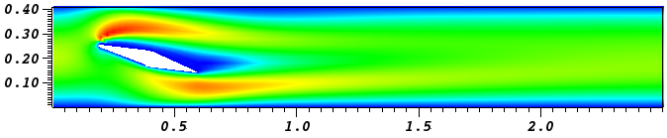
In this research, we introduce three FSI test cases that are treated with different inflow velocities (see Table-1) [8,9,10]. The parameters are chosen such that a visible transient behavior of the double wedge airfoil can be seen. To ensure a 'fair' comparison of results, we calculate the comparison values using the ALE method. For all cases, a uniform time-step size of $k = 0.0167s$ is used. But to ensure the convergence of numerical simulations, different time-step sizes and schemes are used and same result obtained.

Table 1: Parameter setting for the FSI test cases

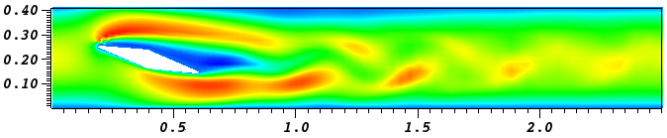
Parameter	Test-1	Test-2	Test-3
Structure model	STVK	STVK	STVK
$\rho_f[kgm^{-3}]$	1000	1000	1000
$\rho_s[kgm^{-3}]$	2710	2710	2710
$\nu_f[m^{-2}s{-}1]$	1×10^{-3}	1×10^{-3}	1×10^{-3}
ν_s	0.33	0.33	0.33
$\mu_s[kgm^{-1}s^{-2}]$	68.9×10^6	68.9×10^6	68.9×10^6
$U_m[ms^{-1}]$	0.5	1.0	2.0

Table 2: Results for the test case 1, 2, and 3

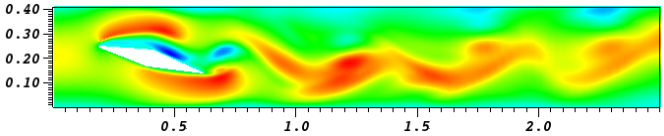
	Test-1	Test-2	Test-3
DoF	83767808	83767808	331805056
$k[s]$	0.0167	0.0167	0.0167
$u_x(A)[\times 10^{-5}]$	0.1604	0.5078±0.061	2.675±4.973
$u_y(A)[\times 10^{-5}]$	0.5627	1.747±0.241	9.242±19.219
F_D	15.693	61.025±0.963	292.733±84.496
F_L	24.988	69.111±5.607	285.01±456.08
$\Delta P[\times 10^3]$	0.138	0.827±0.0295	6.306±2.462



(a) Test-1



(b) Test-2



(c) Test-3

Figure 3: Velocity field

The computed values of the FSI test-1,2 and 3 are summarized in the Table-2 and the velocity fields are displayed in Figure-3. We begin with the FSI-1 test case. The time-dependent behavior of the deflections, Drag, Lift and pressure difference in between left end and right lower end of aerofoil became steady after $3.6s$. Displacement in the x and y direction became steady at 0.1604×10^{-5} and 0.5627×10^{-5}, respectively. We monitored a steady pressure difference (144.751), as well as the lift and drage force 24.988 and 15.693, respectively.

In FSI test case 2 and 3, the time-dependent behavior of the displacement, pressure difference, lift and drag force are not steady. We monitored an oscillation with a range of amplitude. In a simple sense we can say that vibration is an oscillatory motion of a mechanical dynamic system or structure around same reference state, which is often the state of static equilibrium. In fact vibrations often are undesirable in mechanical structures as they cause fatigue failure and lead to increase of stress and bearing loads etc. For example, if an aircraft wing vibrates excessively, especially with the frequencies in the range of the natural frequencies (approx. $4 - 8\ Hz$) of the human body and organs, passengers inside the aircraft will feel uncomfortable and it can cause serious internal trauma (Leatherwood and Dempsey, 1976 NASA TN D-8188). But if aircraft wings vibrate with large amplitudes for an extended period of time, there will be fatigue failure in wings, which would potentially cause the aircraft to crash with massive fatalities. The Tacoma Narrows Bridge disaster in 1940 was one of the most famous engineering disasters of all time, and it failed due to the same type of self-excited vibration behavior that occurs in aircraft wings.

5 CONCLUSIONS

This project is focused on computing prototype configurations to test the code, which will help us to use the software to computing any realistic applications. The numerical simulations FSI on a sample aircraft wing (3D) still under test. This paper mainly deal with a double wedge airfoil ($2D$), where the left end of this airfoil is considered rigid and the control points $A(t)$ are fixed at the trailing edge with $A(t)|_{t=0} = (0.6, 0.147)$. We observed different behaviors of the double wedge airfoil with different FSI test cases. In test cases FSI-1, the deflection of trailing edge of the airfoil became steady, while in FSI test-2 and 3, we observed oscillating behavior. Therefore, we conclude that the deflective behavior of structures becomes unsteady under high-speed fluid flow, which can be cause of massive fatalities.

ACKNOWLEDGMENT

Thanks go to W. Wollner, T. Wick and C. Goll for developing the wonderful software packages `DOpE lib`.

REFERENCES

[1] J. Struckmeier, *"Advanced Fluid Dynamics"*, University of Hamburg, 2011.
[2] W. Bangerth, T. Heister, G. Kanschat, *`deal.II`: Differential Equations Analysis Library*, web: www.dealii.org.
[3] C. Goll, T. Wick, W. Wollner, *`DOpElib`: The `deal.II` based optimization library*, web: www.dopelib.uni-hamburg.de.
[4] T. Richter, *"A Fully Eulerian Formulation for Fluid-Structure-Interaction Problems"* , Institute for Applied Mathematics, University of Heidelberg, Germany, 2012.
[5] T. Wick, *"Adaptive Finite Elements for Monolithic Fluid-Structure Interaction on a Prolongated Domain: Applied to an Heart Valve Simulation' '*, Computer Methods in Mechanics, Warsaw, Poland, 2011.
[6] T. Dunne, *"Adaptive Finite Element Approximation of Fluid-Structure Interaction Based on Eulerian and Arbitrary Lagrangian-Eulerian Variational Formulations"*, Institute for Applied Mathematics, University of Heidelberg, Germany, 2007.
[7] T. Dunne, *"Adaptive Finite Element Simulation of Fluid Structure Interaction Based on Monolithic Variational Formulations"*, Institute for Applied Mathematics, University of Heidelberg, Germany, 2009.
[8] H. J. Bungartz, M. Schaefer (Eds.). *"Fluid-Structure Interaction"*. Vol. 53. Springer, 2006.
[9] Bungartz, H. J., Miriam M., Schaefer, M. (Eds.). *"Fluid-Structure Interaction-II"*. Vol. 73, 1st Ed. Springer, 2010.
[10] G. P. Galdi, R. Rannacher, R. (Eds.). *"Fundamental Trends in Fluid-Structure Interaction"*. Vol. 1. World Scientific Publishing Co. Pte. Ltd. 2010.
[11] W. Wollner, *"Optimization of Complex System"*, University of Hamburg, 2010-2011.
[12] M. Hinze, *"Numerical Methods for PDEs Finite Elements"*, Department of Mathematics, University of Hamburg, 2011.
[13] T. Richter, *"Numerical Methods for Fluid-Structure Interaction Problems"*, Institute for Applied Mathematics, University of Heidelberg, Germany, 2010.
[14] B.S.M. Ebna Hai, M. Bause. *"Adaptive Multigrid Methods for Fluid-Structure Interaction (FSI) Optimization in an Aircraft and design of integrated Structural Health Monitoring (SHM) Systems"*. 2nd ECCOMAS Young Investigators Conference (YIC2013), Bordeaux, France, 2013.

The Coupled Lattice Boltzmann and Discrete Element Method and Applications

Yuntian FENG
Swansea University, UK

This talk will present the development of advanced numerical techniques essential for coupling the Lattice Boltzmann Method (LBM) and the discrete element method (DEM) to simulate particle transport problems in turbulent fluid flows encountered in many engineering applications. Key computational issues involved are: 1) the standard LB formulation for the solution of incompressible fluid flows; 2) the incorporation of large eddy simulation (LES) based turbulence models in the LB equations for turbulent flows; 3) the computation of hydrodynamic interaction forces of the fluid and moving particles; and 4) the DE modeling of the interaction between solid particles. A number of applications will be presented to demonstrate the capability of the coupling strategy.

Simulations of sheared suspensions using a fictitious domain method

Stany Gallier [1] [2], Elisabeth Lemaire [2], Laurent Lobry [2], François Peters [2]
[1] Herakles
[2] Laboratoire Physique de la Matière Condensée, Université Nice Sophia Antipolis, France

In this work, we have developed a fully Eulerian, Lagrange multiplier-free, fictitious domain approach to compute low Reynolds suspensions. Since lubrication forces between particles play an important role in the suspension rheology, they must be accounted properly in the model. The lubrication model consists in transposing the classical approach used in Stokesian Dynamics to our present direct numerical simulation. This lubrication model has also been adapted to account for solid boundaries such as walls. Finally, contact forces between particles are modeled using a classical Discrete Element Method (DEM) which is widely used in granular matter physics. The implementation of DEM in the fluid solver is such that it is tightly coupled with short-range lubrication forces since dense suspensions are predominantly controlled by both lubrication and contact forces (e.g., through particle roughness).
After some selected validations, we present some numerical results on the rheology of dense bounded sheared non-Brownian suspensions, including particle roughness and friction. Preliminary results show that both confinement and friction have a marked impact on the suspension rheology, especially on normal stresses, which could partially help understand discrepancies noted experimentally on those normal stresses.

Sand-layer evolution in a partially filled-pipe

L. Girolami [1] & F. Charru [1]
[1] IMFT, 2 Allée du Professeur Camille Soula, 31400 Toulouse, France

In petrolium engineering, the extraction of oil in ducts is commonly associated with the presence of solid particles, coming directly from the source and/or from the abrasion of the pipe during propagation. The extraction procedure can require the initiation and stop of the flow which entails the sedimentation of particles such as a roughly flat sediment layer accumulates progressively at the base. This bed can develop different patterns whose morphology is principally governed by the bottom shear-stress exerted by the overlying fluid. The amplitude of these perturbations can disturb the flow (by increasing the pressure drop) and lead ultimately to the flow blockage. Therefore, petroleum companies require predictions of the temporal evolution of the fluid-solid interface in order to improve the sand management for operational decisions.

The physical model developed in this study presents a 1D model for the sand transport in pipes under laminar and turbulent regimes in which equations are integrated over the wet section. The novelty here is to propose consistent Saint-Venant equations able to describe non-equilibrium situations. Following this target, the fluid phase modeling is represented by an elongated flow (i.e. characterized by a small aspect ratio) and confined between a slowly varying bottom (associated with long-wave perturbations) and an upper rigid wall. The flow is considered as fully developed, such as the vertical momentum diffusion is made instantaneously at each location. Following these assumptions, we used the boundary-layer equations of Prandl, assuming a no slip-condition at the two extremities. Theoretical predictions are thus compared with direct numerical simulations and highlight a very statifying agreement between the theoretical predictions of the bottom shear-stress, which govern the net deposition rate, and numerical simulations. On the other side, the modeling of the solid-phase considers non-equilibrium sand-transport laws, including relaxation effects.

The motion of spherical particles falling in a cellular flow field

Gilles Bouchet [1], Laurence Bergougnoux [1], and Elisabeth Guazzelli [1]
[1] Aix Marseille University, CNRS, IUSTI UMR 7343, 13453 Marseille cedex 13

The objective of the present study is to understand the influence of turbulence on the settling of particles under the action of gravity. This effect is intimately related to the interactions of particles with local spatial structures of the flow, e.g. large vortices. These vortical structures have a significant effect on the local particle transport and concentration. We present a jointed experimental and numerical study to examine these issues. The two-dimensional model experiment uses electroconvection to generate a two-dimensional arrays of controlled vortices which mimic a simplified turbulent flow. Particle image-velocimetry or tracking are used to examine the motion of the particles within this vortical flow. The numerical simulation is inspired by the model developed by Maxey (Phys. Fluids 30, 1915, 1987).

This work was undertaken under the auspices of ANR-12-BS09-0017-01, CNRS-PICS05848, ANR-11-LABX-0092, and ANR-11-IDEX-0001-02.

Molecular dynamics simulations of driven granular mixtures

Nagi Khalil*and Vicente Garzó†

Departamento de Física, Universidad de Extremadura, E-06071 Badajoz, Spain

The Boltzmann kinetic theory is used as a starting point to model a suspension of bidisperse solid particles immersed in a viscous fluid. The grains are modeled as hard disks or spheres that undergo inelastic collisions characterized by velocity-independent coefficients of normal restitution. As a first approximation, the effect of the fluid on the grains dynamics is accounted for two terms: (i) a drag force, proportional to the velocity of particles (Stokes' law), that attempts to model the friction of the grains on the interstitial fluid and (ii) a stochastic force, where the particles are randomly kicked between collisions. The last contribution tries to simulate the kinetic energy gain due to eventual collisions with the (more rapid) particles of the surrounding fluid. Molecular dynamics simulations of the model are based on a event-driven algorithm where the two additional forces are incorporated, after a proper identification of the characteristic time scales of the system. Some numerical results are compared with the theoretical predictions [1, 2] obtained by solving the Boltzmann equation from the Chapman-Enskog method. Here, we focus on steady homogeneous states as they are fundamental for the construction of a hydrodynamic description of the system, see [1, 2].

References

[1] V. Garzó, M. G. Chamorro, and F. Vega Reyes. Phys. Rev . E **87**, 032201 (2013)

[2] Nagi Khalil and Vicente Garzó. ArXiv:1309.0464

*nagi@us.es

†vicenteg@unex.es

Numerical and modeling issues in fluid-grain simulations

Bertrand Maury
Laboratoire de mathématiques, Université Paris-Sud, France

The direct simulation of immersed suspensions poses many difficulties. We shall mainly focus on two of them:

1) The domain occupied by the fluid is highly complicated and singular in quasi-contact situations, which makes it difficult (yet, not impossible) to generate meshes that are respectful of the geometry. Most methods therefore rely on a cartesian mesh that covers the whole mixture. Since this mesh does not fit to the boundary of particles, it degrades the quality of the velocity fields description. And this happens at the very location where dynamic interactions between the two phases take place. We shall present some attemps that have been made to overcome this difficulty, and propose a method based on a smooth extension of the velocity field within the particle, to recover good approximation properties in spite of the non boundary fitted mesh.

2) When grains get close to each other, the interstitial flow is poorly described by most direct methods (especially those that are based on a fixed mesh). Yet, the interaction between the two grains is driven by lubrication forces, and those forces can be shown to play, in some regimes, a very significant role in the overall behavior of the suspension.
We shall present a method to account for those lubrication forces, based on an asymptotic expansion in the interparticle distance: the "gluey contact" model.

This work has been done in collaboration with B. Fabrèges, L. Gouarin (fluid-grain solver), S. Faure and A. Lefebvre-Lepot (gluey contact).

Numerical modeling of the onset of immersed granular avalanches

P. Mutabaruka [a], J.-Y. Delenne [b], F. Radjaï [a] and K. Soga [c]

[a] LMGC, UMR 5508 University Montpellier 2 – CNRS,
F-34095 Montpellier cedex 5, France

[b] IATE, UMR1208 INRA–University Montpellier 2–Cirad–Montpellier Sup Agro,
F-34060 Montpellier cedex 1, France

[c] Cambridge University, Engineering Department,
Cambridge CB2 1PZ, UK

A dense granular bed immersed in a viscous fluid and inclined above its angle of repose is stabilized by a tensile overpressure induced by the slow creep and expansion of the bed due to dialatncy. We analyze this transient creep flow and onset of slope failure as a function of the initial packing fraction and for different slope angles by means of 3D coupled molecular dynamics/Lattice Boltzmann simulations. Our data are in excellent agreement with the experimental results reported by Pailha et al. [1]. We find a parabolic increase of the overpressure with depth, as correctly predicted from the Darcy law and dilation rate, and show that the failure is triggered at the free surface and propagates consequently to the bottom of the bed. The mean triggering time scales with a characteristic time depending on the fluid viscosity and slope angle, and it increases nonlinearly with the initial packing fraction. We also derive the time evolution of the packing fraction that nicely fits the simulation data. Interestingly, the packing fraction at failure decreases only slightly as the initial packing fraction is reduced but remains always above the critical packing fraction, which is reached well after the failure during steady flow on the inclined plane. As in experiments, the cumulative shear strain at failure varies quite weakly with the initial packing fraction. A detailed analysis of the evolution of the granular texture during creep suggests that the pile expands for a nearly constant number of contacts. This leads to the rotation of the mean contact direction, accompanied by an increase of contact anisotropy and a decrease of force anisotropy. The rotation drives the system, increasingly fragilized by the increase of porosity, from a Mohr-Coulomb behavior to a fluid-like state. The difference between the principal stress directions with respect to the flow direction explains correctly the fact that the instability occurs for a nearly constant value of the cumulative shear strain with a weak dependence on the initial packing fraction and slope angle.

[1] M. Pailha, M. Nicolas, and O. Pouliquen. Initiation of underwater granular avalanches : Influence of the initial volume fraction. Physics of Fluids, 20(11) :111701, 2008.

A 2D DEM-LBM numerical simulation of localized fluidization in an immersed granular media

Jeff Ngoma [1], Jean-Yves Delenne [1], Farhang Radjai [1], Pierre Philippe [2], Stéphane Bonelli [2]
[1] Laboratoire de mécanique et génie civil de Montpellier, France
[2] Institut national de recherche en sciences et technologies pour l'environnement et l'agriculture

In this work, a two-dimensional simulation has been developed using coupled Discrete Element and Lattice Boltzmann Methods (DEM-LBM) to simulate the fluid- particlesinteraction in a granular medium in order to investigate the frontier between motionless and fluidizedparticles, subjected to fluid flow. Specific aspects of the coupled system are developed taking into account the interaction of the two phases.
The LBM simulates fluid flows within pores spaces while the solid grains are modeled using the DEM. This paper reports numerical results of fluidized zone development in an immersed granular media under the effect of a locally injected upward fluid flow: transient and stationary regimes, thresholds, influence of injection diameter of fluid flow, hysteresis effect of the fluidized cavity regime, and interaction between two separate injections. These numerical results are compared with previous experimental data by P. Philippe and M. Badiane [Physical Review E 87, 042208, 2013].

Free falling ad rising of spherical and angular particles

Mona Rahmani [1,] Anthony Wachs [1]
[1] IFP Energies nouvelles, Establissement de Lyon, 69360, Solaize, France

Direct numerical simulations of freely falling and rising particles in an infinitely long domain, with periodic lateral boundary conditions, are performed. The focus is on characterizing the free motion of cubical and tetrahedral particles for different Reynolds numbers, as an extension to the well-studied behaviour of freely falling and rising spherical bodies. The vortical structure of the wake, dynamics of particle movement, and the interaction of the particle with its wake are studied. The results reveal mechanisms of path instabilities for angular particles that are different from those for spherical ones. The rotation of the particle plays a more significant role in the transition to chaos for angular particles. The balance of forces and torques acting on particles is discussed to gain more insight into path instabilities of angular particles.

Simulation of wetting regimes in a 2D granular packing

Vincent Richefeu, Jean-Yves Delenne, Farhang Radjai

In wet granular materials, the liquid – in the form of clusters that bind the grains together or in the form of thin adsorbed phase – plays a key role in the rheological properties. Manufacture of pharmaceutical pills, shear strength of wet and coarse soils, triggering of landslides or pollution transport in zones above water tables are some examples.

The talk is concerned with a numerical analysis of liquid distribution within a 2D granular packing. The thermodynamics of phase change is based on Carnahan-Starling's equation-of-state from which the interactions between liquid, gas and solid (grains) are derived using nonlocal potentials in the framework of Multiphase Lattice Boltzmann method. These potentials are calculated on a regular mesh between the fluid particles and neighboring lattice nodes that control the surface tension and the contact angle between fluid and solid.

Injecting, slowly and homogeneously, vapor that condense in-between the grains and in the liquid phase, increases the saturation degree. A flood-fill algorithm is used to identify the liquid clusters and to determine their volume and connectivity with grains. The latter feature provides rich information that is analyzed from the point of view of the liquid phase and also of the grains. The pressures of the clusters are analyzed as a function of liquid content. This gives access to the global liquid-retention curve as well as the forces acting on each grain. By integrating these forces, we compute the negative pressure in the sample due to capillary forces and hence the capillary cohesion of the material. The plot of cohesive strength as a function of saturation degree reveals four different states reflecting the connectivity of the liquid phase and local grain environments. We find that the liquid phase undergoes a percolation transition for a liquid content well below full saturation. Interestingly, the cohesive strength has its peak value below this transition, dividing thus the funicular regime into an ascending cohesion regime followed by a descending cohesion in the late funicular regime.

On continuum dilute and dense granular-air flow modeling with GRAIN - Current state and challenges

Sebastian Schmidt [1], Dariusz Niedziela [2], Konrad Steiner [2]
[1] ETH Zürich, Switzerland
[2] Fraunhofer-Institut für Techno- und Wirtschaftsmathematik ITWM

We present a hydrodynamic granular flow modeling approach for both dilute and densely packed regimes. A continuous transition between the dilute phase, based on granular gas kinetics and the dense phase based on a granular-temperature-independent pressure is assured. This initially single-phase modeling approach is extended to a two- or multiphase method via friction-based coupling to a compressible air phase and incompressible fluid phases. The potential of the method is exemplified via a three phase suspension flow granular bead mill scenario.
Furthermore we discuss the limits of the continuum modeling approach regarding the parameter setup for actual granular materials from powders to granulates. The continuum and hence macroscopic approach requires macroscopic, material-dependent constituting relations, from volume-fraction dependent pressure, through granular viscosity to fluid permeability of the granular bed, which are non-trivial to obtain. Here we present our view on future extensions of the method, both experimentally as well as through upscaling ideas.

Modeling wave induced pore pressure and effective stress in an inelastic seabed

Luc Scholtès [1], Bruno Chareyre [2] and Hervé Michallet [3]
[1] CNRS/Université de Lorraine, UMR GeoRessources, 54518 Vandoeuvre-lès-Nancy cedex
[2] CNRS/UJF/G-INP, UMR 3S-R, 38041 Grenoble cedex 9
[3] CNRS/UJF/G-INP, UMR LEGI, 38041 Grenoble cedex 9

The response of a sandy seabed under wave loading is investigated on the basis of numerical modeling using a multi-scale approach. To that aim, the discrete element method is coupled to a finite volume method specially enhanced to describe compressible fluid flow. Both solid and fluid phase mechanics are upscaled from considerations established at the pore level. Model's predictions are validated against poroelasticity theory and discussed in comparison with experiments where a sediment analog is submitted to wave action in a flume. Special emphasis is put on the mechanisms leading the seabed to liquefy under wave induced pressure variation on its surface. Liquefaction is observed in both dilative and compactive regimes. It is shown that the instability can be triggered for a well identified range of hydraulic conditions. Particularly, the results confirm that the gas content, together with the permeability of the medium are key parameters affecting the transmission of pressure inside the soil.

A Lagrangian VOF tensorial penalty method for the DNS of resolved particle-laden flows. Application to fluidized beds

Stéphane VINCENT
Ecole Nationale Supérieure de Chimie, de Biologie et de Physique, Bordeaux, France

The direct numerical simulation of particle flows is investigated by a Lagrangian VOF approach and penalty methods of second order convergence in space for incompressible flows interacting with resolved particles on a fixed structured grid. A specific Eulerian volume of fluid method is developed with a Lagrangian tracking of the phase function while the solid and divergence free constraints are ensured implicitly in the motion equations thanks to fictitious domains formulations, adaptive augmented Lagrangian approaches and viscous penalty methods. A specific strategy for handling particle collisions and lubrication effects is also presented. Various dilute particle laden flows are considered for validating the models and numerical methods. Convergence studies are proposed for estimating the time and space convergence orders of the global DNS approach. Finally, two dense particle laden flows are simulated, namely the flow a cross a fixed array of cylinders and the fluidization of 2133 particles in a vertical pipe.The numerical solutions are compared to existing theoretical and experimental results with success.

Authors Index

Edited by CCSd (Centre pour la Communication Scientifique Directe) on Tue, 29 Oct 2013