November 5-6 2015

Workshop on

Low Velocity Flows

Application to Low Mach and Low Froude regimes

MAP5, Université Paris Descartes

BOOK OF ABSTRACTS

This workshop is organised by the GdT CDMATH http://cdmath.jimdo.com/

We would like to gratefully acknowledge partial support from the following sponsors:

Financial supports

Partners

Secondary supports

Last update of this online version: October 29, 2015. Possible changes will also be reported on the website of the workshop at the address:

https://indico.math.cnrs.fr/event/LowMach/

Welcome from the Organising Committee

Welcome to Paris and to this workshop entitled “Low Velocity Flows – Application to Low Mach and Low Froude regimes”.

This 2-day workshop is dedicated to the modelling of fluid flows in specific regimes for which theoretical and numerical issues are numerous. It is aimed at gathering young and expert international researchers in order to present a state of the art of the low Mach/Froude problems and new developments upon this topic. About 100 people from all around Europe decided to join us. We especially thank all the speakers for accepting to share their knowledge and expertise.

A poster session is scheduled on Friday morning. Posters will remain displayed throughout the workshop.

A volume of proceedings of the workshop will be published in “ESAIM Proceedings and Surveys”. All contributors (lectures and posters) were proposed to submit a paper.

Finally, we wish to thank the whole administrative MAP5 laboratory staff for their participation in the organisation of this workshop.

The organising committee:

- Stéphane Dellacherie (CEA Paris-Saclay et École Polytechnique de Montréal)
- Gloria Faccanoni (IMATH, Université de Toulon)
- Bérénice Grec (MAP5, Université Paris Descartes)
- Frédéric Lagoutiere (Université Paris-Sud)
- Yohan Penel (CEREMA, Inria, UPMC, CNRS)

Practical Informations

Internet Connexion:

Login: LOWMACH15

Password: CDMATH

Address:

45 rue des Saints Pères,
75006 PARIS
Subway lines: 4 (Saint Germain des Prés), 10 (Mabillon), 12 (Rue du bac)
Bus lines: 39, 95

Locations:

Talks: Amphi Lavoisier A, 3rd floor

Posters: Espace Lavoisier, 3rd floor

Coffe Break: Espace Lavoisier, 3rd floor

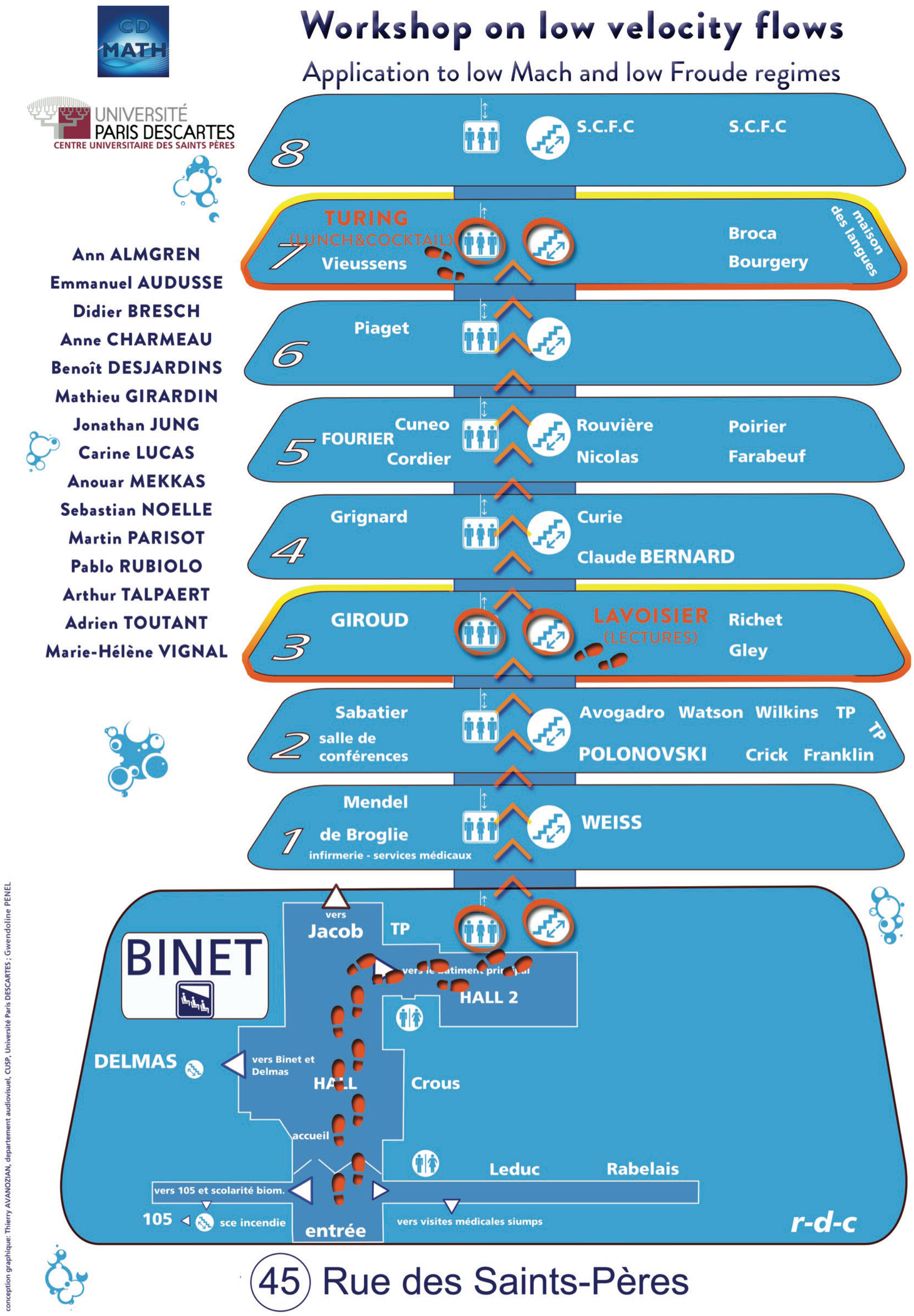
Lunch Break: Espace Turing, 7th floor

Cocktail: Espace Turing, 7th floor

Contact: lowmach2015@gmail.com

Map of the building – access to the conference room and the lunch/cocktail room

Enter the building, walk through a large hall, turn right into another hall. Take an elevator or the steps on your left to reach the third floor. Turn left, the conference room is at the end of the corridor on your left.



Program At A Glance

Time	Thursday 5/10	Friday 6/10
9h00–9h45	**Welcome**	**Didier Bresch** – see page 6 Chair: Boris Haspot
9h45–10h30	**Benoît Desjardins** – see page 7 Chair: François Bouchut	**Poster Session – see pages 9-12:** **W. Barsukow, A. Mekkas, M. Pelanti, C. Perrin, H. Zakerzadeh, C. Zaza**
10h30–11h00	**Coffee Break**	**Coffee Break**
11h00–11h45	**Anne Charmeau** – see page 6 Chair: Jean-Marc Hérard	**Mathieu Girardin** – see page 7 Chair: Caterina Calgaro
11h45–12h30	**Pablo Rubiolo** – see page 8 Chair: Jean-Marc Hérard	**Martin Parisot** – see page 8 Chair: Caterina Calgaro
12h30–14h00	**Lunch Break**	**Lunch Break**
14h00–14h45	**Sebastian Noelle** – see page 8 Chair: Christophe Chalons	**Ann Almgren – see page 6** Chair: Marie-Hélène Vignal
14h45–15h30	**Anouar Mekkas - Arthur Talpaert** – see page 8 Chair: Christophe Chalons	**Adrien Toutant** – see page 9 Chair: Marie-Hélène Vignal
15h30–16h00	**Coffee Break**	**Coffee Break**
16h00–16h45	**Marie-Hélène Vignal** – see page 9 Chair: Marica Pelanti	**Carine Lucas** – see page 7 Chair: Clément Cancès
16h45–17h30	**Jonathan Jung** – see page 7 Chair: Marica Pelanti	**Emmanuel Audusse** – see page 6 Chair: Clément Cancès
17h30–18h00		
18h00–20h00	**Cocktail**	

1 Abstracts of lectures

Low Mach Number Modeling of Stratified Astrophysical Flows

Ann Almgren
Lawrence Berkeley Lab, USA
asalmgren@lbl.gov

Computational astrophysics has traditionally relied on discretizations of either the fully compressible equations for fluid dynamics, or the anelastic approximation, supplemented by equations describing the thermonuclear reactions and heat release. The low Mach number formulation, like the anelastic approximation, analytically removes acoustic wave propagation from the system. However, the more general low Mach number approach retains nonlinear compressibility effects resulting from nuclear burning, compositional changes and changing radial stratification. This model is a generalization of the pseudo-incompressible approximation to systems with a non-ideal gas equation of state and a time-varying base state. I will discuss the derivation of the low Mach number equation set for astrophysics, focusing on the similarities with and differences from numerical models of the Earth's atmosphere.

Godunov type schemes for low Froude flows with Coriolis force

Emmanuel Audusse
LAGA, Université Paris 13
audusse@math.univ-paris13.fr

Our objective is to study the relation between two well known categories of numerical schemes: the well-balanced schemes that are designed to preserve some stationary states of the model and the low froude number schemes that are designed to remain accurate in some asymptotic regime. We study two particular cases that are related to the presence of topographic and coriolis source terms. More particularly, we analyze the linearized system in order to identify the kernel of the associated operator and to derive stable and accurate numerical schemes.

On some models for bifluid flows.

Didier Bresch
Université de Savoie
didier.bresch@univ-smb.fr

In this talk I will discuss some low Mach number systems for bifluid flows. More precisely I will focus on mathematical structures for systems used for bilayer shallow-water flows in a fixed channel or incompressible bifluid models and mixture systems used for instance for pollutant spreading or aerated avalanches (miscible flows).

This corresponds respectively to joint works with M. Renardy and with V. Giovangigli, E. Zatorska.

Low Mach number flow in plate-type fuel cores

Anne Charmeau
DEN/DANS/DM2S, CEA Saclay
anne.charmeau@gmail.com

Not communicated

Mathematical analysis of low Mach number flows for some single and two phase models

Benoit Desjardins
FMJH
benoit.desjardins@fondation-hadamard.fr

There has been a lot of progress in the last 10 years for the mathematical analysis of low Mach number flows in the case of single phase flows. Similar questions can be raised in the two phase framework, where one or the two phases are slightly compressible: this talk presents some of the additional difficulties of low Mach number analysis for two phase compressible models, which are widely used in industrial applications. Only one pressure four equations models will be considered (no temperature equations).

An all-regime Lagrange-Projection like scheme for 2D homogeneous models for two-phase flows on unstructured meshes

Mathieu Girardin
CMAP, École Polytechnique
girardin.mat@gmail.com

We propose an all regime Lagrange-Projection like numerical scheme for 2D homogeneous models for two-phase flows. By all regime, we mean that the numerical scheme is able to compute accurate approximate solutions with an under-resolved discretization, *i.e.* a mesh size and time step much bigger than the Mach number M of the mixture. The key idea is to decouple acoustic, transport and phase transition phenomenon using a Lagrange-Projection decomposition in order to treat implicitly (fast) acoustic and phase transition phenomenon and explicitly the (slow) transport phenomena. Then, extending a strategy developed in the case of the usual gas dynamics equations, we alter the numerical flux in the acoustic approximation to obtain an uniform truncation error in term of M. This modified scheme is conservative and endowed with good stability properties with respect to the positivity of the density and preserving the mass fraction within the interval $(0, 1)$. Numerical evidences are proposed and show the ability of the scheme to deal with tests where the flow regime may vary from low to high Mach values.

A low Mach correction for the Godunov scheme applied to the linear wave equation with porosity

Jonathan Jung
LMAP, Université de Pau et des Pays de l'Adour
jonathan.jung@univ-pau.fr

We study the low Mach number behavior of the Godunov finite volume scheme applied to the linear wave equation with porosity. More precisely, we extend the Hodge decomposition to a weighted L^2 space. We illustrate the influence of the cell geometry on the accuracy property at low Mach number. In the triangular case, the stationary space of the Godunov scheme approaches well enough the continuous space of constant pressure and divergent-free velocity while this is not the case in the Cartesian case. We study the properties of the modified equation associated to this Godunov scheme and we propose some correction that is continuous with respect to the Mach number.

Asymptotic limits of the Shallow Water equations

Carine Lucas
MAPMO, Université d'Orléans
carine.lucas@univ-orleans.fr

In this talk, I will explain the asymptotic limits that can be obtained from the Shallow Water equations. In the Shallow Water equations, the values of two non-dimensional numbers (the Strouhal and the Froude numbers) can be chosen in order to catch various physical phenomena. We will see that, depending on the links between the Strouhal number, the Froude number, and the scales of the topography, we get other well-known equations such as the "Lake equation" for example.

This work has been done in collaboration with Didier Bresch (LAMA, Université de Savoie, France) and Rupert Klein (Free University of Berlin, Germany).

On the stability of IMEX schemes for singular hyperbolic PDE's

Sebastian Noelle
IGPM, RWTH Aachen, Germany
noelle@igpm.rwth-aachen.de

I will discuss systems of conservation laws where some wave speeds become singular. The classic example is the low Mach number limit in gas dynamics. In the singular limit, hyperbolicity gets lost, and near the limit, explicit time discretizations become either inefficient or unstable, both due to the CFL condition. The established concept to design efficient and stable algorithms near the singular limit is a time-Implicit-Explicit splitting, called IMEX. The recent asymptotic preserving (AP) IMEX schemes are consistent with the singular limit. A key question is the asymptotic stability of these schemes. I discuss two examples: a well-known, but unstable, scheme, and an also well-known, but stable scheme. Then I present a new stability analysis for IMEX schemes, which explains the outcomes of these experiments. I also give an outlook to a new concept of splittings, the so called RS-IMEX schemes (Reference Solution IMEX), and first aplications.

Low-velocity scheme for hyperbolic conservation laws with constrains

Martin Parisot
ANGE, Inria Paris Rocquencourt
martin.parisot@inria.fr

This talk is devoted to the numerical approximation of first order conservation laws under constraints. The proposed strategy is based on a relaxation of the constraints in order to get a system of hyperbolic conservation laws. However, the new system have to be considerate at the limit of the relaxation parameter goes to zero to ensure the (formal) consistency with the initial model. We will explain how this limit is analogous to a low-Mach asymptotic and we will propose several examples and illustrations.

High temperature thermalhydraulics modeling of a Molten salt: application to the molten salt nuclear reactor

Pablo Rubiolo
IN2P3, CNRS Grenoble
pablo.rubiolo@lpsc.in2p3.fr

An overview of the ongoing efforts in the area of the thermal-hydraulics modeling of a Molten Salt Fast Reactor (MSFR) is presented. The MFSR employs a flowing liquid fuel based on a high temperature lithium fluoride salt. A molten fuel salt flow can be considered in many situations as an incompressible flow (low Match). However, several phenomena intrinsic to a molten fuel salt flow posse unique challenges (radiative heat transfer, volumetric heat source, phase change, strong neutronics feedbacks, etc.). To study some of these phenomena and to improve current CFD models an experimental facility called SWATH (Salt at WAll: Thermal ExcHanges) will be built as part of the European project SAMOFAR (2015-2019).

CDMATH library and low-Mach models applied to two-phase flows with Adaptive Mesh Refinement

Anouar Mekkas[1] **and Arthur Talpaert**[1,2]
[1] *DEN/DANS/DM2S, CEA Saclay*
[2] *CMAP*
anouar.mekkas@cea.fr and arthur.talpaert_personalwebsite@polytechnique.org

This work presents the incremental implementation of the simulation of two-phase flows in low-Mach conditions, particularly for dilating bubbles in a nuclear core. We use CDMATH, a new easy-to-use and open-source library, which relies on the rich MED Coupling library for powerful visualization. We provide simple tools for patch-based Adaptive Mesh Refinement (AMR) designed with parallel and balanced computing in mind. With AMR, we refined the coarse mesh on a set of patches in order to locally improve the precision in regions of interest and such that we finely capture changes located at the interface. The models we present are simplified but necessary steps to obtain efficient simulation of the incompressible Navier-Stokes model and of the low Mach model with interface.

This research is a joint work together with Grégoire Allaire, Stéphane Dellacherie and Samuel Kokh. It is sponsored by the CEA (French Atomic Energy Commission) and the DGA (French ministry of defense).

Turbulent kinetic energy transfers in low-Mach wall-bounded flows

Adrien Toutant
PROMES, Université de Perpignan
adrien.toutant@univ-perp.fr

The context of the study is high temperature solar receivers (SR). The main characteristics of flow inside a SR are turbulence (because flow rates are important) and temperature gradient (because only one face of the SR receives concentrated sunlight). A specific algorithm for the resolution of low-Mach equations is proposed in order to improve mass and energy conservations. Direct numerical simulations (DNS) of a simplified solar receiver in a bi-periodic plane channel are carried out in order to analyze the coupling between turbulence and temperature gradients. The results show that the temperature gradient creates asymmetric profiles of mean and fluctuating velocities. This asymmetry cannot only be explained by the fluid property variations as a function of temperature. It is truly related to the coupling of velocity / temperature, which mainly leads to an increase of the turbulent intensity at the cold side and to its decrease at the hot side. The analysis of the turbulent kinetic energy in the physical and in the spectral domains shows that temperature gradient changes the mechanisms of production, transfer and dissipation.

Finite volumes schemes preserving the low Mach number limit for the Euler system

Marie-Hélène Vignal
Mathematics Institute of Toulouse, University Toulouse 3, France
mhvignal@math.univ-toulouse.fr

I am interested in the so-called Asymptotic preserving schemes. These schemes are well known to be well adapted for the resolution of multiscale problems in which several regimes are present.

I will present the particular case of the low Mach number limit for the Euler system.

We recall that when the Mach number tends to zero, the pressure waves are very fast and this yields the fluid incompressible. When a standard explicit finite volume scheme is used, it is well known that its time step is constrained by the C.F.L. (Courant-Friedrichs, Levy) condition. In the low Mach number regime, this leads to time steps invertely proportional to the very large pressure waves velocity. Thus, explicit schemes suffer from a severe numerical constraint in low-Mach regimes. Furthermore, these schemes are not consistent in this regime. This means that they do not capture the incompressible limit even if they are used with constrained meshes.

Then, it is necessary to develop new schemes for bypassing these limitations. These new schemes must be stable and consistent in all regimes: from low Mach numbers to order one Mach numbers

I will show how to construct such a scheme for the Euler system and I will present numerical results showing the good behavior of these schemes in all regimes.

This work is done in collaboration with R. Loubère (Mathematics Institute of Toulouse, CNRS, France) and G. Dimarco (University of Ferrara, Italy).

2 Abstracts of posters

A Low-Mach solver for the Euler equations allowing for gravity source terms

Wasilij Barsukow[1], Philipp V.F. Edelmanny[2], Christian Klingenberg[1], Fritz Röpkey[2]
[1] *Department of Mathematics, Würzburg University, Germany*
[2] *Heidelberg Institute for Theoretical Studies and Heidelberg University, Germany*
w.barsukow@mathematik.uni-wuerzburg.de>

The Euler equations of inviscid hydrodynamics in the presence of a gravity g are

$$\begin{aligned} \partial_t \rho + \operatorname{div}(\rho v) &= 0, \\ \partial_t(\rho v) + \operatorname{div}\left(\rho v \otimes v + \frac{p}{M^2}\right) &= \frac{1}{M^2}\rho g, \\ \partial_t e + \operatorname{div}((e+p)v) &= \rho g \cdot v. \end{aligned}$$

These equations admit static non-vanishing solutions (called hydrostatic equilibria) and for $g = 0$ they converge towards an incompressible flow for $M \to 0$, $t > 0$.

The inability of many solvers, initially devised to capture shocks and other supersonic features, to capture the low Mach regime is a challenging problem. Apart from the explicit time integration inforcing time steps $\in O(M)$ for stability and thus making simulations time-expensive, it has been found that the schemes' diffusion grows typically as $\mathcal{O}(1/M)$. Therefore even an implicit integration in time has to overcome this additional problem. The issue has been so far addressed in a variety of different manners.

The focus of this work lies on Roe-type schemes, which, given the physical flux function $f(q)$ define the interface flux as

$$f_{i+\frac{1}{2}} : = \frac{1}{2}\left(f(q_{i+1}) + f(q_i)\right) - \frac{1}{2}D(q_{i+1} - q_i).$$

Given the Jacobian $A = f'$, the usual Roe scheme is obtained by setting $D = |A|$, evaluated in a suitable mean state between q_i and q_{i+1}.

The artificially high diffusion in the low Mach regime coming from the matrix D, modifications have been found (Weiss & Smith 1995, Turkel 1999) that lead to schemes able to resolve the low Mach limit for homogeneous Euler equations. They replace D by $P^{-1}|PA|$ for suitable invertible matrices P, and are called preconditioned schemes mainly for historical reasons.

We show that when gravity source terms are included, these schemes are not asymptotic preserving, and thus are unable to display their low Mach properties when applied to a gas in hydrostatic equilibrium. We suggest a different low Mach modification of the diffusion matrix (Miczek+ 2015, Barsukow+ in prep.), such that its scaling now does not violate the conditions of such equilibria.

In order to compare with earlier low Mach schemes we study its properties in the case of homogeneous Euler equations. In the limit $M \to 0$ we obtain a discretization of the incompressible system. We study von Neumann stability of this scheme and show experimentally that it is able to resolve features of low Mach flow even for $M \sim 10^{-10}$. For the limit system the kinetic energy emerges as an additional conserved quantity. Its evolution over time is a measure of the diffusion of the numerical scheme and is verified to be independent of M.

Numerical algorithm of the drift-flux model of two-phase flow in a porous media on staggered grid

Anouar Mekkas[1] **and Sami Kouraichi**[2]

[1] *CEA, DEN, DM2S, STMF F-91191, Gif-sur-Yvette, France*

[2] *Université Paris 13, F-93430, Villetaneuse, 99 avenue Jean-Baptiste Clément, France*

[1] `anouar.mekkas@cea.fr` and [2] `sami.kouraichi@math.univ-paris13.fr`

FLICA4 is a 3D compressible code specially devoted to reactor core analysis which solves a compressible drift-flux model for two-phase flows in a porous medium [1]. To define convective fluxes, FLICA4 uses a specific finite volume numerical method based on an extension of the Roe's approximate Riemann colocated solver [2]. Nevertheless, an analysis of this type of method shows that in low-Mach number, it is necessary to apply modifications to the 2D or 3D geometries on a Cartesian mesh otherwise this method does not converge to the right solution when the Mach number tends to zero [3]. For this reason, we apply a so-called "pressure correction". Although this correction is necessary to reach the required precision, it may produces some checkerboard oscillations in space, especially in the 1D case.

Since these checkerboard oscillations are sometimes critical and may lead to unstable resolutions or even divergence in some cases, we also investigate another numerical algorithm to solve this compressible drift-flux model in the low Mach regime. The key point is to develop a compressible solver on staggered grid since checkerboard oscillations cannot exist on this type of discretisation. The aim of this work is to present such a compressible scheme and to validate it in low Mach regime with test cases describing a simplified nuclear core.

The compressible solver on staggered grid that we develop follows the finite volume approach for all the balance equations. The time discretization of the equations is based on a semi-implicit scheme. As the equations are not linear, the solution at each time step is obtained by a Newton-Raphson iterative method. This method gives a linear system of equations for the increments of the principal variables. The chosen solution algorithm [4] consists at first in eliminating the velocity increments as functions of the pressure increments by rewriting the momentum equations. Substituting the velocity increments into the non-linear system gives a system involving only the pressure increments. The successive elimination of the scalar variables other than the pressure variable gives a linear system on the pressure. The resolution of this linear system will allow to determine the velocity and the other variables. We will present preliminary numerical experiments will be presented and compare them with analytic solutions [5].

References

[1] A. Bergeron and Ph. Fillion and D. Gallo and E. Royer, *FLICA4 v1.8. Modèles physiques*, Note Technique CEA 2005, SFME/LETR/RT/02-005/A.

[2] S. Clerc and Ph. Fillion, *FLICA4 v1.8. Méthode numérique*, Note Technique CEA 2002, SFME/LETR/RT/02-005/A.
[3] S. Dellacherie, *Analysis of Godunov type schemes applied to the compressible Euler system at low Mach number*, J. Comp, Phys., 4(229):978-1016, 2010.
[4] M. Chandesris, *Discrétisation des équations du module 3D de Cathare 2 & Comparaison avec le module 1D - État des lieux pour la version v2.52 et perspectives*, Note Technique CEA 2012, DER/SSTH/LDLD/NT/2011-046/A.
[5] M. Bernard, S. Dellacherie, G. Faccanoni, B. Grec and Y. Penel, *Study of a low Mach nuclear core model for two-phase flows with phase transition I: stiffened gas law*, 2015, https://hal.archives-ouvertes.fr/hal-01111730.

A preconditioned Roe-type scheme for a two-phase compressible flow model at low Mach number

Marica Pelanti
École Nationale Supérieure de Techniques Avancées - ENSTA ParisTech
marica.pelanti@ensta-paristech.fr

We describe two-phase compressible flows by a hyperbolic six-equation single-velocity two-phase flow model with stiff mechanical relaxation [1]. In particular, we are interested in the simulation of liquid-gas mixtures such as cavitatingflows, which are relevant in numerous engineering devices. The model equations are numerically approximated via a fractional step algorithm, which alternates between the solution of the homogeneous hyperbolic portion of the system through wave propagation finite volume schemes, and the solution of a system of ordinary differential equations that takes into account the pressure relaxation terms. When used in this algorithm, classical schemes such as Roe's or HLLC prove to be very efficient to simulate the dynamics of transonic and supersonic flows. Unfortunately, these methods suffer from the well known difficulties of loss of accuracy and efficiency for low Mach number regimes encountered by upwind finite volume discretizations. This issue is particularly critical for liquid-gas flows due to the large and rapid variation of the Mach number, since the speed of sound may range from very low values in the two-phase mixture to very large values in the liquid medium. To cure the problem of loss of accuracy at low Mach number, in this work we extend to the considered two-phase flow model the strategy of the preconditioned Roe-Turkel scheme of Guillard-Viozat [2] for the single-phase Euler's equations. A suitable set of entropic variables for the two-phase system is chosen to define the Turkel-type preconditioning matrix that corrects the numerical dissipation term of the finite volume scheme at low Mach number. As for the single-phase case, the resulting technique alters only the acoustic characteristic fields of the model at low Mach number, while interface (contact) waves are preserved unchanged. We present numerical results for two-dimensional liquid-gas channel flow tests that show the effectiveness of the proposed preconditioned method for the two-phase system. In particular, we show that in the low Mach number limit pressure perturbations at the discrete level correctly scale with the square of the reference Mach number, in agreement with the theoretical results for the continuous two-phase flow model. Preliminary numerical experiments for problems involving liquid-vapor transition for the full model with heat and mass transfer terms will be also presented.

References

[1] M. Pelanti and K.-M. Shyue, *A mixture-energy-consistent six-equation two-phase numerical model for fluids with interfaces, cavitation and evaporation waves*, J. Comput. Phys. 259 (2014), 331–357.
[2] H. Guillard and C. Viozat, *On the behaviour of upwind schemes in the low Mach number limit*, Computers and Fluids 28 (1999), 63–86.

Singular compressible Navier-Stokes equations leading to an incompressible system with pressure dependent viscosity

Charlotte Perrin
LAMA, Université de Savoie, France
charlotte.perrin@univ-smb.fr

I will present a work dedicated to the singular limit passage between a model for suspension flows and an original incompressible model for granular media. The model for suspension flows consists of compressible Navier-Stokes equations with pressure, representing repulsion forces, and viscosities, representing resistance to deformations, singular close to the maximal close-packing volume fraction. Performing an appropriate limit passage, this model is shown to converge towards a fully incompressible system with pressure dependent viscosity and with an additional equation linking the Lagrange multiplier associated to the incompressible constraint with the evolution of new quantity, the adhesion potential, which keeps track of the history of the flow.

On the Mach-Uniformity of Lagrange-Projection Scheme

Hamed Zakerzadeh
Department of Mathematics, IGPM, RWTH Aachen, Germany
h.zakerzadeh@igpm.rwth-aachen.de

In the present work, we show that the Lagrange–projection scheme presented in Coquel et al.'s paper (Math. of Comp. **79**.271 (2010): 1493–1533), is asymptotic preserving for isentropic Euler equations, *i.e.* at the discrete level it preserves the incompressible limit, satisfies the div-free condition as well as the asymptotic expansion for the density in the continuous level. Moreover, we prove that the scheme is positivity-preserving, L_∞-stable and entropy-admissible under some Mach-uniform restrictions. The analysis is similar to what has been presented in the original paper, but with the emphasis on the uniformity regarding the Mach number.

An explicit second order scheme on staggered finite volume meshes for reactive flows with stiff kinetics

Chady Zaza and Jean-Claude Latché
Institut de Radioprotection et de Sûreté Nucléaire, IRSN/PSN-RES/SA2I/LIE, 13115 Saint-Paul-Lez-Durance
{chady.zaza,jean-claude.latche}@irsn.fr

In hydrogen safety issues distinct physical phenomena, following each other, have to be addressed. The typical process yielding an hydrogen-oxygen explosion starts with the mixing of the reactants (subsonic, low speed), the ignition and propagation of a flame front (subsonic, high speed) which eventually turns into a detonation wave (supersonic). Simulating the entire process from end to end may involve different flow solvers, in which case the use of a common mesh discretization becomes an important feature, since it drastically simplifies code-to-code communications. This is the strategy followed for the development of the P^2REMICS (Partially PREMIxed Combustion Solver) software at IRSN.

The choice of staggered finite volumes is interesting for its inferred stability properties in the incompressible limit. An explicit second order finite volume scheme on staggered grids for solving the compressible Euler equations [1] was previously introduced. We now extend this numerical scheme to the reactive Euler equations in the ZND framework (finite reaction rate) for detonation problems. Consistent results are obtained on 1D problems as long as the space and time scales of the chemical reaction are well resolved, which may require unreasonably fine discretizations.

Indeed on coarser discretizations, a small amount of diffusion can have a dramatic impact on the computation of the reaction rate and unphysical solutions are likely to appear [2]. We use a local reconstruction with Harten's subcell resolution technique similar to [3] in order to improve the accuracy of the source term and prevent an early transition from an intermediate chemical state to the burnt state. Solutions obtained from 1D numerical tests feature a consistent wave pattern and the correct wave propagation speed.

References

[1] R. Herbin, J.-C. Latché and N. Therme, *A MUSCL-type stable explicit staggered scheme for the compressible Euler equations*, (in preparation), 2015.
[2] R. J. LeVeque and H. C. Yee, *A study of numerical methods for hyperbolic conservation laws with stiff source terms*, Journal of Computational Physics, vol. 86, pp. 187–210, 1990.
[3] W. Wang, C.-W. Shu, H.C. Yee and B. Sjogreen, *High order finite difference methods with subcell resolution for advection equations with stiff source terms*, Journal of Computational Physics, vol. 231, pp. 190–214, 2012.