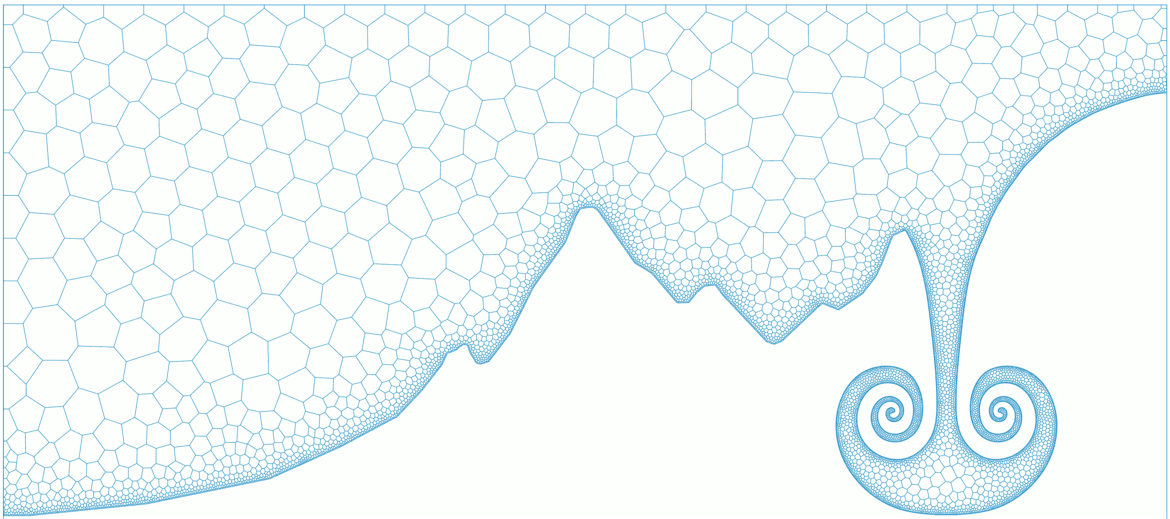


NumHyp 2021

7th international conference on

Numerical methods for Hyperbolic Problems

Book of abstracts

26 – 30 July 2021

Hotel Villa Madruzzo, Trento, Italy

Organized by the University of Trento

The instrument that mediates between theory and practice, between thought and observation, is mathematics; it builds the connecting bridge and makes it stronger and stronger. Thus it happens that our entire present-day culture, insofar as it rests on intellectual insight into and harnessing of nature, is founded on mathematics. Already, Galileo said: Only he can understand nature who has learned the language and signs by which it speaks to us; but this language is mathematics and its signs are mathematical figures. Kant declared, "I maintain that in each particular natural science there is only as much true science as there is mathematics." In fact, we do not master a theory in natural science until we have extracted its mathematical kernel and laid it completely bare. Without mathematics today's astronomy and physics would be impossible; in their theoretical parts, these sciences unfold directly into mathematics. These, like numerous other applications, give mathematics whatever authority it enjoys with the general public.

David Hilbert

Radio address given in Königsberg on 8 September 1930, on the occasion of the yearly meeting of the Society of German Natural Scientists and Physicians (English translation by James T. Smith)

This conference is dedicated to the 75^{th} birthday of Prof. Dr. Dr. h.c. Eleuterio F. Toro, OBE

Scientific committee:

Emmanuel Audusse *(University of Paris 13, France)*

Christophe Berthon *(University of Nantes, France)*

Mária Lukáčová-Medvid'ová *(University of Mainz, Germany)*

Pep Mulet *(University of Valencia, Spain)*

Sebastian Noelle *(RWTH Aachen, Germany)*

Carlos Parés *(University of Málaga, Spain)*

Gabriella Puppo *(University La Sapienza, Italy)*

Giovanni Russo *(University of Catania, Italy)*

Organizing committee:

Michael Dumbser *(University of Trento, Italy)*

Saray Busto *(Universidad Politécnica de Madrid, Spain)*

Ilya Peshkov *(University of Trento, Italy)*

Simone Chiocchetti *(University of Trento, Italy)*

Laura del Río-Martín *(University of Trento, Italy)*

Conference secretary:

Lorena Galante (University of Trento, Italy)

This conference has been supported by:

European Commission, via the FET-HPC project ExaHyPE

European Research Council (ERC)

University of Trento

Ministero dell'Istruzione e dell'Università

Agenzia per il Turismo di Trento

Welcome to NumHyp 2021!

We are very pleased to welcome you to NUMHYP 2021, *the 7^{th} International Conference on Numerical Methods for Hyperbolic Problems, held in Trento, Italy.*

NumHyp 2021 is the seventh in a series of biannual conferences that began with a meeting in Castro Urdiales, Spain in 2009. Further editions of this conference were held in Roscoff, France in 2011, Aachen, Germany in 2013, Cortona, Italy in 2015, Monte Verità, Switzerland in 2017 and in Málaga, Spain in 2019. These conferences focus on recent developments and new directions in the area of numerical methods for hyperbolic and convection dominated time-dependent partial differential equations (PDEs). These PDEs arise in a large number of models in science and engineering. Prominent examples include the compressible and incompressible Euler and Navier-Stokes equations, the shallow water equations, the magneto-hydrodynamics equations, multiphase fluid models, hyperbolic formulations of continuum mechanics and even general relativity. Examples of application areas are aerodynamics, oceanography, plasma physics, solid mechanics, computational astrophysics etc.

These PDEs have been subject to extensive analytical and numerical studies over the last decades. It is widely known that their solutions can exhibit very complex behavior including the simultaneous presence of smooth waves, like acoustic or electromagnetic waves, as well as discontinuities such as shock waves and material interfaces. They also exhibit a sensitive dependence on initial conditions, presence of multiple scales in space and time, appearance of turbulent regimes, etc. The design and the analysis of numerical methods with good properties to solve them are still major challenges even nowadays.

NumHyp 2021 is a key activity of the PRIN 2017 project *Innovative numerical methods for evolutionary partial differential equations and applications*, funded by the Italian Ministry of University and Research (MIUR).

We would like to thank the scientific committee, our sponsors and all the participants for their enrichening contributions and we wish you a very pleasant stay in Trento and productive scientific and personal interactions during the conference!

Michael Dumbser
Chairman of the NumHyp 2021 organization committee

Saray Busto, Ilya Peshkov, Simone Chiocchetti, Laura del Río-Martín
NumHyp 2021 local organizing committee

NumHyp 2021, 26 – 30 July 2021, Trento, Italy

Program of the Seventh International Conference on Numerical Methods for Hyperbolic Problems

	Monday, 26.7.2021	Tuesday, 27.7.2021	Wednesday, 28.7.2021	Thursday, 29.7.2021	Friday, 30.7.2021
08:40 – 09:00	*Opening and Welcome*				
09:00 – 09:30	Abgrall	Klingenberg	Bassi	Loubère	Després
09:30 – 10:00	Munz	Gómez-Bueno	Russo	Puppo	Iollo
10:00 – 10:30	Thomann	Kurganov	Groppi	Cristiani	Boscheri
10:30 – 11:30	*Coffee Break*	*Coffee Break*	*Coffee Break*	*Coffee Break*	*Coffee Break*
11:30 – 12:00	Drikakis	Chen	M. Castro	Pareschi	Hidalgo
12:00 – 12:30	Lukáčová-Medvid'ová	Parés	Peshkov	Cordero-Carrión	Vergara
12:30 – 13:00	Vázquez-Cendón	Tscherpel	Busto	Gaburro	Müller
13:00 – 15:00	*Lunch*	*Lunch*	*Lunch*	*Lunch*	*Lunch*
15:00 – 15:30	Río-Martín / Torlo	Macca / Yuan	Falcone / Carlino	Gavrilyuk	Dimarco / Bertaglia
15:30 – 16:00	Putti / Scholz	Siviglia / Öffner	Semplice / Solán	Parisot	Titarev / Celant
16:00 – 16:30	C. Castro / Pimentel	Caballero / Nikodemou	Albi / Groom	Iske	Gerster / Ghitti
16:30 – 17:30	*Coffee Break*	*Coffee Break*	*Coffee Break*	*Coffee Break*	Coclite / Spilimbergo
17:30 – 18:00	Fambri / Chrysanthou	Sherwin / Echeverribar	Chiocchetti / Wallis	Chertok	*Coffee Break*
18:00 – 18:30	She / Orlando	Gassner / Del Grosso	Dhaouadi / Dematté	Nikiforakis	*Room Madruzzo* / *Room Belvedere*
18:30 – 19:00	Güçlü / Millmore	Schneider / Conde	Dumbser / Mi	Toro	
19:00 – 19:30	*Room Madruzzo* / *Room Belvedere*	*Room Madruzzo* / *Room Belvedere*	*Room Madruzzo* / *Room Belvedere*		
20:30 – . . .	*Welcome Reception*			*Conference Dinner*	

Daily program: Monday, 26 July 2021

08:40 – 09:00 OPENING AND WELCOME – ROOM MADRUZZO

09:00 – 09:30 **Rémi Abgrall** (University of Zürich)
On the notion of conservation for hyperbolic problem

09:30 – 10:00 **Claus-Dieter Munz** (University of Stuttgart)
The Riemann problem in the sharp interface approximation of two-phase flow with evaporation

10:00 – 10:30 **Andrea Thomann** (University of Mainz)
An all-speed scheme for isentropic two phase flow

10:30 – 11:30 COFFE BREAK

11:30 – 12:00 **Dimitris Drikakis** (University of Nicosia)
Uncertainty reduction in turbulent simulations using high-order methods

12:00 – 12:30 **Mária Lukáčová-Medvid'ová** (University of Mainz)
Approximating viscosity solutions of the Euler equations

12:30 – 13:00 **María Elena Vázquez-Cendón** (University of Santiago de Compostela)
Evolution of a hybrid finite volume/finite element scheme for low-Mach number flows to all Mach number flows

13:00 – 15:00 LUNCH

	ROOM MADRUZZO	ROOM BELVEDERE
15:00 – 15:30	**Laura Río-Martín** (University of Trento) *A family of semi-implicit hybrid FV/FE methods for computational fluid dynamics using an efficient MPI parallel implementation*	**Davide Torlo** (Inria Bordeaux Sud-Ouest) *Continuous Galerkin high order well-balanced discrete kinetic model for shallow water equations*
15:30 – 16:00	**Mario Putti** (University of Padua) *Geometrically intrinsic shallow water equations on moving surfaces*	**Ullika Scholz** (RWTH Aachen University) *Dispersive moment equations for shallow flow*
16:00 – 16:30	**Cristóbal Castro** (University of Taparacá) *High-Order Splitting schemes for the shallow water equations with applications to tsunami wave propagation*	**Ernesto Pimentel-García** (University of Málaga) *Well-balanced methods for relativistic fluids on a Schwarzschild background*

16:30 – 17:30 COFFE BREAK

17:30 – 18:00	**Francesco Fambri** (Max Planck Institute for Plasma Physics) *A novel structure preserving semi-implicit finite volume method for viscous and resistive magnetohydrodynamics*	**Maria Chrysanthou** (University of Cambridge) *A computational multi-physics approach for nuclear fusion reactor simulations*
18:00 – 18:30	**Bangwei She** (Institute of Mathematics of the CAS, Prague) *On convergence of numerical solutions for the compressible MHD system*	**Giuseppe Orlando** (Politecnico di Milano) *An efficient and accurate implicit DG solver for the incompressible Navier–Stokes equations*
18:30 – 19:00	**Yaman Güçlü** (Max Planck Institute for Plasma Physics) *Efficient compatible finite element solution of the time-dependent Maxwell equations on mapped grids*	**Stephen Millmore** (University of Cambridge) *Multi-physics simulations of lightning strikes on elastoplastic substrates*

20:30 – . . . WELCOME RECEPTION

Daily program: Tuesday, 27 July 2021

09:00 **Opening – Room Madruzzo**

09:00 – 09:30 **Christian Klingenberg** (University of Würzburg)
Structure preserving numerical methods for the Euler equations with gravity

09:30 – 10:00 **Irene Gómez-Bueno** (University of Málaga)
Collocation methods for high-order well-balanced methods for 1D systems of balance laws

10:00 – 10:30 **Alexander Kurganov** (Southern University of Science and Technology, Shenzhen)
High-order path-conservative central-upwind schemes

10:30 – 11:30 **Coffe Break**

11:30 – 12:00 **Guoxian Chen** (Wuhan University)
A unified surface-gradient and hydrostatic reconstruction scheme for the shallow water equations

12:00 – 12:30 **Carlos Parés** (University of Málaga)
Well-balanced high-order finite difference WENO methods for systems of balance laws

12:30 – 13:00 **Tabea Tscherpel** (Bielefeld University)
Boundary conditions for time-discrete Green-Naghdi equations

13:00 – 15:00 **Lunch**

	Room Madruzzo	Room Belvedere
15:00 – 15:30	**Emmanuele Macca** (University of Catania) *Adaptive high order well balanced compact approximate method for systems of balance law*	**Yuhuan Yuan** (University of Mainz) *Convergence of the Godunov method for multidimensional compressible Euler equations*
15:30 – 16:00	**Annunziato Siviglia** (University of Trento) *A second-order well-balanced splitting scheme for the non-conservative Saint-Venant-Exner model*	**Philipp Öffner** (University of Mainz) *Convergence of DG schemes for the Euler equations via dissipative weak solutions*
16:00 – 16:30	**Celia Caballero-Cárdenas** (University of Málaga) *An exactly well-balanced semi-implicit Lagrange-projection type scheme for the shallow-water system*	**Maria Nikodemou** (University of Cambridge) *A unified multi-phase and multi-material formulation for combustion modelling*

16:30 – 17:30 **Coffe Break**

17:30 – 18:00	**Spencer Sherwin** (Imperial College London) *Development and application of a spectral/hp element, implicit compressible solver*	**Isabel Echeverribar** (University of Zaragoza) *Evaluation of the performance of two non-hydrostatic shallow water models for the simulation of steady and unsteady flows*
18:00 – 18:30	**Gregor Gassner** (University of Cologne) *On compatible Legendre-Gauss-Lobatto subcell low order finite volume methods (and what we can do with it)*	**Alessia Del Grosso** (University of Versailles) *On second-order well-balanced Lagrange-projection schemes for shallow water Exner system*
18:30 – 19:00	**Kleiton A. Schneider** (University of Mato Grosso do Sul) *Multidimensional approximate Riemann solvers for hyperbolic nonconservative systems*	**Daniel Conde** (ETH Zürich) *An efficient implementation of turbulent-diffusive processes and suspended sediment transport in shallow-water models: hyperbolization, flux splitting approach and GPU acceleration*

Daily program: Wednesday, 28 July 2021

09:00 OPENING – ROOM MADRUZZO

09:00 – 09:30 **Francesco Bassi** (University of Bergamo)
Oscillation control in discontinuous Galerkin solutions of the 1D Euler equations

09:30 – 10:00 **Giovanni Russo** (University of Catania)
Conservative semi-Lagrangian methods for kinetic equations

10:00 – 10:30 **Maria Groppi** (University of Parma)
BGK models for gas mixtures: asymptotics and numerics

10:30 – 11:30 COFFE BREAK

11:30 – 12:00 **Manuel Castro** (University of Málaga)
Artificial viscosity to get both robustness and discrete entropy inequalities

12:00 – 12:30 **Ilya Peshkov** (University of Trento)
Computational aspects of the unified hyperbolic formulation for continuum mechanics

12:30 – 13:00 **Saray Busto** (Universidad Politécnica de Madrid)
Thermodynamically compatible schemes for continuum mechanics

13:00 – 15:00 LUNCH

	ROOM MADRUZZO	ROOM BELVEDERE
15:00 – 15:30	**Maurizio Falcone** (Università di Roma "La Sapienza") *A tree structured method for high-dimensional evolutive Hamilton-Jacobi equations and applications*	**Michele Giuliano Carlino** (Inria Bordeaux Sud-Ouest) *ADER scheme for incompressible Navier-Stokes equations on overset grids with a compact transmission condition*
15:30 – 16:00	**Matteo Semplice** (Università dell'Insubria, Como) *One- and multi-dimensional CWENOZ reconstructions for implementing boundary conditions without ghost cells*	**Pablo Solán-Fustero** (University of Zaragoza) *Application of approximate dispersion-diffusion analyses to under-resolved Burgers turbulence using high resolution WENO and UWC schemes*
16:00 – 16:30	**Giacomo Albi** (University of Verona) *IMEX multistep method for hyperbolic systems with relaxation*	**Michael Groom** (University of Sydney) *Comparison of high-resolution reconstruction schemes in unsteady low Mach number flows*

16:30 – 17:30 COFFE BREAK

17:30 – 18:00	**Simone Chiocchetti** (University of Trento) *Tips and tricks for simple hyperbolic viscous flow*	**Tim Wallis** (University of Cambridge) *A Flux-enriched Godunov method for multi-material problems with interface slide and void opening*
18:00 – 18:30	**Firas Dhaouadi** (University of Trento) *A hyperbolic augmented model for the Nonlinear Schrödinger equation*	**Riccardo Dematté** (Univeristy of Cambridge) *Reacting condensed phase explosives in direct contact*
18:30 – 19:00	**Michael Dumbser** (University of Trento) *High order ADER discontinuous Galerkin schemes for nonlinear hyperelasticity with material failure*	**XiaoCheng Mi** (McGill University, Montréal) *GPU-accelerated meso-resolved simulation of detonation waves in multiphase energetic materials*

Daily program: Thursday, 29 July 2021

09:00 OPENING – ROOM MADRUZZO

09:00 – 09:30 **Raphaël Loubère** (University of Bordeaux)
Towards bridging Lagrangian and Eulerian Riemann solvers

09:30 – 10:00 **Gabriella Puppo** (Università di Roma “La Sapienza”)
Traffic models, or what we can learn coupling transport and source terms

10:00 – 10:30 **Emiliano Cristiani** (CNR, Roma)
Macroscopic and multi-scale models for multi-class vehicular dynamics with uneven space occupancy: a case study

10:30 – 11:30 COFFE BREAK

11:30 – 12:00 **Lorenzo Pareschi** (University of Ferrara)
Hyperbolic models and numerical methods for the spatial spread of infectious diseases

12:00 – 12:30 **Isabel Cordero-Carrión** (University of Valencia)
Numerical evolution of the resistive relativistic magnetohydrodynamic equations: a minimally implicit Runge-Kutta scheme

12:30 – 13:00 **Elena Gaburro** (Inria Bordeaux Sud-Ouest)
A well balanced finite volume scheme for general relativity

13:00 – 15:00 LUNCH

15:00 – 15:30 **Sergey Gavrilyuk** (Aix-Marseille University)
Singular solutions of the BBM equation: analytical and numerical study

15:30 – 16:00 **Martin Parisot** (Inria Bordeaux Sud-Ouest)
On the 1D steady states of the 1D Green-Naghdi equations

16:00 – 16:30 **Armin Iske** (University of Hamburg)
Flexible kernels for particle-based fluid flow simulations

16:30 – 17:30 COFFE BREAK

17:30 – 18:00 **Alina Chertok** (North Carolina State University, Raleigh)
Well-balancing via flux globalization: applications to shallow water equations with wet/dry fronts

18:00 – 18:30 **Nikos Nikiforakis** (University of Cambridge)
Computational multiphysics for interacting states of matter under extreme conditions

18:30 – 19:30 **Eleuterio Francisco Toro** (University of Trento)
Some models and methods for physiological flows in collapsible conduits

20:30 – . . . CONFERENCE DINNER

Daily program: Friday, 30 July 2021

09:00 OPENING – ROOM MADRUZZO

09:00 – 09:30 **Bruno Després** (LJLL Sorbonne University, Paris)
The implicit Lagrangian Riemann problem: how and why?

09:30 – 10:00 **Angelo Iollo** (Inria Bordeaux Sud-Ouest)
Discretization of a simple hyperbolic system rising in incompressible fluid-structure interaction

10:00 – 10:30 **Walter Boscheri** (University of Ferrara)
3D cell-centered Finite Volume schemes for solving updated Lagrangian hyperelasticity on unstructured grids

10:30 – 11:30 COFFE BREAK

11:30 – 12:00 **Arturo Hidalgo** (Universidad Politécnica de Madrid)
An ADER-WENO numerical scheme for a porous-medium mathematical model of atherosclerosis

12:00 – 12:30 **Christian Vergara** (Politecnico di Milano)
Fluid-structure interaction problems for blood flow in carotids

12:30 – 13:00 **Lucas Müller** (University of Trento)
Hyperbolic equations in computational haemodynamics: models, numerics and physiology

13:00 – 15:00 LUNCH

	ROOM MADRUZZO	ROOM BELVEDERE
15:00 – 15:30	**Giacomo Dimarco** (University of Ferrara) *High order finite volume schemes with IMEX time stepping for the Boltzmann model on unstructured meshes*	**Giulia Bertaglia** (University of Ferrara) *Stochastic asymptotic-preserving IMEX Finite Volume methods for viscoelastic models of blood flow*
15:30 – 16:00	**Vladimir Titarev** (FRC Computer Science and Control, Moscow) *ALE-type discrete velocity scheme for kinetic equations as applied to rapid gas expansion problems*	**Morena Celant** (University of Trento) *AENO: a novel reconstruction method in conjunction with ADER schemes for hyperbolic equations*
16:00 – 16:30	**Stephan Gerster** (RWTH Aachen) *Hypocoercivity of Stochastic Galerkin formulations for stabilization of Kinetic Equations*	**Beatrice Ghitti** (University of Trento) *Blood flow simulations in hybrid 1D-0D networks based on a priori model selection criteria*
16:30 – 17:00	**Alessandro Coclite** (Politecnico di Bari) *Strategies for time integration in fluid/structures interaction problems within dynamic-IB methods*	**Alessandra Spilimbergo** (University of Trento) *One-dimensional blood flow with discontinuous properties and transport: mathematical analysis and numerical schemes*

17:00 – 18:00 COFFE BREAK

On the notion of conservation for hyperbolic problem

R. Abgrall†
† Institut für Mathematik, Universität Zürich, Zürich, Switzerland
(remi.abgrall@math.uzh.ch)

ABSTRACT

Let us consider a general hyperbolic problem (H) with initial and boundary conditions. Since the celebrated Lax-Wendroff theorem, one knows what should be the structural form of a finite volume schemes to garanty that, if the scheme converges towards some element of L^2 (say), that this function is a weak form of the problem. The same applies to all possible entropy conditions attached to (H). It is also known that if this condition fails, then the limit solution, if any, will not be a weak solution, see [7]. This knowledge is also extended to discontinuous Galerkin methods. The Lax-Wendroff theorem rely on the notion of consistant flux; they are attached to edges/faces of the tessaletion that is used to approximate (H). However, there exists "working" schemes that do not fall into this framework. A good example is that of the streamline diffusion method, [8] for a convergence proof.

In this talk, which is based on [3, 1, 2, 6, 5, 4], I will show that using the framework of residual distribution scheme, where the notion of local conservation writes differently than in the classical setting (which is mostly one dimensional), on can rewrite equivalently most if not all the schemes I know as finite volume scheme, provide formula for the flux (that are not classical), as well as the control volumes. Going further, this framework provides answers to the following questions: given my favorite scheme, how can I modify it so that it satisfies additional constraints. For example, how can I make it entropy conservative, or entropy dissipative? In this talk I will provide more examples about these questions, and explain what cannot yet be done within this framework.

This is a joint work with many colleagues, former PhDs and postdocs: Mario Ricchiuto (INRIA), Paola Bacigaluppi (Poli Milano), Sveltana Tokareva (Los Alamos), Philipp Öffner (mainz), Henrik Ranocha (Munster), Fatemeh Mojarrad (UZH)

References

[1] R. Abgrall. A general framework to construct schemes satisfying additional conservation relations. Application to entropy conservative and entropy dissipative schemes. *J. Comput. Phys.*, 372:640–666, 2018.

[2] R. Abgrall, P. Bacigaluppi, and S. Tokareva. A high-order nonconservative approach for hyperbolic equations in fluid dynamics. *Comput. & Fluids*, 169:10–22, 2018.

[3] Rémi Abgrall. Some remarks about conservation for residual distribution schemes. *Comput. Methods Appl. Math.*, 18(3):327–351, 2018.

[4] Rémi Abgrall, Konstantin Lipnikov, Nathaniel Morgan, and Svetlana Tokareva. Multidimensional staggered grid residual distribution scheme for Lagrangian hydrodynamics. *SIAM J. Sci. Comput.*, 42(1):A343–A370, 2020.

[5] Rémi Abgrall and Svetlana Tokareva. Staggered grid residual distribution scheme for Lagrangian hydrodynamics. *SIAM J. Sci. Comput.*, 39(5):A2317–A2344, 2017.

[6] Rémi Abgrall, Philipp Öffner, and Hendrik Ranocha. Reinterpretation and Extension of Entropy Correction Terms for Residual Distribution and Discontinuous Galerkin Schemes, 2020.

[7] Thomas Y. Hou and Philippe G. LeFloch. Why nonconservative schemes converge to wrong solutions: error analysis. *Math. Comp.*, 62(206):497–530, 1994.

[8] Anders Szepessy. Convergence of a shock-capturing streamline diffusion finite element method for a scalar conservation law in two space dimensions. *Math. Comp.*, 53(188):527–545, 1989.

IMEX multistep method for hyperbolic systems with relaxation

G. Albi[†], **G. Dimarco**[‡] **and L. Pareschi**[‡]
[†] Department of Computer Science, University of Verona (giacomo.albi@univr.it)
[‡] Department of Mathematics and Computer Science, University of Ferrara
(giacomo.dimarco@unife.it,lorenzo.pareschi@unife.it)

ABSTRACT

In this talk, we consider the development of Implicit-Explicit (IMEX) Multistep time integrators for hyperbolic systems with relaxation. More specifically, we consider the case in which the transport and the relaxation part of such systems have different time and space scales. The consequence is that the nature of the asymptotic limit can differ, passing from an hyperbolic to a parabolic character. From the computational point of view, this causes many drawbacks that standard time integrators, even implicit ones, are not able to handle: loss of efficiency and loss of capability in describing the limit regime. In this work, we construct highly stable numerical methods which describe all the different regimes with high accuracy in time and space that are able to capture the correct asymptotic limit. Several numerical examples confirm the consistency and linear stability analysis and show that the proposed methods outperform existing ones.

Keywords: implicit-explicit methods, linear multistep methods, hyperbolic balance laws, fluid-dynamic limit, diffusion limit, asymptotic-preserving schemes

References

[1] G. Albi, G. Dimarco, L. Pareschi, Implicit-Explicit multistep methods for hyperbolic systems with multiscale relaxation, *SIAM J. Sci. Comp.*, Vol. 42, No. 4: pp. A2402-A2 435, 2020.

[2] S. Boscarino, L. Pareschi, and G. Russo, A unified IMEX Runge-Kutta approach for hyperbolic systems with multiscale relaxation, SIAM J. Numer. Anal., 55, pp. 2085–2109, 2017.

Oscillation Control in Discontinuous Galerkin Solutions of the 1D Euler Equations

F. Bassi†

† retired, former professor at Engineering and Applied Sciences Department, University of Bergamo, Italy (`francesco.bassi@unibg.it, francesco.bassi50@gmail.com`)

ABSTRACT

This work presents an approach to the control of oscillations in numerical solutions of the 1D Euler equations, discretised in space by means of a high-order Discontinuous Galerkin (DG) method. The proposed approach is of the artificial viscosity type, with special care devoted to the formulation of the viscosity function and to its distribution within elements. Such viscosity is defined on an element-by-element basis and depends on the residuals of the DG discretisation, hence does not affect the accuracy of smooth solutions and avoids using any smoothness indicator. It is worth noting that the method does not entail any interface flux term related to the artificial viscosity operator. Comprehensive details about the formulation will be given at the presentation.

The following Figures show the numerical results of some classical 1D test problems for the Euler equations. The filled symbols represent the numerical results of the *k-th* degree polynomial solutions at the k+1 Gauss-Legendre integration points within each element, and allow to appreciate the resolution capability of the method and its suitability also for very high-order DG approximations.

Keywords: Euler equations; Discontinuous Galerkin schemes; High-order discretisation; Artificial viscosity; Nonlinear stabilisation.

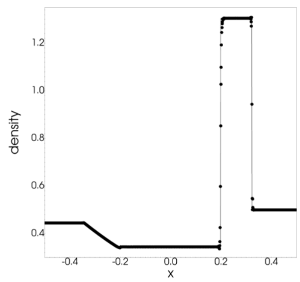


Figure 1: Lax problem, 200 P^4 elements

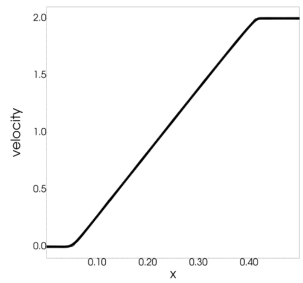


Figure 2: Einfeldt problem, 8 P^{40} elements

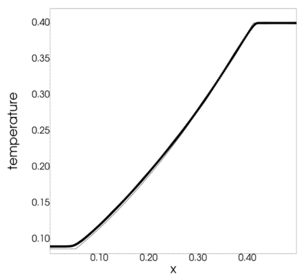


Figure 3: Einfeldt problem, 8 P^{40} elements

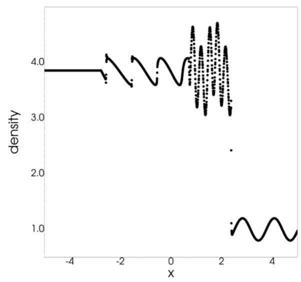


Figure 4: Shu-Osher problem, 400 P^4 elements

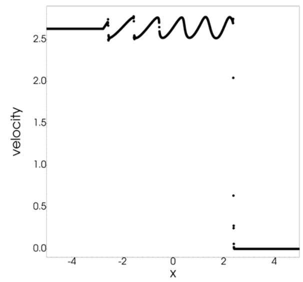


Figure 5: Shu-Osher problem, 400 P^4 elements

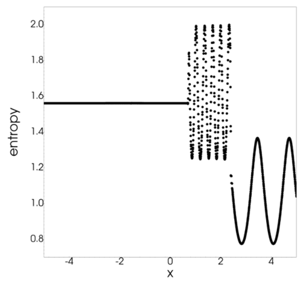


Figure 6: Shu-Osher problem, 400 P^4 elements

Stochastic asymptotic-preserving IMEX Finite Volume methods for viscoelastic models of blood flow

G. Bertaglia[†], **V. Caleffi**[‡], **L. Pareschi**[†] **and A. Valiani**[‡]

[†] Department of Mathematics and Computer Science, University of Ferrara

({giulia.bertaglia,lorenzo.pareschi}@unife.it)

[‡] Department of Engineering, University of Ferrara

({valerio.caleffi,alessandro.valiani}@unife.it)

ABSTRACT

When applying computational blood flow models to patient-specific simulations for clinical decision-making, the computational inputs that need to be personalized constitute possible sources of error, given the large biological variability and the uncertainty underlying all measurements [4]. Therefore, the development and application of efficient computational methods for the assessment of the impact of parametric fluctuations on numerical solutions is essential for a correct interpretation of the results. Moreover, viscoelastic properties of vessel walls have been recognized as one of the features which must be realistically included in a mathematical model to obtain accurate simulations and correctly estimate pressure trends [3].

To investigate the effects of uncertainties of parameters involved in computational hemodynamics, with particular concern on those defining the viscoelastic vessel wall behavior, we propose a stochastic asymptotic-preserving IMEX Finite Volume scheme [1], which guarantees spectral convergence in the stochastic space and ease of implementation, avoiding the risk of loss of hyperbolicity of the system of stochastic equations. The method is applied to solve the 1D a-FSI blood flow model [2], presenting numerical results of univariate and multivariate uncertainty quantification analyses on baseline and patient-specific tests. Computed pressure waveforms are compared with in-vivo records.

Keywords: Uncertainty quantification; Stochastic collocation methods; IMEX Runge-Kutta schemes; Finite volume methods; Blood flow models; Fluid-structure interaction.

References

[1] G. Bertaglia, V. Caleffi, L. Pareschi, and A. Valiani. Uncertainty quantification of viscoelastic parameters in arterial hemodynamics with the a-FSI blood flow model. *Journal of Computational Physics*, 430:110102, 2021.

[2] G. Bertaglia, V. Caleffi, and A. Valiani. Modeling blood flow in viscoelastic vessels: the 1D augmented fluid–structure interaction system. *Computer Methods in Applied Mechanics and Engineering*, 360(C):112772, 2020.

[3] G. Bertaglia, A. Navas-Montilla, A. Valiani, M. I. Monge García, J. Murillo, and V. Caleffi. Computational hemodynamics in arteries with the one-dimensional augmented fluid-structure interaction system: viscoelastic parameters estimation and comparison with in-vivo data. *Journal of Biomechanics*, 100(C):109595, 2020.

[4] M. Petrella, S. Tokareva, and E. F. Toro. Uncertainty quantification methodology for hyperbolic systems with application to blood flow in arteries. *Journal of Computational Physics*, 386:405–427, 2019.

3D cell-centered Finite Volume schemes for solving updated Lagrangian hyperelasticity on unstructured grids

W. Boscheri[†], **R. Loubère**[†] **and P.-H. Maire**[††]
[†] Dipartimento di Matematica e Informatica, University of Ferrara, Italy
(walter.boscheri@unife.it)
[‡] Institut de Mathématiques de Bordeaux (IMB) , Talence, France
(raphael.loubere@u-bordeaux.fr)
[††] CEA-CESTA, Le Barp, France (pierre-henri.maire@cea.fr)

ABSTRACT

In this talk, we present a conservative cell-centered Lagrangian Finite Volume scheme for solving the hyperelasticity equations on unstructured multidimensional grids. The starting point of the present approach is the cell-centered FV discretization named EUCCLHYD and introduced in the context of Lagrangian hydrodynamics. Here, it is combined with the *a posteriori* Multidimensional Optimal Order Detection (MOOD) limiting strategy to ensure robustness and stability at shock waves with piecewise linear spatial reconstruction. The ADER (Arbitrary high order schemes using DERivatives) approach is adopted to obtain second-order of accuracy in time. This approach has been tested in an hydrodynamics context and the present work aims at extending it to the case of hyperelasticity. Here, the hyperelasticty equations are written in the updated Lagrangian framework and the dedicated Lagrangian numerical scheme is derived in terms of nodal solver, Geometrical Conservation Law (GCL) compliance, subcell forces and compatible discretization. The Lagrangian numerical method is implemented in 3D under MPI parallelization framework allowing to handle genuinely large meshes. A relative large set of numerical test cases is presented to assess the ability of the method to achieve effective second order of accuracy on smooth flows, maintaining an essentially non-oscillatory behavior and general robustness across discontinuities and ensuring at least physical admissibility of the solution where appropriate. Pure elastic neo-Hookean and non-linear materials are considered for our benchmark test problems in 2D and 3D. These test cases feature material bending, impact, compression, non-linear deformation and further bouncing/detaching motions.

Keywords: Cell-centered Lagrangian finite volume schemes; moving unstructured meshes; *a posteriori* MOOD limiting; ADER; hyper-elasticity .

References

[1] W. Boscheri, R. Loubère and P.-H. Maire. "A 3D cell-centered ADER MOOD Finite Volume method for solving updated Lagrangian hyperelasticity on unstructured grids", *Journal of Computational Physics*, (submitted).

Thermodynamically compatible schemes for continuum mechanics

S. Busto†, **M. Dumbser**†, **I. Peshkov**† **and E. Romenski**‡
* Department of Applied Mathematics to Industrial Engineering, Universidad Politécnica de Madrid (`saray.busto@unitn.it`)
† Department of Civil, Environmental and Mechanical Engineering, University of Trento (`michael.dumbser@unitn.it, ilya.peshkov@unitn.it`)
‡ Sobolev Institute of Mathematics (`evrom@math.nsc.ru`)

ABSTRACT

In this talk we present a novel family of thermodynamically compatible finite volume schemes (HTC) for the first order hyperbolic Godunov-Peshkov-Romenski model of continuum mechanics, [2]. The proposed methodology directly solves the entropy inequality, while the total energy conservation law, which does not need to be approximated, is obtained as a consequence of all other discretized equations. We start by considering only the Euler subsystem, for which the Godunov formalism, put forward in [1] in the continuous framework, can be exactly mimicked at the discrete level [3]. Then, all other terms in the GPR equations are carefully discretized to achieve a discrete thermodynamic compatibility. As a result, the proposed schemes are provably marginally stable in the energy norm and satisfy a discrete entropy inequality by construction. The proposed methodology is carefully assessed both in the fluid and solid limits of the model [4].

Keywords: Thermodynamically compatible finite volume schemes; discrete Godunov formalism; Entropy inequality; Hyperbolic thermodynamically compatible PDE systems; Overdetermined hyperbolic PDE systems; Unified model for solid mechanics and fluid mechanics.

References

[1] S.K. Godunov. "An interesting class of quasilinear systems", *Computers and Fluids*, 139(3), . S.K. Godunov. An interesting class of quasilinear systems. Dokl. Akad. Nauk SSSR, 139(3), 521–523, 1961.

[2] I. Peshkov and E. Romenski. "Hyperbolic model for viscous Newtonian flows", *Continuum Mechanics Thermodynamics*, 28, pp. 85–104, 2014.

[3] S. Busto, M. Dumbser, S. Gavrilyuk and K. Ivanova. "On thermodynamically compatible finite volume methods and path-conservative ADER discontinuous Galerkin schemes for turbulent shallow water flows", *Journal of Scientific Computing*, 88, pp. 28, 2021.

[4] S. Busto, M. Dumbser, I. Peshkov and E. Romenski. "On thermodynamically compatible finite volume schemes for continuum mechanics", *Submitted.*

The Authors acknowledge financial support received from MIUR (Italy) in the frame of the Departments of Excellence Initiative 2018–2022 and in the frame of PRIN 2017 . S.B. was also funded by INdAM via a GNCS grant for young researchers and by an *UniTN starting grant* of the UNITN. E.R., M.D. and I.P. were supported by the Mathematical Center in Akademgorodok with the Ministry of Science and Higher Education of the Russian Federation.

An exactly well-balanced semi-implicit Lagrange-projection type scheme for the shallow-water system

C. Caballero-Cárdenas[†], **M. J. Castro**[†],
T. Morales de Luna[‡] **and M.L. Muñoz-Ruiz**[†]
[†] Dpto. Análisis Matemático, Estadística e Investigación Operativa y Matemática Aplicada,
Universidad de Málaga (`celiacaba@uma.es, mjcastro@uma.es, mlmunoz@uma.es`)
[‡] Dpto. Matemáticas, Universidad de Córdoba (`tomas.morales@uco.es`)

ABSTRACT

In this work we present a numerical approximation of the shallow water equations based on a Lagrange-projection type finite volume scheme.

The Lagrangian-projection scheme can be interpreted as a two-step algorithm consisting in first solving the shallow water system in Lagrangian coordinates, which is known as the Lagrangian step, and then projecting the results in Eulerian coordinates, which is known as the Projection step. This strategy allows us to decouple the acoustic and transport phenomena and to design implicit-explicit and large time step schemes in which the CFL restriction is based on the slower transport waves and not on the acoustic ones. In this work we follow the strategy described in [1, 3] to define the LP scheme and [2] to ensure its well-balanced character.

For the Lagrangian step we propose two explicit versions, one first order and another one second order of accuracy, and also another two implicit versions, again first and second order. The projection on the Eulerian coordinates will always be done explicitly, preserving the total order of the scheme. Special care is done for ensuring the well-balanced properties of the scheme.

Keywords: Lagrange-projection schemes; Shallow-water system; Semi-implicit schemes; Well-balanced schemes.

References

[1] Manuel J. Castro, Christophe Chalons, and Tomás Morales de Luna. "A Fully Well-Balanced Lagrange–Projection-Type Scheme for the Shallow-Water Equations", *SIAM Journal on Numerical Analysis*, 56(5), pp. 3071–3098, 2018.

[2] Manuel J. Castro and C. Parés. "Well-Balanced High-Order Finite Volume Methodsfor Systems of Balance Laws", *Journal of Scientific Computing*, 82, 2020.

[3] Tomás Morales De Luna, Manuel J. Castro, and Christophe Chalons. "High-order fully well-balanced Lagrange-Projection scheme for shallow water", *Communications in Mathematical Sciences*, 18, pp. 781–807, 2020.

This research has been partially supported by the Spanish Government and FEDER through the coordinated Research project RTI2018-096064-B-C1 and RTI2018-096064-B-C2, the Junta de Andalucía research project P18-RT-3163 and the Junta de Andalucía-FEDER-University of Málaga Research project UMA18-FEDERJA-161. C. Caballero-Cárdenas is also supported by a Grant from "Ministerio de Ciencia e Innovación", Spain (FPI2019/087773).

ADER scheme for incompressible Navier-Stokes equations on overset grids with a compact transmission condition

M.G. Carlino†, **M. Bergmann**†, **A. Iollo**† **and H. Telib***
† INRIA Bordeaux Sud-Ouest, MEMPHIS team, ({michele-giuliano.carlino, michel.bergmann, angelo.iollo}@inria.fr)
* Optimad Engineering srl (haysam.telib@optimad.it)

ABSTRACT

The simulation of flows in complex geometries or fluid-structure interaction, with freely moving objects, requires specific numerical modelling. Our investigations focus on Chimera grids. They consist of multiple overlapping mesh blocks that together define an overset grid. Usually, one has a background mesh that includes one or more foreground mesh patches that are fitted to the physical domain boundaries. This mesh generation approach considerably simplifies the task of mesh adaptation in the case of boundary layers, changing geometry for an unsteady problem and for unsteady multiply connected domains. Once the multiple mesh patches are generated, they are collated in order to obtain an appropriate overlapping zone between the neighboring blocks. In the overlapping zones, the exchange of solution information from one grid to another is performed. A compact transmission condition is generally sought in order to limit communications between the grids. In this framework, we propose a space-time Finite Volume scheme on Chimera grids for the incompressible Navier-Stokes equations. The scheme follows a classical fractional step method by Chorin-Temam with a second order accuracy in space and time. The ADER method [1, 2] provides an ideal setting for the resolution of the nonlinear unsteady convection-diffusion equation with a moving grid. The numerical scheme treats the temporal variable indistinctly with respect to the spatial variables by defining the solution on a space-time slab. Thus, instead of time-dependent spatial transmission conditions between relatively moving grid blocks, we define interpolation polynomials on arbitrarily intersecting space-time cells at the block boundaries. The ADER method is extended for overset grids with a new Local Lax-Friederichs stabilization approach for the computation of fluxes [3]. For the resolution of the Poisson equation for the pressure, we propose a hybrid FV method [4]. On internal cells, a classical reconstruction of the gradient through the diamond formula is employed. On fringe cells, the reconstruction of the gradient is performed by interpolating the information on the local stencil through a smart minimization of the L^{∞}-norm of local coefficients by exploiting of the geometry of the stencil.

Keywords: Chimera mesh; Overset grid; Compact transmission condition; Incompressible Navier-Stokes.

References

[1] V.A. Titarev and E. Toro, Ader schemes for three dimensional non-linear hyperbolic systems, *Journal of Computational Physics* 204 (2) (2005) 715-736.

[2] M. Dumbser, W. Boscheri, M. Semplice and G. Russo. Central weighted eno schemes for hyperbolic conservation laws on fixed and moving unstructured meshes, *SIAM Journal on Scientific Computing*, 39 (6)(2017) A2564–A2591.

[3] M. Bergmann, M.G. Carlino and A. Iollo. econd order scheme for unsteady advection-diffusion on moving overset grids with a compact transmission condition, *SIAM Journal on Scientific Computing*, (Submitted).

[4] A. Raeli, M. Bergmann and A. Iollo, A finite-difference method for the variable coefficient poisson equation on hierarchical cartesian meshes, *Journal of Computational Physics* 355 (2018) 59-77.

High-Order Splitting Schemes for the Shallow Water Equations with applications to tsunami wave propagation

C. E. Castro[§], **D. Vanzo**[‡], **A. Siviglia**[†] **and E. F. Toro**[†]
[§] Departamento de Ingeniería Mecánica, Facultad de Ingeniería, Universidad de Tarapacá, Chile (ccastro@academicos.uta.cl)
[‡] Laboratory of Hydraulics, Hydrology and Glaciology VAW, ETH Zurich, Switzerland (vanzo@vaw.baug.ethz.ch)
[†] Laboratory of Applied Mathematics, DICAM, University of Trento, Italy (eleuterio.toro@unitn.it, annunziato.siviglia@unitn.it)

ABSTRACT

We present a splitting method for the shallow water equations in one and two space dimensions following the splitting framework of Toro and Vázquez [1]. The technique splits the full system into two subsystems, one called the advection system and another called the pressure system. The numerical scheme is then constructed from fluxes from each of the subsystems. The resulting first-order schemes turn out to be exceedingly simple, with accuracy and robustness comparable to that of the sophisticated Godunov method. The basic methodology constitutes the building block for the construction of numerical schemes of very high order of accuracy following the ADER approach in one and two space dimensions as presented in [2]. The practical applicability of the schemes is illustrated through applications like Riemann problems, stationary shocks and simulation of tsunami wave propagation in the Pacific Ocean.

Keywords: Hyperbolic equations; Finite volume; Shallow water equations; Numerical flux; Splitting; Advection system; Pressure system; ADER schemes.

References

[1] E. F. Toro and M. E. Vázquez-Cendón. "Flux splitting schemes for the Euler equations", *Computers and Fluids*, 70, pp. 1–12, 2012.

[2] Castro, Cristóbal E and Toro, Eleuterio F and Käser, Martin "ADER scheme on unstructured meshes for shallow water: simulation of tsunami waves", *Geophysical Journal International*,189(3), pp. 1505–1520, 2012.

The Authors acknowledge support by Proyecto Mayor UTA 8718-16, Universidad de Tarapacá.

Artificial viscosity to get both robustness and discrete entropy inequalities

C. Berthon[†], **M. J. Castro**[‡], **A. Duran**[◇], **T. Morales de Luna**[*], **K. Saleh**[◇]

[†] Laboratoire de Mathématiques Jean Leray, Université de Nantes, France. (christophe.berthon@univ-nantes.fr)

[‡] Departamento de Análisis Matemático, Estadística e Investigación Operativa, y Matemática Aplicada, Universidad de Málaga, Spain. (mjcastro@uma.es)

[◇] Institut Camille Jordan, CNRS UMR 5208, Université Claude Bernard, Villeurbanne. France. (arnaud.duran@univ-lyon1.fr, saleh@math.univ-lyon1.fr)

[⋆] Departamento de Matemáticas, Universidad de Córdoba. Córdoba, Spain.(tomas.morales@uco.es)

ABSTRACT

This work concerns with the stability of numerical schemes to approximate the weak solutions of hyperbolic systems of the form:

$$\partial_t w + \partial_x f(w) + A(w)\partial_x w = 0, \qquad x \in \mathbb{R}, \quad t > 0, \quad \omega \in \Omega \subset \mathbb{R}^n. \tag{1}$$

In addition, we assume the system (1) to be endowed with an entropy inequality given by

$$\partial_t \eta(w) + \partial_x G(w) \leq 0, \qquad \text{in the weak sense,} \tag{2}$$

where the entropy function η is convex and the entropy flux function G is defined by $\nabla_w G(w) = \nabla_w \eta(w) \cdot (\nabla_w f(w) + A(w))$. In the present work, we focus our attention on the 3-point finite volume explicit schemes:

$$\begin{aligned} w_i^{n+1} = w_i^n &- \frac{\Delta t}{\Delta x}\left(f_\Delta(w_i^n, w_{i+1}^n) - f_\Delta(w_{i-1}^n, w_i^n)\right) \\ &- \frac{\Delta t}{2\Delta x}\left(A_\Delta^L(w_i^n, w_{i+1}^n)\cdot(w_{i+1}^n - w_i^n) + A_\Delta^R(w_{i-1}^n, w_i^n)\cdot(w_i^n - w_{i-1}^n)\right), \end{aligned} \tag{3}$$

where f_Δ denotes a consistent numerical flux and $A_\Delta^{L,R}$ is a matrix consistent with $A(\cdot)$. Here, w_i^n approximates $w(x,t)$ for all x in a cell $(x_{i-1/2}, x_{i+1/2})$ of size Δx at time t^n. For the sake of simplicity, both Δx and Δt are constant. Now, three natural questions arise: How is the time increment Δt restricted? Do the updated states $(w_i^{n+1})_{i\in\mathbb{Z}}$ be in Ω as soon as $w_i^n \in \Omega$ for all $i \in \mathbb{Z}$? How can we restore, at the numerical level, an entropy inequality (2)? In the present work, we address the three asked questions by adopting artificial viscosity approach for any given 3-point FV first order scheme.

Keywords: finite volume schemes, robustness, discrete entropy inequalities.

Manuel J. Castro acknowledges financial support from the Spanish Government and FEDER through the coordinated Research project RTI2018-096064-B-C21 and the Andalusian Government Research projects UMA18-FEDERJA-161 and P18-RT-3163. Arnaud Duran acknowledges financial support from the French National Research Agency project NABUCO, grant ANR-17-CE40-0025 and from the French National program INSU-CNRS (Institut National des Sciences de l'Univers - Centre National de la Recherche Scientifique) program LEFE-MANU (Les Enveloppes Fluides et l'Environnement - Méthodes Mathématiques et Numériques), project DWAVE. Tomás Morales acknowledges financial support from the Spanish Government and FEDER through the coordinated Research project RTI2018-096064-B-C22.

AENO: a novel reconstruction method in conjunction with ADER schemes for hyperbolic equations

M. Celant[†], **A. Santacá**[†], **G. Montecinos**[§], **L.O. Müller**[†] **and E.F. Toro**[‡]

[†] Mathematics Department, University of Trento, Italy (`morena.celant@unitn.it`, `andrea.santaca@gmail.com`, `lucas.muller@unitn.it`)

[‡] Laboratory of Applied Mathematics, DICAM, University of Trento, Italy (`eleuterio.toro@unitn.it`)

[†] [*] Department of Natural Sciences and Technology, Universidad de Aysén, Obispo Vielmo 62, Coyhaique, Chile (`gmont000@gmail.com`)

ABSTRACT

We propose a novel spatial reconstruction method that is akin to both ENO and WENO. The method, called **AENO**, results from averaging two polynomials, the classical ENO polynomial and its closest neighbour, while the search for the stencil remains commanded by ENO. A variant of the scheme, called **m-AENO**, results from averaging the modified ENO polynomial of Shu [1] and its closest neighbour. Here, the reconstruction scheme is applied in conjunction with the fully discrete high-order ADER approach. Schemes of up to 7-th order of accuracy in space and time are implemented for the linear advection equation [2, 3] and then schemes of up to 5-th order are tested for a non-linear hyperbolic system [2], namely the blood flow equations. For both problems we first carry out experiments to compare numerical solutions with exact solutions for short and long evolution times. Results for five reconstruction methods are compared, namely ENO, modified ENO, WENO and the novel AENO and modified AENO. Then we carry out a convergence rate study for both types of problems, the linear advection equation and the blood flow equations. Overall, the results of the new AENO reconstruction methods are comparable to the ENO, m-ENO and WENO. AENO shows a distinctive advantage over ENO for long-time evolution problems; this is more obvious for second and third-order methods, but will also be apparent for high-order methods on coarse meshes. Crucially, AENO turns out to be upto one order of magnitude more eficient than WENO, for a chosen test problem. Our results show that the L_1-errors of the novel AENO/m-AENO approach are the smallest for most cases considered.

Keywords: Hyperbolic equations; High-order ADER; ENO/WENO reconstruction; Novel reconstruction technique AENO.

References

[1] C.W. Shu. "Numerical Experiments on the Accuracy of ENO and Modified ENO Schemes", *Journal of Scientific Computing*, 5, pp. 127–149, 1990.

[2] E.F. Toro, A. Santacá, G. Montecinos, M. Celant and L.O. Müller. "AENO: a novel reconstruction method in conjunction with ADER schemes for hyperbolic equations", *Communications on Applied Mathematics and Computation (CAMC)* (submitted).

[3] G. Montecinos, A. Santacá, M. Celant, L.O. Müller and E.F. Toro. "ADER scheme with a simplified solver for the generalized Riemann problem and novel reconstruction procedure. Application to blood flow in a network", (in preparation).

Well-Balancing Via Flux Globalization: Applications to Shallow Water Equations with Wet/Dry Fronts

Alina Chertock[†]
[†] Department of Mathematics, North Carolina State University (chertock@math.ncsu.edu)

ABSTRACT

We study the flux globalization based central-upwind scheme from [2] for the Saint-Venant system of shallow water equations. We first show that while the scheme is capable of preserving moving-water equilibria, it fails to preserve much simpler "lake-at-rest" steady states. We then modify the computation of the global flux variable and develop a well-balanced scheme, which can accurately handle both still- and moving-water equilibria. In addition, we extend the flux globalization based central-upwind scheme to the case when "dry" and/or "almost dry" areas are present. To this end, we introduce a hybrid approach: we use the flux globalization based scheme inside the "wet" areas only, while elsewhere we apply the central-upwind scheme from [1], which is designed to accurately capture wet/dry fronts. We illustrate the performance of the proposed schemes on a number of numerical examples.

Joint work with Alexander Kurganov, Xin Liu, Yongle Liu and Tong Wu.

Keywords: Flux globalization; central-upwind schemes; well-balanced schemes; "lake-at-rest" steady states; "dry lake" steady states.

References

[1] A. Bollermann, G. Chen, A. Kurganov, and S. Noelle. "A well-balanced reconstruction of wet/dry fronts for the shallow water equations, *J. Sci. Comput.*, 56, pp. 267–290, 2013.

[2] Y. Cheng, A. Chertock, M. Herty, A. Kurganov, and T. Wu. "A new approach for designing moving-water equilibria preserving schemes for the shallow water equations", *J. Sci. Comput.*, 80, pp. 538–554, 2019.

Tips and tricks for simple Hyperbolic Viscous Flow

S. Chiocchetti[†], **I. Peshkov**[†] **and M. Dumbser**[†]
[†] Laboratory of Applied Mathematics, DICAM, University of Trento
(`simone.chiocchetti@unitn.it, ilya.peshkov@unitn.it, michael.dumbser@unitn.it`)

ABSTRACT

Several recent works [2, 3, 4] have been devoted to the development and numerical solution of a unified model for continuum mechanics (often termed GPR) that can describe elasto-plastic solids and viscous or inviscid fluid motion with the same set of hyperbolic PDEs.

Additional physical effects, conventionally modelled by means of equations involving second derivatives, such as Fourier's law of heat conduction [2] or surface tension forces [1], can also be reformulated (from first principles) as first-order-hyperbolic PDEs and used to extend the capabilities of the above mentioned unified model of continuum mechanics.

Such first order systems often include stiff nonlinear relaxation sources that can prove challenging to solve efficiently, since they mandate a locally implicit time discretisation.

In this talk we propose a simple strategy for efficiently solving the evolution equations of the inverse deformation gradient $\mathbf{A}$ (a 3×3 nonsymmetric matrix) in presence of the *arbitrarily stiff* nonlinear source terms governing relaxation of the associated strain tensor.

Finally, we will showcase the capabilities of the presented hyperbolic reformulation of continuum mechanics with computational examples applied to two-phase viscous flow with surface tension, which also represents a complex benchmark for the treatment of involutive constraints that can be shown to be satisfied by the continuum equations, but that must be actively enforced by choosing a suitable discretisation.

Keywords: First order Hyperbolic reductions; Curl involution constraints; Stiff relaxation sources; Asymptotic preserving schemes; Surface tension

References

[1] S. Chiocchetti, I. Peshkov, S. Gavrilyuk, M. Dumbser. "High order ADER schemes and GLM curl cleaning for a first order hyperbolic formulation of compressible flow with surface tension". *Journal of Computational Physics*, 2021.

[2] M. Dumbser, I. Peshkov, E. Romenski, O. Zanotti. "High order ADER schemes for a unified first order hyperbolic formulation of continuum mechanics: Viscous heat-conducting fluids and elastic solids", *Journal of Computational Physics*, 2016.

[3] A.A. Gabriel, D.Li, S. Chiocchetti, M. Tavelli, I. Peshkov, E. Romenski, M. Dumbser. "A unified first order hyperbolic model for nonlinear dynamic rupture processes in diffuse fracture zones". *Philosophical Transactions of the Royal Society A*, 2021.

[4] I. Peshkov, M. Dumbser, W. Boscheri, E. Romenski, S. Chiocchetti, M. Ioriatti. "Modeling solid-fluid transformations in non-Newtonian viscoplastic flows with a unified flow theory". *Computers and Fluids*, 2021.

S.C. acknowledges the financial support received by the Deutsche Forschungsgemeinschaft (DFG) under the project *Droplet Interaction Technologies (DROPIT)*, grant no. GRK 2160/1 and of the HPC-Europa3 project, under European Union's Horizon 2020 research and innovation programme, grant no. 730897.

A computational multi-physics approach for nuclear fusion reactor simulations

M. Chrysanthou†, A. Farmakalides†, S. T. Millmore†, N. Nikiforakis†

† Laboratory for Scientific Computing, Cambridge (`mc2173@cam.ac.uk, aaf37@cam.ac.uk`)

ABSTRACT

This work is concerned with the implementation of computational multiphysics algorithms for the whole-system simulation of nuclear fusion reactors. The objective is to account for all regions of the reactor (plasma, vacuum and containment vessel) within the same simulation and in a Cartesian frame of reference, which represents a significant departure from current (segregated solutions on physics-driven mesh alignment) approaches on both counts. A key element of this methodology is the discretisation of topologically complex rigid boundaries as well as material and state of matter interfaces. The former implies mesh generation in the conventional CFD sense, while the latter is discretisation on both sides of the material or matter interface. In this presentation we will discuss the implementation of sharp and diffuse interface finite volume methods for the solution of the magnetohydrodynamic equations for transient case studies. To this end, we employ a ghost-fluid method, previously used for simulations involving the interaction of a plasma arc with elastoplastic substrates (lightning strike on aerospace composites) [1], and a novel diffuse interface method based on flux-modifiers and interface seeding routines [2]. The latter is shown to perform as well as sharp interface methods, without any of their shortfalls.

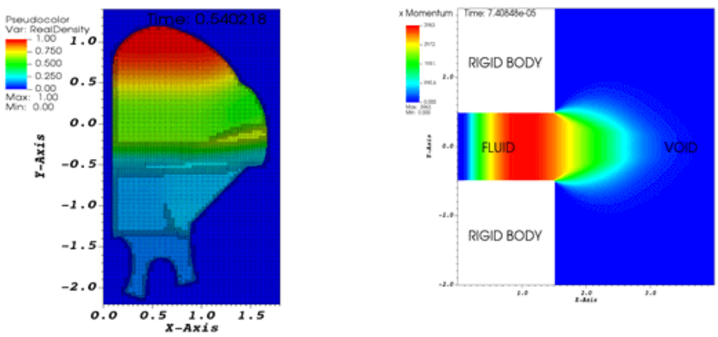


Figure 1: Left: An MHD Riemann problem in a rigid-boundary vessel discretised by means of ghost fluid method. Right: Interaction of plasma with void and a rigid wall using a diffuse interface method.

Keywords: magnetohydrodynamics, multi-material interactions, level-set methods, diffuse boundaries, sharp boundaries, rigid bodies, vacuum.

References

[1] Michael, L., Millmore, S.T. and Nikiforakis N. (2019). A multi-physics methodology for four-states of matter. Communications on Applied Mathematics and Computation, https://doi.org/10.1007/s42967-019-00047-4

[2] Wallis. T., Barton, P.T. and Nikiforakis, N. (2020). A Flux-enriched Godunov Method for Multi-material Problems with Interface Slide and Void Opening. Journal of Computational Physics, 110499 https://doi.org/10.1016/j.jcp.2021.110499

Strategies for time integration in fluid/structures interaction problems within dynamic-IB methods

A. Coclite[†] **and T. Politi**[†]
[†] Dipartimento di Ingegneria Elettrica e dell'Informazione (DEI),
Politenico di Bari, Via Re David 200 – 70125 Bari, Italy
(alessandro.coclite@poliba.it, tiziano.politi@poliba.it)

ABSTRACT

The aim of the present work is to critically compare explicit and implicit schemes for time integration for structural dynamics in fluid/structure interaction problems. Specifically, this systematic analysis is carried out by measuring their sustainability in term of the established compromise between accuracy and computational burden. This study is composed by three steps; firstly, the selected schemes are compared by measuring their abilities in reproducing the analytical solution describing the motion of a mass responding to a elastic potential; then, such schemes are challenged computing the dynamics of deforming structures under prescribed loads; finally, explicit and implicit schemes for time integration are considered by computing the settling orbit of a particle in a lid-driven cavity as a function of the Reynolds number. The transport of deformable objects is also considered and the deformation pattern of a membrane under shear for different mechanical stiffness is assessed. Here, the dynamic-Immersed–Boundary method combined with a BGK-Lattice–Boltzmann technique is developed for discussing fluid/structure interaction problems. The fluid evolution is obtained on a three-dimensional lattice with 19 reticular velocities (D3Q19 computational molecule) while the immersed body surface is modeled as a collection of Lagrangian points responding to an elastic potential and a bending resistance. A moving least squares reconstruction is used to accurately interpolate flow quantities and the forcing field needed to enforce the boundary condition on immersed bodies [1, 2].

Keywords: Immersed Boundary; Moving Least Squares; Coupling Strategies; Deforming Particles; Strong Coupling.

References

[1] A. Coclite, S. Ranaldo, M.D. de Tullio, P. Decuzzi, and G. Pascazio. Kinematic and dynamic forcing strategies for predicting the transport of inertial capsules via a combined lattice Boltzmann immersed boundary method. *Computers & Fluids*, 180 2019.

[2] A. Coclite, S. Ranaldo, G. Pascazio, and M. D. de Tullio. A lattice boltzmann dynamic-immersed boundary scheme for the transport of deformable inertial capsules in low-re flows. *Computers & Mathematics with Applications*, 80 (12) 2020.

A. C. acknowledges the project "Research for Innovation" (REFIN) - POR Puglia FESR FSE 2014-2020 - Asse X - Azione 10.4 (Grant No. CUP - D94120001410008). A. C. and T. P. are members of Gruppo Nazionale per il Calcolo Scientifico (GNCS) of Istituto Nazionale di Alta Matematica (INdAM).

An efficient implementation of turbulent-diffusive processes and suspended sediment transport in shallow-water models: hyperbolization, flux splitting approach and GPU acceleration

D. Conde[†], **D. Vanzo**[†], **A. Siviglia**[‡] **and D. F. Vetsch**[†]
[†] Laboratory of Hydraulics, Hydrology and Glaciology. ETH, Swiss Federal Institute of Technology, Zurich, Switzerland
(conde@vaw.baug.ethz.ch, vanzo@vaw.baug.ethz.ch, vetsch@vaw.baug.ethz.ch)
[‡] Dept. of Civil, Environmental and Mechanical Engineering. University of Trento, Trento, Italy (annunziato.siviglia@unitn.it)

ABSTRACT

The BASEMENT v3 software [1] is a freely available GPU-accelerated river modelling suite developed at VAW, ETH Zurich, and is widely adopted in engineering practice and research. The hydro-morphodynamics core of BASEMENT is supported by the shallow-water and Exner equations, respectively, with explicit first-order HLLC-type schemes implemented on unstructured finite-volume meshes. The full parallelization on GPUs allows a ten- to hundred-fold performance increase, allowing larger and more detailed computational solutions for increasingly complex scenarios.

In this work we introduce multiple new features coming to BASEMENT v3, with particular emphasis on turbulent-diffusive phenomena and suspended sediment transport. Following the approach developed by Vanzo et al. [2], we reformulate the system of governing equations as hyperbolic balance laws with stiff source terms and then we use a splitting approach for the numerical solution. The main novelty in this work is the generalization of the approach to simultaneous multiple diffusive processes, such as pollutant transport of multiple passive scalars, depth-averaged turbulence modelling and suspended sediment transport. We present a set of benchmarks with experimental data, for the most commonly used closure models in depth-averaged turbulence and suspended sediment modelling.

The achieved computational and numerical performances confirm that the hyperbolization of the mathematical model used in combination with a numerical splitting approach result in very efficient numerical simulations and that BASEMENT v3 provides a suitable platform to simulate complex river flows.

Keywords: Finite-volume; Unstructured meshes; Advection-diffusion; Relaxation schemes; Flux splitting; 2DH Turbulence; Suspended sediment transport; GPU-CUDA;

References

[1] D. Vanzo, S. Peter, L. Vonwiller, M. Burgler, M. Weberndorfer, A. Siviglia, D. Conde and D. F. Vetsch, “Basement v3: a modular freeware for river process modelling over multiple computational backends”, Environmental Modelling & Software, 105102, 2021.

[2] D. Vanzo, A. Siviglia and E. F. Toro “Pollutant transport by shallow water equations on unstructured meshes: Hyperbolization of the model and numerical solution via a novel flux splitting scheme”, Journal of Computational Physics, 321, pp. 1-20, 2016.

The development of the software BASEMENT is financially supported by the Swiss Federal Office for the Environment (FOEN).

Numerical evolution of the resistive relativistic magnetohydrodynamic equations: a minimally implicit Runge-Kutta scheme

I. Cordero-Carrión[†], **S. Santos-Pérez**[†] **and C. Martínez-Vidallach**
[†] Departmento de Matemáticas, University of Valencia (Spain) (isabel.cordero@uv.es, samuel.santos@uv.es, clamarv2@alumni.uv.es)

ABSTRACT

I will present the theoretical development of Minimally-Implicit Runge-Kutta (MIRK) methods for the numerical evolution of the resistive relativistic magnetohydrodynamic equations [1]. Previous methods rely on Implicit-Explicit (IMEX) Runge-Kutta schemes [2] and need to apply the recovery of the primitive variables from the conserved variables in numerous addionational times. This recovery can potentially have convergence problems due to the use of iterative methods. Moreover, the computational cost of the previous IMEX methods in comparison with explicit ones is much higher. The MIRK methods reduce the number of recoveries needed and can be easily adapted from explicit schemes.

Keywords: Relativistic magnetohydrodynamic equations; Stiff source terms; Finite-Diferences meshes; Implicit Runge-Kutta methods.

References

[1] S.S. Komissarov. "Multidimensional numerical scheme for resistive relativistic magnetohydrodynamics", *Monthly Notices of the Royal Astronomical Society*, 382, 995–1004, 2007.

[2] C. Palenzuela, L. Lehner, O. Reula and L. Rezzolla. "Beyond ideal MHD: towards a more realistic modelling of relativistic astrohpysical plasmas", *Monthly Notices of the Royal Astronomical Society*, 394, 1727–1740, 2009.

The Authors acknowledge support by the Spanish Grant No. PGC2018-095984-B-I00 (Agencia Estatal de Investigación / Ministerio de Ciencia, Innovación y Universidades), Grant No. PROMETEO/2019/071 (Generalitat Valenciana) and Grant No. ACIF2019/169 (European Social Fund).

Macroscopic and Multi-Scale Models for Multi-Class Vehicular Dynamics with Uneven Space Occupancy: A Case Study

M. Briani†, **E. Cristiani**† **and P. Ranut**‡
† Istituto per le Applicazioni del Calcolo, CNR (m.briani@iac.cnr.it, e.cristiani@iac.cnr.it)
‡ Autovie Venete S.p.A.

ABSTRACT

In this talk we propose two models based on ODEs and hyperbolic PDEs describing the dynamics of heavy and light vehicles on a road network, taking into account the interactions between the two classes. The models are tailored for two-lane highways where heavy vehicles cannot overtake. In these conditions the creeping phenomenon can appear, i.e. one class of vehicles can proceed even if the other class has reached the maximal density.

The first model we propose is macroscopic (fluid-dynamic) and couples two first-order hyperbolic PDEs with phase transition. The second model is instead multi-scale and couples a system of second-order ODEs with a first-order hyperbolic PDE.

Numerical results show that both models are able to catch some second-order (inertial) phenomena like stop & go waves.

Models are calibrated by means of real data measured by fixed sensors placed along the A4 Italian highway Trieste-Venice and its branches, provided by Autovie Venete S.p.A.

Keywords: traffic models; LWR model; follow-the-leader model; phase transition; creeping; seepage; networks.

On second-order well-balanced Lagrange-projection schemes for shallow water Exner system

A. Del Grosso[†] **and C. Chalons**[†]
[†] Laboratoire de Mathématiques de Versailles, Université de Versailles Saint-Quentin-en-Yvelines (alessia.del-grosso2@uvsq.fr, christophe.chalons@uvsq.fr)

ABSTRACT

In this work we aimed to develop and implement second-order well-balanced Lagrange-projection numerical methods for the shallow water Exner system. The Lagrange-Projection formalism entails a decomposition of the mathematical model into two different systems: the acoustic (or Lagrangian) one, which takes into account (fast) acoustic waves, and the transport (or projection) step based on the (slow) transport waves [3]. Furthermore, we are interested in the well-balanced property of the numerical method, namely the ability of the scheme of preserving the stationary solutions of the model [3].

When it comes to the choice of the mathematical model, here we consider the well-known shallow water system coupled with the so-called Exner equation. The former has been widely used to describe the evolution in time of fluid flows for instance in rivers or coastal areas. While the Exner equation simulates the bedload sediment transport due to the mechanical action of the flow [1]. In particular, the Grass model is taken into account to model the solid transport discharge contributions.

It is known that it is not a trivial task to numerically simulate the resulting shallow water Exner model, as a decoupled method could lead to the presence of spurious oscillations in the numerical outputs [2]. Moreover, considering the Lagrangian-projection formalism, it is not clear how to take into account the solid transport discharge contributions and, thus, how to split the equation. For this reason we explored different possible approaches. The numerical strategies have been extended to second-order of accuracy as well.

Keywords: Lagrange-projection splitting; Shallow water equations; Exner equation; Well-balanced property; Approximate Riemann solver; Second-order of accuracy.

References

[1] M. J. Castro Díaz, E. D. Fernández-Nieto and A.M. Ferreiro. Sediment transport models in Shallow Water equations and numerical approach by high order finite volume methods. *Computers & Fluids*, Vol. 37, Issue 3, Pages 299-316, 2008.

[2] S. Cordier, M. Le, T. Morales de Luna. Bedload transport in shallow water models: why splitting (may) fail, how hyperbolicity (can) help. *Advances in Water Resources*, 34(8): 980–989, 2011.

[3] T. Morales De Luna, M. J. Castro Díaz and C. Chalons. High order fully well-balanced Lagrange-Projection scheme for Shallow-water. *Communications in Mathematical Sciences*, Vol. 18, No. 3, pp. 781–807, 2020.

Reacting condensed phase explosives in direct contact

R. Dematté[†]**, L. Michael**[†] **and N. Nikiforakis**[†]
[†] Laboratory for Scientific Computing, Cambridge (rd609@cam.ac.uk)

ABSTRACT

In this work we present a new formulation and an associated algorithm for the simultaneous numerical simulation of condensed phase explosives in direct contact with each other, which may also be confined by (or interacting with one or more) compliant inert materials. Examples include composite rate-stick (i.e. involving two explosives in contact) problems and interaction of shock waves with chemically-active particles in condensed-phase explosives. There are several formulations which address the compliant or structural response of confiners and particles due to detonations, but the direct interaction of explosives remains a challenge for most formulations and algorithms. The proposed formulation addresses this problem by extending the conservation laws and the mixture rules of an existing hybrid augmented Euler/multi-phase model [1]. An algorithm for the solution of the resulting system of partial differential equations is presented, which includes a new robust method for the retrieval of the densities of the constituents of each explosive mixture. The algorithm is implemented in a hierarchical adaptive mesh refinement framework and validated against results from problems with known solutions. It is shown that the method can simulate the interaction of detonation waves produced by military grade and commercial explosives in direct contact, each with its own distinct equation of state and reaction rate law. The ability of the new model to simulate reactive particles which are explicitly resolved in a heterogeneous explosive is demonstrated by a case-study of a shock wave interacting with a high explosive bead embedded in liquid nitromethane.

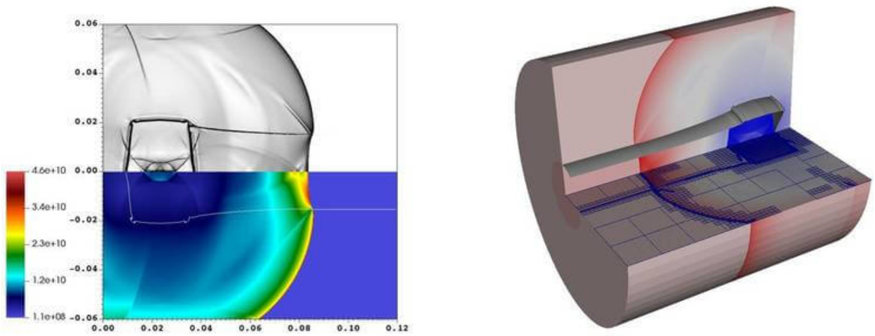


Figure 1: Numerical solutions for a four equation of state rate stick problem [2]. The figure illustrates the pressure field and density-based mock Schlieren plot (left) as well as the three-dimensional density field and the distribution of AMR grids (right).

Keywords: condensed phase explosives, hybrid augmented Euler/multi-phase model, hierarchical adaptive mesh refinement.

References

[1] L. Michael and N. Nikiforakis. "A hybrid formulation for the numerical simulation of condensed phase explosives", *J. of Computational Physics*, 316, pp. 193–217, 2016.

[2] R. Dematte. "Multi-physics modelling of multiple interacting reactive states of matter", *MPhil. Thesis, Dept. of Physics, University of Cambridge, UK.*

The implicit Lagrangian Riemann problem: how and why?

S. Delpino[‡], B. Després[†] and A. Plessier[‡†]
[†] LJLL-Sorbonne university-France
[‡] CEA-France

ABSTRACT

The implicit solution of non linear equations that arise in gas dynamics problems has always aroused interested since the seminal works of Godunov [4]. Indeed implicitation seems an efficient way to address some severe stability issues which are common to this class of problems. However, as always with non linear equations, existence and uniqueness of the solutions are not guaranteed.

Since then, the importance of the topics has been recognized by the community. It is sufficient to review the contributions [1, 2, 3] of E. T. Toro to be convinced of that fact. So far, implicitation produces interesting results, even though a solid mathematical basis still misses.

In the context of the PhD of Alexiane Plessier, we decided to work on the mathematical and numerical structure of implicit Riemann solvers for Lagrangian equations. We discovered that one particular version is endowed with strong convex structure. On this basis we proved existence and uniqueness of the implicit solution, which seems to be an original result. Moreover we discovered that the scheme is extremely efficient for tracking the position of the contact discontinuity, uniformly with respect to the time step $\Delta t > 0$ which can be taken very large. An explanation will be provided based on exact integration of the implicit Riemann Lagrangian equations. A preprint will soon be available.

Keywords: implicit Lagrangian Riemann problem, implicit solvers.

References

[1] Montecinos-Toro. Reformulations for general advection-diffusion reaction equations and locally implicit ADER schemes, JCP 2014.

[2] Toro-Montecinos. Implicit, semi-analytical solution of the generalized Riemann problem for stiff hyperbolic balance laws, JCP 2015.

[3] Contarino-Montecinos-Borsche-Kall-Toro. Junction-Generalized Riemann Problem for Stiff Hyperbolic Balance Laws in Networks: an Implicit Solver and ADER Schemes. JCP 2016.

[4] Godunov et al, Numerical Solving Many-Dimensional Problems of Gas Dynamics, Nauka, 1976

A hyperbolic augmented model for the Nonlinear Schrödinger equation

F. Dhaouadi[†], **N. Favrie**[‡], **S. Gavrilyuk**[‡] **and J-P. Vila**[⋆]
[†] Laboratory of Applied Mathematics, DICAM, University of Trento, Italy.
(firas.dhaouadi@unitn.it)
[‡] Aix-Marseille Université, UMR CNRS 7343, IUSTI, France.
[⋆] Institut de Mathématiques de Toulouse, INSA de Toulouse, France.

ABSTRACT

We present an approximate first-order hyperbolic model for the hydrodynamic form of the defocusing nonlinear Schrödinger equation (NLS). This Euler-Korteweg type system can be seen as an Euler-Lagrange equation to a Lagrangian submitted to a mass conservation constraint. Due to the presence of dispersive terms, the latter depends explicitly on the gradient of density. The idea is to create a new dummy variable that accurately approximates the density via a penalty method. Then, we take its gradient as a new independent variable and apply Hamilton's principle to rederive the system.

We explain the main ideas behind the method, how the resulting system is hyperbolic, and present some numerical results for gray solitons and dispersive shockwaves, obtained using second-order accurate TVD schemes.

Keywords: First order Hyperbolic reductions; Finite volume schemes; Euler-Korteweg equations; Dispersive Hyperbolic equations;

References

[1] Dhaouadi, F., Favrie, N., and Gavrilyuk, S. "Extended Lagrangian approach for the defocusing nonlinear Schrödinger equation", *Studies in Applied Mathematics*, 142(3), 336-358, 2019.

[2] Dhaouadi, F. "An augmented lagrangian approach for Euler-Korteweg type equations". Diss. *Université de Toulouse, Université Toulouse III-Paul Sabatier*, 2020.

High order finite volume schemes with IMEX time stepping for the Boltzmann model on unstructured meshes

G. Dimarco[†]

[†] Department of Mathematics and Computer Science First &
Center for Modeling, Computing and Statistics, CMCS
University of Ferrara, Italy. (giacomo.dimarco@unife.it)

ABSTRACT

In this talk, we present a family of time and space high order finite volume schemes for the solution of the full Boltzmann equation. The velocity space is approximated by using a discrete ordinate approach while the collisional integral is approximated by spectral methods. The space reconstruction is implemented by integrating the distribution function, describing the state of the system, over arbitrarily shaped and closed control volumes using a Central Weighted ENO (CWENO) technique. The full discretization is then obtained by combining the previous phase-space approximation with high order Implicit-Explicit (IMEX) Runge Kutta schemes. Comparisons of the Boltzmann model with simpler relaxation type kinetic models (like BGK) are proposed showing the capability of the Boltzmann equation to capture different physical solutions. The methods are also tested on several standard two-dimensional benchmark problems in comparison with Direct Simulation Monte Carlo results.

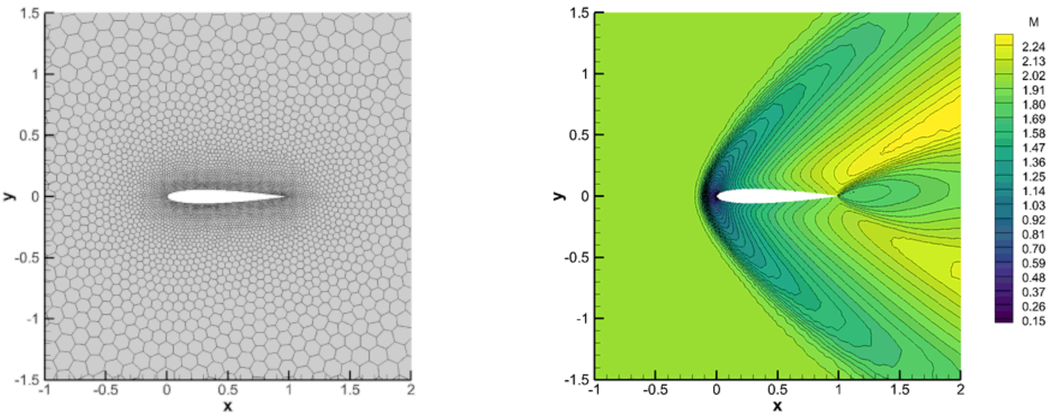


Figure 1: Supersonic flow around NACA 0012 airfoil in a rarefied regime. Mesh and Mach number contours.

Uncertainty reduction in turbulent simulations using high-order methods

I. W. Kokkinakis[†], **D. Drikakis**[‡]
[†] University of Strathclyde, Glasgow, G1 1XJ, UK
[‡] University of Nicosia, Nicosia, CY-2417, Cyprus

ABSTRACT

The paper presents an overview of recent developments in turbulence using implicit Large Eddy Simulations and high-order methods. We demonstrate through several examples the impact of the design of the numerical methods on the accuracy of (compressible) turbulent flows. Prototypical examples include the Taylor Green vortex, Richtmyer-Meshkov mixing, shock-boundary-layer interaction, and sudden expansion flows. We discuss how an understanding of the numerical dissipation and properties of the non-linear numerical schemes, in general, can lead to reduced modelling and simulation uncertainty. The above has an impact on several applications, including high-speed, compressible, and chemically reacting flows. As an indicative example, Figure 1 shows the complex vortex ring structure captured by a high-resolution 9th-order simulation at a two-gas planar interface shortly after re-shock.

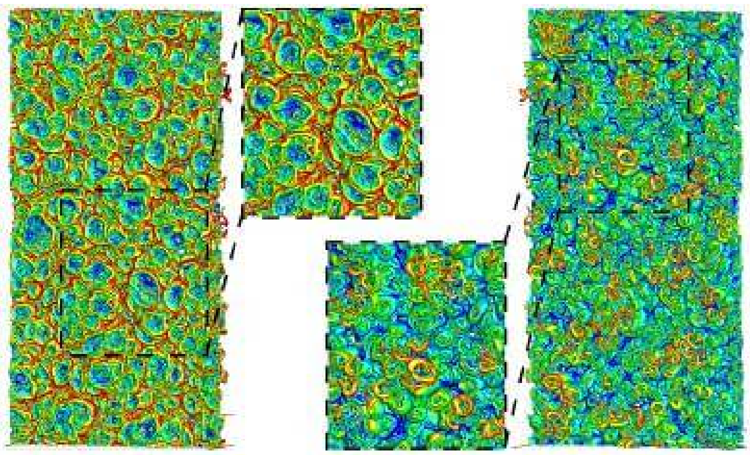


Figure 1: Vortex rings of compressible turbulent mixing flows using 9th-order methods.

Keywords: Turbulence; high-order methods.

References

[1] I. W. Kokkinakis, D. Drikakis, D. L. Youngs. "Vortex morphology in Richtmyer–Meshkov-induced turbulent mixing". *Physica D*, 407-132459, 2020.

[2] K. Ritos, I. Kokkinakis, D. Drikakis. "Physical insight into the accuracy of finely resolved iLES in turbulent boundary layers", *Computers and Fluids*, 169, 309-316, 2018.

High order ADER discontinuous Galerkin schemes for nonlinear hyperelasticity with material failure

M. Dumbser[†], **I. Peshkov**[†], **E. Romenski**[‡], **M. Tavelli**[†], **S. Chiocchetti**[†]
[†] Department of Civil, Environmental and Mechanical Engineering,
University of Trento, Italy (michael.dumbser@unitn.it)
[‡] Sobolev Institute of Mathematics, Novosibirsk, Russia,

ABSTRACT

In this talk we present high order discontinuous Galerkin finite element schemes with a posteriori subcell finite volume limiter on space-time adaptive Cartesian meshes (AMR) for the solution of the unified first order hyperbolic formulation of continuum mechanics of Godunov, Peshkov and Romenski (GPR model) that is able to describe simultaneously nonlinear elasto-plastic solids at large strain, as well as viscous Newtonian and non-Newtonian fluids. The GPR model can describe nonlinear dynamic rupture processes and even material fatigue by adding an additional scalar advection-reacting equation to the governing PDE system. The model is thermodynamically compatible and obeys the first and second law of thermodynamics. A key feature of the model is the use of a twofold diffuse interface approach that allows the cracks to form anywhere and at any time, independently of the chosen computational grid. This is substantially different from many fracture modeling approaches that need to resolve discontinuities explicitly, such as for example dynamic shear rupture models used in computational seismology, where the geometry of the rupture fault needs to prescribed a priori. We show extensive numerical comparisons with experimental results for stress-strain diagrams of different real materials and for the generation and propagation of fracture in rocks and pyrex glass at low and high velocities. Overall, a very good agreement between numerical simulations and experiments is obtained.

Keywords: First order hyperbolic and thermodynamically compatible formulation of continuum mechanics; high order Discontinuous Galerkin schemes; *a posteriori* subcell finite volume limiter; large-strain elasto-plasticity; material failure and cracks.

References

[1] E.I. Romenski. “Hyperbolic systems of thermodynamically compatible conservation laws in continuum mechanics”, *Math. Comput. Modell.*, 28, pp. 115–130, 1998.

[2] M. Tavelli, E. Romenski, S. Chiocchetti, A. Gabriel and M. Dumbser. “Space-time adaptive ADER discontinuous Galerkin schemes for nonlinear hyperelasticity with material failure”, *Journal of Computational Physics*, 422, pp. 109758, 2020.

[3] A. Gabriel *et al.* “ A unified first order hyperbolic model for nonlinear dynamic rupture processes in complex diffuse fracture zones”, *Philosophical Transactions of the Royal Society A*, 379, pp. 20200130, 2021.

[4] S. Busto, S. Chiocchetti, M. Dumbser, E. Gaburro and I. Peshkov. “High order ADER schemes for continuum mechanics”, *Frontiers in Physics*, 8, pp. 32, 2020.

The Authors acknowledge financial support from the Italian Ministry of University, Education and Research (MIUR) in the frame of the PRIN 2017 project and of the European Commission in the frame of the ExaHyPE project.

Evaluation of the performance of two non-hydrostatic shallow water models for the simulation of steady and unsteady flows

I. Echeverribar†‡, **P. Brufau**† **and P. García-Navarro**†
† Fluid Mechanics, I3A-EINA, University of Zaragoza (`echeverribar@unizar.es`, `brufau@unizar.es`, `pigar@unizar.es`)
‡ Hydronia Europe, S.L.

ABSTRACT

The simulation of steady states in non-hydrostatic pressure (NHP) systems does not always lead to a physically based solution, presenting static oscillations. The smulation of discontinuos unsteady cases also present a challenge for these NHP models. Two diferent systems for NHP free surface flows are analyzed in this work. First, a system (a) considering mass and x- and z-momentum conservation equations, combined with a divergence free condition that gives the system a hyperbolic-elliptic nature, is solved by an explicit-implicit finite-volume method [2]. Secondly, a hyperbolic relaxation of the system [1] by means of artificial compressibility, in exchange for the presence of some parameters on the equations, is solved by a finite-volume Roe Solver (model (b)). The research shows that an equilibrium must be found between the robustness of the (a) model and the potential efficiency of the (b) model.

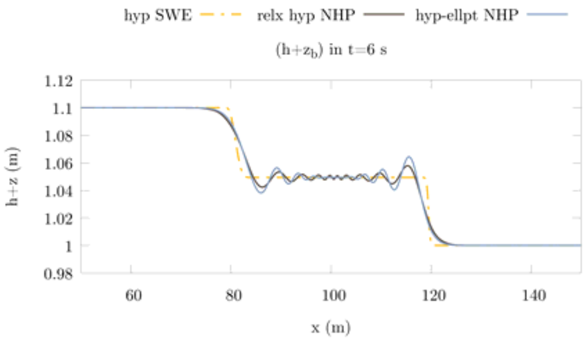


Figure 1: Comparison between a hydrostatic model (SWE), and NHP models (a) and (b) for a dam break simulation.

Keywords: Non-hydrostatic Pressure Model; Hyperbolic-Elliptic; Hyperbolic relaxation; Roe Solver; Well-balanced methods.

References

[1] Escalante, C. and Dumbser, M. and Castro, M. J. "An efficient hyperbolic relaxation system for dispersive non-hydrostatic water waves and its solution with high order discontinuous Galerkin schemes", *Journal of Computational Physics*, 394, pp. 385–416, 2019.

[2] Yamazaki, Y. and Kowalik, Z. and Cheung, K. F. "Depth-integrated, non-hydrostatic model for wave breaking and run-up", *International Journal for Numerical Methods in Fluids*, 61, pp. 473-497 (2009)

The authors would like to thank Hydronia Europe for their collaboration and support.

A tree structured method for high-dimensional evolutive Hamilton-Jacobi equations and applications

A. Alla[†]**, M. Falcone**[*] **and L. Saluzzi**[‡]
[†] Department of Mathematics, PUC, Rio de Janeiro ((alla@mat.puc-rio.br)
[*] Dipartimento di Matematica, Università di Roma "La Sapienza"
(falcone@mat.uniroma1.it)
[‡] Dipartimento di Matematica, Università di Roma "La Sapienza"
(luca.saluzzi@uniroma1.it)

ABSTRACT

The approximation of high-dimensional PDEs is a difficult task since many classical methods require a space discretization to work on a (structured or unstructured) grid. This is a bottleneck in real applications where a large number of state variables is needed even for modeling of the continuous problem. Optimal control for systems governed by ODEs or PDEs is one of the fields where this is particularly relevant because controlled dynamical systems can easily have more than 10 components in industrial applications (e.g. robotics). The Dynamic Programming (DP) approach to optimal control problems is based on the characterization of the value function as the unique viscosity solution of a Hamilton-Jacobi-Bellman (HJB) equation, so we need to solve a nonlinear PDE in high-dimension. DP (also called semi-Lagrangian) schemes are typically based on a time discretization that is projected on a fixed space triangulation of the numerical domain [2]. The projection on the grid requires a polynomial interpolation.

We present a new approach to solve the evolutive HJB equation related to finite horizon optimal control problems where we compute the value function on a tree structure generated by the time discrete dynamics avoiding the construction of a space grid to solve the HJB equation [1, 3]. This allows to drop the cost of the space interpolation, moreover the tree will guarantee a perfect matching with the discrete dynamics. A pruning technique is introduced to cut branches of the tree in order to reduce the computational cost. We prove first order convergence to the value function for a first order discretization of the dynamics and we discuss some extensions and applications.

Keywords: Lagrangian methods, Hamilton-Jacobi equations, optimal control, high-dimensional problems

References

[1] A. Alla, M. Falcone and L. Saluzzi. *An efficient DP algorithm on a tree-structure for finite horizon optimal control problems*, SIAM Journal on Scientific Computing, (41) 4, 2019, A2384-A2406

[2] M. Falcone, R. Ferretti, *Semi-Lagrangian Approximation Schemes for Linear and Hamilton-Jacobi Equations*, Society for Industrial and Applied Mathematics, Philadelphia, 2013.

[3] L. Saluzzi, A. Alla and M. Falcone. *Error estimates for a tree structure algorithm for dynamic programming equations*, submitted, 2018 https://arxiv.org/abs/1812.11194

A novel structure preserving semi-implicit finite volume method for viscous and resistive magnetohydrodynamics

Francesco Fambri[†]
[†] Max Planck Institute for Plasma Physics, Boltzmannstraße 2, Garching bei München, 85748, Germany (francesco.fambri@ipp.mpg.de)

ABSTRACT

In this work we introduce a novel semi-implicit structure-preserving finite-volume/finite-difference scheme for the viscous and resistive equations of magneto-hydrodynamics (VRMHD) based on an appropriate 3-split of the governing PDE system, which is decomposed into a first convective subsystem, a second subsystem involving the coupling of the velocity field with the magnetic field and a third subsystem involving the pressure-velocity coupling. The nonlinear convective terms are discretized explicitly, while the remaining two subsystems accounting for the Alfvén waves and the magneto-acoustic waves are treated implicitly. Thanks to this, the final algorithm is at least formally constrained only by a mild CFL stability condition depending on the velocity field of the pure hydrodynamic convection. To preserve the divergence-free constraint of the magnetic field exactly at the discrete level, a proper set of overlapping dual (staggered) meshes is employed. The resulting linear algebraic systems are shown to be symmetric and therefore can be solved by means of an efficient standard matrix-free conjugate gradient algorithm. The final scheme can be regarded as a novel shock-capturing, conservative and structure preserving semi-implicit scheme for VRMHD. Several numerical tests are presented to show the main features of our novel divergence-free semi-implicit FV/FD solver: linear-stability in the sense of Lyapunov is verified at a prescribed constant equilibrium solution; a second-order of convergence is obtained for a smooth time-dependent solution; shock-capturing capabilities are proven against a standard set of stringent MHD shock-problems; accuracy and robustness are verified against a non-trivial set of two- and three-dimensional MHD problems.

Keywords: semi-implicit; structure-preserving; divergence-free; finite-difference; finite-volume; conservative; shock-capturing; three-split; staggered grids; viscous and resistive MHD.

References

[1] F. Fambri “A novel structure preserving semi-implicit finite volume method for viscous and resistive magnetohydrodynamics”, (submitted), see arXiv:2012.11218.

[2] M. Dumbser, D. Balsara, M. Tavelli, F. Fambri “A divergence-free semi-implicit finite volume scheme for ideal, viscous, and resistive magnetohydrodynamics”, *International Journal for Numerical Methods in Fluids*, 89(1-2): 16-42, 2019.

[3] Dumbser M, Casulli V. “A conservative, weakly nonlinear semi-implicit finite volume method for the compressible Navier- Stokes equations with general equation of state”, *Applied Mathematics and Computation*, 272: 479–497, 2016.

A well balanced finite volume scheme for general relativity

E. Gaburro[*], **M. J. Castro**[†] **and M. Dumbser**[‡]
[*] Team CARDAMOM, Inria Bordeaux-Sud-Ouest, France (elena.gaburro@inria.fr)
[†] University of Málaga, Spain [‡] University of Trento, Italy

ABSTRACT

The aim of this talk is to present a novel second order accurate well balanced (WB) finite volume (FV) scheme for the solution of the general relativistic magnetohydrodynamics (GRMHD) equations and the first order CCZ4 formulation (FO-CCZ4) of the Einstein field equations of general relativity, as well as the fully coupled FO-CCZ4 + GRMHD system. These systems of *first order hyperbolic* PDEs allow to study the dynamics of the *matter* and the dynamics of the *space-time* according to the theory of general *relativity*.

The new well balanced finite volume scheme exploits the knowledge of an equilibrium solution of interest when integrating the conservative fluxes, the nonconservative products and the algebraic source terms, and also when performing the piecewise linear data reconstruction. This results in a rather simple modification of the underlying second order FV scheme, which, however, being able to cancel numerical errors committed with respect to the equilibrium component of the numerical solution, substantially improves the accuracy and long-time stability of the numerical scheme when simulating small perturbations of stationary equilibria. In particular, the need for well balanced techniques appears to be more and more crucial as the applications increase their complexity. We close the presentation with a series of numerical tests of increasing difficulty, where we study the evolution of small perturbations of accretion problems and of stable TOV neutron stars. Our results show that the well balancing significantly improves the long-time stability of the finite volume scheme compared to a standard scheme.

Keywords: First order hyperbolic systems, finite volume schemes (FV), well balanced schemes (WB), General Relativistic magnetohydrodynamics (GRMHD), First order conformal and covariant reformulation of the Einstein field equations (FO-CCZ4), Michel accretion disk, TOV neutron star.

References

[1] E. Gaburro, M.J. Castro and M. Dumbser. “A well balanced finite volume scheme for general relativity”, *SIAM Journal on Scientific Computing*, submitted.

[2] E. Gaburro. “A Unified Framework for the Solution of Hyperbolic PDE Systems Using High Order Direct Arbitrary-Lagrangian–Eulerian Schemes on Moving Unstructured Meshes with Topology Change”, *Archives of Computational Methods in Engineering*, 28(3), pp. 1249–1321, 2021.

[3] M. Dumbser, F. Fambri, E. Gaburro, A. Reinarz. “On GLM curl cleaning for a first order reduction of the CCZ4 formulation of the Einstein field equations”, *Journal of Computational Physics*, 404, 109088, 2020.

Acknowledgment. E. Gaburro gratefully acknowledges the support received from the European Union’s Horizon 2020 research and innovation programme under the Marie Skłodowska-Curie grant agreement *SuPerMan* (No 101025563).

On Compatible Legendre-Gauss-Lobatto Subcell Low Order Finite Volume Methods

(and what we can do with it)

Gregor Gassner[†] **and Andrés M. Rueda-Ramírez**[†]

[†] Department of Mathematics and Computer Science, Center for Data and Simulation Science, University of Cologne (ggassner@uni-koeln.de, arueda@uni-koeln.de)

ABSTRACT

In this talk, we explain how to construct a compatible subcell finite volume scheme that can be seamlessly blended with a high order spectral element discontinuous Galerkin method. Our goal applications are the compressible Euler equations and the ideal MHD equations in multiple dimensions. Starting with an entropy-dissipative split-form discontinuous Galerkin scheme on collocated Legendre-Gauss-Lobatto (LGL) nodes, we show how to carefully design a finite volume type discretization on the LGL subcell grid such that it is (i) still provably entropy-dissipative/stable/conservative and (ii) compatible to the high order DG scheme in the sense for each element, we can seamlessly blend between the low order finite volume and the high order DG discretization. This new hybrid scheme enables us to do shock capturing and preserve positivity of the solution: the straight forward idea is to use the arbitrary blending in the hybrid scheme smartly such that the amount of FV is high in regions that need a lot of stabilization (e.g. shocks), whereas its amount is low in regions where the DG scheme can stay alive without much help. We will present some applications in hydrodynamics and magnetohydrodynamics, e.g. Figure 1.

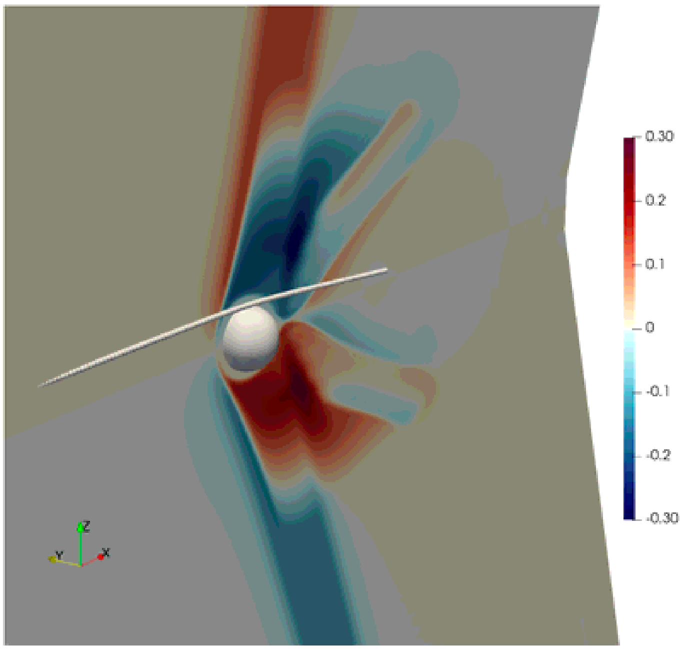


Figure 1: Simulation of Jupiter moon Io and the interaction with its plasma torus. Plot shows the z-component of velocity and the Galileo spacecraft trajectory used for comparison.

Keywords: discontinuous spectral element, Legendre-Gauss-Lobatto subcell finite volume, shock capturing, entropy-stability, positivity

Singular solutions of the BBM equation: analytical and numerical study

Sergey Gavrilyuk[†], **Keh-Ming Shyue**[‡]
[†] Aix Marseille Univ, CNRS, IUSTI, UMR 7343, Marseille, France,
(sergey.gavrilyuk@univ-amu.fr)
[‡] Institute of Applied Mathematical Sciences, National Taiwan University, Taipei 106, Taiwan,
(shyue@ntu.edu.tw)

ABSTRACT

We show that the Benjamin-Bona-Mahony (BBM) equation admits stable travelling wave solutions representing a sharp transition front linking a constant state with a periodic wave train. The constant state is determined by the parameters of the periodic wave train : the wave length, amplitude and phase velocity, and satisfies both the Rankine-Hugoniot conditions for the corresponding Whitham modulation system and generalized Rankine-Hugoniot conditions for the exact BBM equation. Such stable shock-like travelling structures exist if the phase velocity of the periodic wave train is not less than the periodic solution mean value. To validate the accuracy of the numerical method, we derive the (singular) solitary limit of the Whitham system for the BBM equation and compare the corresponding numerical and analytical solutions. We find good agreement between analytical results and numerical solutions.

Hypocoercivity of Stochastic Galerkin Formulations for Stabilization of Kinetic Equations

Stephan Gerster[†], **Michael Herty**[†] **and Hui Yu**[‡]
[†] RWTH Aachen University, Germany (gerster@igpm.rwth-aachen.de)
[‡] Tsinghua University, China

ABSTRACT

We consider the stabilization of linear kinetic equations with a random relaxation term. The framework of hypocoercivity by J. Dolbeault, C. Mouhot and C. Schmeiser (2015) ensures the stability in the deterministic case. This framework, however, cannot be applied directly for arbitrarily small random relaxation parameters. Therefore, we introduce a Galerkin formulation, which reformulates the stochastic system as a sequence of deterministic ones. We prove for the γ-distribution that the hypocoercivity framework ensures the stability of this series and hence the stochastic stability of the underlying random kinetic equation. The presented approach also yields a convergent numerical approximation.

Keywords: Systems of kinetic and hyperbolic balance laws; exponential stability; asymptotic stability; stochastic Galerkin

References

[1] Jean Dolbeault, Clément Mouhot and Christian Schmeiser, *Hypocoercivity for linear kinetic equations conserving mass*, Transactions of the American Math. Society, 2015

[2] Stephan Gerster, Michael Herty and Hui Yu, *Hypocoercivity of stochastic Galerkin formulations for stabilization of kinetic equations*, Communications in Mathematical Sciences, 2021

This work is supported by DFG 320021702/GRK2326, DFG HE5386/18,19, BMBF ENet 05M18PAA, NSFC 11901339 and NSFC 11971258. We would like to offer special thanks to Giuseppe Visconti, Tabea Tscherpel and to the support from RWTH Aachen-Tsinghua Senior Research Fellowships.

Blood flow simulations in hybrid 1D-0D networks based on *a priori* model selection criteria

B. Ghitti[†], **P.J. Blanco**[‡], **E.F. Toro**[*] **and L.O. Müller**[†]
[†] Department of Mathematics, University of Trento, Italy (`beatrice.ghitti@unitn.it`, `lucas.muller@unitn.it`)
[‡] National Laboratory for Scientific Computing LNCC, Petrópolis, Brazil (`pjblanco@lncc.br`)
[*] Laboratory of Applied Mathematics DICAM, University of Trento, Italy (`eleuterio.toro@unitn.it`)

ABSTRACT

To address the issue of computational cost related to the modelling of blood flow in complex networks, we have developed a new strategy to construct hybrid 1D-0D networks based on *a priori* model selection criteria. Departing from the first-order hyperbolic system of PDEs describing 1D blood flow, we have derived a family of lumped-parameter models preserving important nonlinear properties of the original 1D model [2]. A high-order coupling procedure based on conservation principles has been introduced, which allows to solve fully 1D, fully 0D and hybrid 1D-0D junctions, and to separately evolve each vessel converging at a junction using either a high-order finite volume method, if 1D, or an explicit high-order ODE numerical scheme, if 0D. To construct hybrid networks including coupled 1D and 0D vessels, we have experimented different *a priori* model selection criteria, such as intravascular volume, vessel radius and compliance, which allow to determine, given a network, the most suitable model to be used for each vessel. The objective is to identify relevant criteria and corresponding 1D-0D thresholds, that provide the best trade-off between computational cost of the simulations and good level of accuracy in the predicted results, which have to reproduce the main characteristic features of pressure and flow waveforms in all vessels.
In this contribution, we will describe the methods and discuss the results obtained by applying this new adaptive model selection strategy to different networks of vessels, such as the 37-artery network reported in [3] and the reduced ADAN56 model reported in [1].

Keywords: First-order hyperbolic PDEs; First-order ODEs; Reduced-order blood flow models; High-order couplings; Hybrid 1D-0D networks; Model selection criteria.

References

[1] E. Boileau et al., "A benchmark study of numerical schemes for one-dimensional arterial blood flow modelling", *Int. J. Numer. Meth. Biomed. Eng.*, 31(10), pp. e02732, 2015.

[2] B. Ghitti, E.F.Toro and L.O. Müller, "Nonlinear lumped-parameter models for blood flow simulations in networks of vessels", *ESAIM: M2AN*, (submitted).

[3] K.S. Matthys et al., "Pulse wave propagation in a model human arterial network: assessment of 1-D numerical simulations against in-vitro measurements", *J. of Biomechanics*, 40(15), pp. 3476–3486, 2007.

Collocation methods for high-order well-balanced methods for 1D systems of balance laws

I. Gómez-Bueno[†], **M.J. Castro**[†] **C. Parés**[†] **and G. Russo**[‡]
[†] Dpto. Análisis Matemático, Estadística e Investigación Operativa y Matemática Aplicada, Universidad de Málaga (igomezbueno@uma.es, mjcastro@uma.es, pares@uma.es)
[‡] Dipartimento di Matematica e Informatica, University of Catania (russo@dmi.unict.it)

ABSTRACT

The goal of this work is to develop high-order well-balanced schemes for systems of balance laws. In some previous works, two of the authors introduced a general technique to design high-order numerical methods for one-dimensional balance laws that preserve all their stationary solutions. The key ingredient of these methods is a well-balanced reconstruction operator. A strategy has been also introduced to modify any standard reconstruction operator like MUSCL, ENO, CWENO, etc. in order to be well-balanced. This strategy involves a non-linear problem at every cell, at every time step, for every conserved variable, that consists in finding the stationary solution whose average is the given cell value. In the recent paper [1] a fully well-balanced method is presented where the nonlinear problems to be solved in the reconstruction procedure are interpreted as control problems: they consist in finding a solution of an ODE system whose average at the computation interval is given. These problems are written in functional form and the gradient of the functional is computed on the basis of the adjoint problem. The Newton's method and the RK4 method are applied then to solve the problems. Our goal now is to present another general implementation of this technique that can be applied to any one-dimensional balance laws based on the application of the collocation RK methods. Special care is put to analyze the effects of computing the averages and the source terms using quadrature formulas. A general technique which allows us to deal with resonant problems is also introduced. To check the efficiency of the methods and their well-balancedness, they have been applied to a number of tests, ranging from easy academic systems of balance laws consisting of Burgers equation with some nonlinear source terms to the shallow water equations -without and with Manning friction- or Euler equations of gas dynamics with gravity effects.

Keywords: systems of balance laws; well-balanced methods; finite volume methods; high order methods; reconstruction operators; collocation methods.

References

[1] Gómez-Bueno, I., Castro, M.J., Parés, C. “High-order well-balanced methods for systems of balance laws: a control-based approach.”, *Applied Mathematics and Computation*, 394, 125820, 2021.

Comparison of High-Resolution Reconstruction Schemes in Unsteady Low Mach Number Flows

M. Groom† **and B. Thornber**†
† School of Aerospace, Mechanical and Mechatronic Engineering, The University of Sydney
(michael.groom@sydney.edu.au, ben.thornber@sydney.edu.au)

ABSTRACT

This paper presents a comparison of four different high-resolution upwind numerical methods, consisting of two different reconstruction schemes with and without an additional correction used to improve accuracy at low Mach number. The two reconstruction schemes analysed are the fifth-order total variation diminishing (TVD) scheme of Kim & Kim [1] and the fifth-order targeted essentialy non-oscillatory (TENO) scheme of Fu et al. [2], both of which are implemented in a structured finite-volume code using the method of lines along with the low Mach correction of Thornber et al. [3]. The relative performance of the four numerical methods is assessed using a modified low Mach version of the canonical isentropic vortex test case, demonstrating that all of the benefits that come from using a low Mach correction in conjunction with a TVD reconstruction scheme also apply to newer ENO-based reconstruction schemes such as TENO. Furthermore, in certain settings a TVD scheme with a low Mach correction may produce results of comparable accuracy to those from a TENO scheme without a low Mach correction.

Keywords: High order reconstruction techniques; Finite volume schemes on structured meshes.

References

[1] Kim, K. H. and Kim, C., Accurate, efficient and monotonic numerical methods for multi-dimensional compressible flows Part II: Multi-dimensional limiting process, *J. Comput. Phys.*, pp. 570–615, 2005.

[2] Fu, L., Hu, X. Y. and Adams, N. A., A family of high-order targeted ENO schemes for compressible-fluid simulations, *J. Comput. Phys.*, pp. 333–359, 2016.

[3] Thornber, B., Mosedale, A., Drikakis, D., Youngs, D. and Williams, R., An Improved Reconstruction Method for Compressible Flows with Low Mach Number Features, *J. Comput. Phys.*, pp. 487–4894, 2008.

BGK models for gas mixtures: asymptotics and numerics

M. Groppi[†], **S. Boscarino**[‡], **S.Y. Cho**[‡] **and G. Russo**[‡]
[†] Department of Mathematical, Physical and Computer Sciences, University of Parma, Italy
(`maria.groppi@unipr.it`)
[‡] Department of Mathematics and Computer Science, University of Catania, Italy
(`boscarino@dmi.unict.it, chosy89@skku.edu, russo@dmi.unict.it`)

ABSTRACT

In this talk we overview consistent BGK models for inert mixtures of gases and present their numerical simulations using high order conservative semi-Lagrangian schemes. Relaxation-time approximations of BGK type constitute the most used simplified kinetic models of the true integrodifferential Boltzmann equations of Rarefied Gas Dynamics, since they retain the most significant mathematical and physical features of the Boltzmann description. Their extension to mixtures is not trivial since some inconsistencies can arise, like breakdown of positivity of density and temperature fields and of the indifferentiability principle. Here, three consistent BGK models [1, 2, 3] for inert mixtures of gases are compared, first in their kinetic behavior and then versus the hydrodynamic limits that can be derived in different collision-dominated regimes. In particular, the structure of the BGK model presented in [3] allows to deduce two different hydrodynamic limits (at Euler and Navier-Stokes level), characterized by global velocity and temperature or multi-velocity and multi-temperature, respectively. The comparison is carried out analytically and numerically, for the latter using an asymptotic preserving conservative semi-Lagrangian scheme for BGK models [4]. Because of accuracy and stability properties and exact conservation, the scheme is able to capture the underlying fluid-dynamic limits. Application to realistic binary mixtures of noble gases is also presented [5].

Keywords: Kinetic BGK models for mixtures; hydrodynamic limits; semi-Lagrangian methods.

References

[1] P. Andries, K. Aoki, and B. Perthame. "A consistent BGK-type model for gas mixtures", *Journal of Statistical Physics*, 106, pp. 993–1018, 2002.

[2] M. Groppi, G. Russo and G. Stracquadanio. "Semi-Lagrangian Approximation of BGK Models for Inert and Reactive Gas Mixtures". In: "From Particle Systems to Partial Differential Equations V" (P. Gonçalves and A. Soares Eds.), Springer Proceedings in Mathematics and Statistics, 258, pp. 53–80, 2018.

[3] A. V. Bobylev, M. Bisi, M. Groppi, G. Spiga, and I. F. Potapenko. "A general consistent BGK model for gas mixtures", *Kinetic and Related Models*, 11, pp. 1377-1393, 2018.

[4] S. Y. Cho, S. Boscarino, M. Groppi, and G. Russo. "Conservative semi-Lagrangian schemes for a general consistent BGK model for inert gas mixtures", arXiv preprint, arXiv:2012.0249 (submitted), 2020.

[5] S. Boscarino, S. Y. Cho, M. Groppi, and G. Russo. "BGK models for inert mixtures: comparison and applications", arXiv preprint, arXiv:2102.12757 (submitted), 2021.

Efficient compatible finite element solution of the time-dependent Maxwell equations on mapped grids

Yaman Güçlü† and Martin Campos Pinto†

† Max Planck Institute for Plasma Physics, Garching bei München, Germany

(yaman.guclu@ipp.mpg.de, martin.campos-pinto@ipp.mpg.de)

ABSTRACT

In many engineering applications, a diagonal or block-diagonal mass matrix is essential for efficient transient simulations based on the finite element method (FEM). To the authors' knowledge, the mass lumping techniques that are well established in the FEM community (see e.g. [1, 2]) cannot be directly applied to the theory of finite element exterior calculus (FEEC [3]), especially on mapped structured grids. We present two different approaches to overcome this limitation, with the time dependent Maxwell equations as an application.

The first approach is based on the construction of exact de Rham sequences on broken spaces [4], such that every component of the solution is discontinuous across the cell faces. In analogy with the discontinuous Galerkin method, to which it is closely related, this so-called "broken-FEEC" approach yields block-diagonal mass matrices that can be easily inverted. We present here some new theoretical results (proof of stability and error estimates) for the discretization of the Hodge-Laplace operator [5] on tensor-product spaces, as well as verification tests for a Maxwell source problem.

The second approach uses the conformal spaces typical of FEEC, but introduces appropriate discrete scalar products which also yield block-diagonal mass matrices [6]. This can be viewed as a high-order mass lumping technique for FEEC on non-Cartesian meshes. We present here some preliminary numerical results, where we investigate the robustness of this new scheme with respect to the mesh distortion.

Keywords: Finite element exterior calculus (FEEC); Broken-FEEC method; Mass lumping; Mapped structured grids; Finite element method (FEM); Maxwell's equations.

References

[1] G. Cohen and P. Monk, "Gauss point mass lumping schemes for Maxwell's equations", *Numerical Methods for Partial Differential Equations*, 14, pp. 63–88, 1998.

[2] A. Fisher, R. Rieben, G.H. Rodrigue, and D.A. White, "A Generalized Mass Lumping Technique for Vector Finite-Element Solutions of the Time-Dependent Maxwell Equations", *IEEE Trans. Antennas Propag.*, 53(9), pp. 2900–2910, 2005.

[3] D. Arnold, R. Falk, and R. Winther, "Finite element exterior calculus: from Hodge theory to numerical stability", *Bull. Amer. Math. Soc. (NS)*, 47, pp. 281–354, 2010.

[4] M. Campos Pinto, "Constructing exact sequences on non-conforming discrete spaces", *Comptes Rendus Mathematique*, 354, pp. 691–696, 2016.

[5] M. Campos Pinto and Y. Güçlü, "Broken-FEEC approximations of Hodge Laplace problems", *in preparation* (2021).

[6] M. Campos Pinto and Y. Güçlü, "High-order mass lumping for FEEC on mapped grids", *in preparation* (2021).

A unified surface-gradient and hydrostatic reconstruction scheme for the shallow water equations

Guoxian Chen†, **Sebastian Noelle**‡
† School of Mathematics and Statistics, Wuhan University, Wuhan, 430072, P.R. China
(gxchen.math@whu.edu.cn)
‡ Institute for Geometry and Practical Mathematics, RWTH Aachen University,
Templergraben 55, 52062, Aachen, Germany (noelle@igpm.rwth-aachen.de)

ABSTRACT

In this talk we present our new result on the numerical methods for the shallow water equations. A new second-order accurate hydrostatic reconstruction scheme is proposed. Such a scheme needs to overcome several difficulties: besides the well-known issues of positivity and well-balancing there is also the difficulty of unphysical reflections from bottom reconstructions which create artificial steps. We address all of these problems at once by changing the logic of the reconstruction of the bottom, the water depth and the water surface level. Notably, our bottom reconstruction is continuous across cell interfaces and remains unchanged during the computation, except if the original topography has a jump, or if a wet-dry front passes through a cell. Only in these exceptional cases we apply the new discontinuous bottom approximation and compute the residual via the subcell hydrostatic reconstruction method. The scheme gives excellent results in one and two space dimensions.

Keywords: shallow water equations; surface gradient method; hydrostatic reconstruction; well-balancing; positive preserving; unphysical reflection.

References

[1] E. Audusse, F. Bouchut, M.-O. Bristeau, R. Klein, and B. Perthame. “A fast and stable well-balanced scheme with hydrostatic reconstruction for shallow water flows”. *SIAM Journal on Scientific Computing*, 25 pp. 2050-2065, 2004.

[2] G. Chen, S. Noelle. “A new hydrostatic reconstruction scheme based on subcell reconstructions”, *SIAM Journal on Numerical Analysis*, 55 (2), 758–784, 2016.

[3] J. G. Zhou, D. M. Causon, C. G. Mingham, D. M. Ingram. “The surface gradient method for the treatment of source terms in the shallow-water equations”. *Journal of Computational Physics*, 168 (1) 1–25, 2001.

[4] A. Buttinger-Kreuzhuber, Z. Horváth, S. Noelle, G. Blöschl, J. Waser. “ A fast second-order shallow water scheme on two-dimensional structured grids over abrupt topography”. *Advances in Water Resources*, 127 89–108, 2019.

An ADER-WENO numerical scheme for a porous-medium mathematical model of atherosclerosis

A. Hidalgo[†] **and L. Tello**[‡]

[†] Dept. de Ingeniería Geológica y Minera. Escuela Técnica Superior de Ingenieros de Minas y Energía, Center for Computational Simulation. Universidad Politécnica de Madrid, Ríos Rosas, 21, 28003 Madrid, Spain (arturo.hidalgo@upm.es)

[‡] Dept. de Matemática Aplicada Escuela Técnica Superior de Arquitectura, Center for Computational Simulation, Universidad Politécnica de Madrid, Av. Juan de Herrera, 4, 28040 Madrid, Spain (l.tello@upm.es)

ABSTRACT

The aim of this work is the study and implementation of a local space-time DG ADER approach, in the finite volume framework, in the context of 2D porous media applications. This particular research concerns the first stages of atherosclerosis development, where the artery is considered as a porous medium. The mathematical model is given by a system of two-dimensional nonlinear reaction-diffusion equations with a nonlinear source term in one of the equations, see for instance [1, 2, 3] for details of the mathematical model under study. An interesting feature of this 2D model is that it incorporates a nonlinear non-homogeneous Neumann boundary condition representing the recruitment of immune cells through the upper boundary as a response to the production of cytokines. The numerical resolution is carried out under a finite volume approach, where spatial nonlinear reconstruction is achieved by means of a dimension-by-dimension WENO procedure, making use of entire polynomials, unlike the traditional pointwise WENO reconstruction. The evolution of the inflammation is studied taking into consideration two sets of bio-physical parameters and the *size* of the initial inflammation. Convergence rates of the numerical scheme are obtained. In addition to this, certain theoretical properties on the evolution solution, in accordance to hypothesis satisfied by the initial data, are stated and proved. Some of them are also verified with the numerical simulation carried out.

Keywords: ADER-WENO numerical scheme; High order reconstruction techniques; Finite volume schemes; Mathematical model of atherosclerosis; Porous-medium.

References

[1] N El Khatib, S Génieys, B Kazmierczak and V Volpert. "Mathematical modelling of atherosclerosis as an inflammatory disease", *Philos Trans A Math Phys Eng Sci.*, 367(1908), pp. :4877–86, 2009.

[2] A. Hidalgo and L.Tello. "Numerical simulation of a porous medium-type atherosclerosis initiation model", *Computers & Fluids*, 169, pp. 380–387, 2018.

[3] A. Hidalgo, L.Tello and E.F. Toro. "Numerical and analytical study of an atherosclerosis inflammatory disease model", *Journal of Mathematical Biology*, 68(7), pp. 1785–1814, 2014.

Discretization of a simple hyperbolic system rising in incompressible fluid-structure interaction

M. Bergmann[†], **A. Fondanèche**[†], **A. Iollo**[†]
[†] Institut de Mathématiques, Université de Bordeaux and Memphis team, Inria Bordeaux – Sud Ouest

ABSTRACT

We will investigate some natural upwind discretizations of the 1D system

$$\begin{cases} \partial_t u + u\,\partial_x u = \partial_x \sigma_e(Y), \\ \partial_t Y + u\,\partial_x Y = 0. \end{cases} \tag{4}$$

In the above equation we have that

$$\sigma_e(Y) = G\left[(\partial_x Y)^{-2} - 1\right]. \tag{5}$$

G is the shear modulus, Y is the inverse displacement mapping and u is the material velocity. This system is obtained starting from a fully Eulerian fluid-structure interaction problem in 2D.

It will appear that, although these 1D discretizations are equally consistent and stable, they lead to significantly different stability properties of the 2D full model discretization. An explanation of this phenomenon is given by an analysis in the limit of small deformations that allows to devise an effective scheme for the 2D problem.

Flexible Kernels for Particle-based Fluid Flow Simulations

A. Iske[†]
[†] Department of Mathematics, Universität Hamburg, Germany
(armin.iske@uni-hamburg.de)

ABSTRACT

Particle methods provide flexible discretizations for the numerical simulation of multiscale phenomena in fluid flows. For time-dependent evolution processes, for instance, particle models are particularly well-suited to cope with rapid variation of domain geometries and anisotropic large-scale deformations. Kernel-based approximations are suitable tools for the recovery step of particle simulations [2, 3]. Commonly used kernel approximation schemes are relying on positive definite radially symmetric functions, where Gaussians, inverse multiquadrics and compactly supported characteristic functions are prototypical examples [1]. Despite the good performance of these standard kernels, more flexible kernels are required to better capture anisotropic features in multiscale flows. To this end, we have developed anisotropic weighted kernels whose associated shape functions can better be aligned with preference directions of the flux. In this talk, we will explain the construction of this new class of kernel functions. Supporting numerical examples are provided.

Keywords: Particle-based fluid flow simulation; kernel-based multivariate approximation; anisotropic weighted kernels.

References

[1] A. Iske. *Approximation Theory and Algorithms for Data Analysis*. Texts in Applied Mathematics, volume 68, Springer, 2018.

[2] A. Iske, "Ten good reasons for using polyharmonic spline reconstruction in particle fluid flow simulations". In: *Continuum Mechanics, Applied Mathematics and Scientific Computing: Godunov's Legacy - A Liber Amicorum to Professor Godunov*, G.V. Demidenko, E. Romenski, E. Toro, M. Dumbser (eds.), Springer, 2020, 193–199.

[3] A. Iske. "On the construction of kernel-based adaptive particle methods in numerical flow simulation". In: *Recent Developments in the Numerics of Nonlinear Hyperbolic Conservation*, R. Ansorge, H. Bijl, A. Meister, T. Sonar (eds.), Notes on Numerical Fluid Mechanics and Multidisciplinary Design (NNFM), Springer, 2013, 197–221.

The authors acknowledges the support by the Deutsche Forschungsgemeinschaft (DFG) within the Research Training Group GRK 2583 "Modeling, Simulation and Optimization of Fluid Dynamic Applications".

Structure preserving numerical methods for the Euler equations with gravity

Christian Klingenberg†
† Department of Mathematics, Würzburg University, Germany
(klingen@mathematatik.uni-wuerzburg.de)

ABSTRACT

We consider the Euler equations with gravity.

First we present well-balanced numerical methods for which we can ensure robustness, accuracy and stability, since they satisfies the discrete entropy inequality, see e.g. [1].

Next we look at flow in the low Mach number regime. We present numerical methods that are asymptotic preserving in the low Mach number regime. The well-balancedness of the schemes presented are an important ingredient to make them work, see [2], [3]. We show astrophysical applications of this, see figure 1.

This is joint work among others with Wasilij Barsukow, Claudius Birke, Christophe Chalons, Praveen Chandrasekhar, Fritz Röpke.

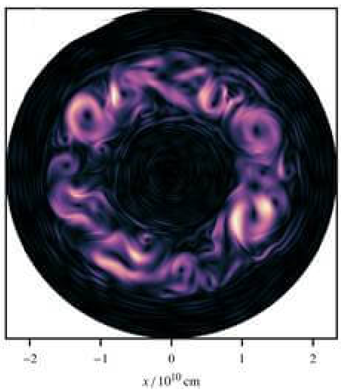


Figure 1: The convective shell in a star, simulated by a well-balanced method that is asymptotic preserving in the low Mach limit

Keywords: Euler with gravity, well-balanced, low Mach limit, Active Flux scheme, astrophysical applications

References

[1] Berberich, J., Chandrashekar, P., Klingenberg, C.:“High order well-balanced finite volume methods for multi-dimensional systems of hyperbolic balance laws, *Computer and Fluids*, Vol. 219 (2021)

[2] C. Birke, C. Chalons, C. Klingenberg: “A low Mach two speed relaxation scheme for the compressible Euler equations with gravity”, *manuscript* (2021)

[3] Wasilij Barsukow; Jonathan Hohm; Christian Klingenberg; Philip L Roe, “The active flux scheme on Cartesian grids and its low Mach number limit”, *Journal of Scientific Computing*, Vol. 81, (2019)

High-Order Path-Conservative Central-Upwind Schemes

A. Kurganov†
† Department of Mathematics, Southern University of Science and Technology
(alexander@sustech.edu.cn)

ABSTRACT

We are interested in developing robust and highly accurate numerical methods for nonlinear nonconservative hyperbolic systems of PDEs. It is well-known that the presence of the nonconservative products introduces several challenges related to the fact that weak solutions of nonconservative systems cannot be understood in the sense of distributions. Instead, one can introduce week solutions of such system as the Borel measures as it was done in [1]. This concept of week solutions was utilized to develop path-conservative finite-volume upwind schemes (see, e.g., a recent review [2] and reference therein) and Riemann-problem-solver-free path-conservative central-upwind (PCCU) schemes [3].

However, the PCCU schemes from [3] are only second-order accurate, which limits their accuracy and the resolution of certain practically important solutions (like the one containing internal waves in the two-layer shallow water systems) may not be sufficiently high unless a very fine mesh is used. In order to enhance the performance of the PCCU schemes, we extend them to the fifth order of accuracy via the alternative weighted essentially non-oscillatory (A-WENO) finite-difference framework.

I will introduce both one- and two-dimensional fifth-order A-WENO PCCU schemes and demonstrate their superb performance on a number of numerical examples for the two-layer shallow water equations.

Keywords: Nonlinear nonconservative hyperbolic systems; Path-conservative schemes; Central-upwind schemes; A-WENO finite-difference schemes; Two-layer shallow water equations.

References

[1] G. Dal Maso, P. G. Lefloch and F. Murat. "Definition and weak stability of nonconservative products", *Journal de Mathématiques Pures et Appliquées. Neuvième Série*, 74(6), pp. 483–548, 1995.

[2] M. J. Castro, T. Morales de Luna and C. Parés. "Well-balanced schemes and path-conservative numerical methods", *Handbook of numerical methods for hyperbolic problems*, Vol. 18, pp. 131–175, Elsevier/North-Holland, Amsterdam, 2017.

[3] M. J. Castro Díaz, A. Kurganov and T. Morales de Luna. "Path-conservative central-upwind schemes for nonconservative hyperbolic systems", *ESAIM. Mathematical Modelling and Numerical Analysis*, 53(3), pp. 959–985, 2019.

Towards bridging Lagrangian and Eulerian Riemann solvers

A. Chan[†,‡], G. Gallice[‡], R. Loubère[†] and P.-H. Maire[‡]

[†] Institut de Mathématiques de Bordeaux, Université de Bordeaux, France
(`raphael.loubere@u-bordeaux.fr`, `agnes.chan@u-bordeaux.fr`)

[‡] CEA Cesta, France (`gerard.gallice@gmail.com`, `pierre-henri.maire@cea.fr`)

ABSTRACT

In this paper, we propose to revisit the notion of simple Riemann solver (RS) following Gallice [1]. It enlightens the linkage between the Lagrangian RS and its Eulerian counterpart. We provide in this work the relation between the Eulerian and Lagrangian forms of systems of conservation laws [4]. Then an approximate (simple) Lagrangian Riemann solver for the gas dynamics is derived based on the notions of positivity preservation and entropy control. A key ingredient is the Lagrangian nodal solver of Maire [3] which allows to determine the contact velocity at mesh nodes, further used in the RS. Its Eulerian counterpart is further deduced using the Lagrangian-Eulerian relationships. Next we build upon it the associated cell-centered 2nd order Lagrangian and Eulerian Godunov-type Finite Volume (FV) schemes [2, 5]. They inherit by construction the properties of the Lagrangian solver in terms of positivity preservation, well-defined CFL condition and correct wave ordering [5]. Numerical results on general unstructured grids will be presented in 1D, 2D and 3D. For instance in Figure 1 are presented two examples of simulations, a cone-flare in 3D and the hypersonic Mach 20 flow over a cylinder in 2D on several meshes. While classical two-point RS based scheme may present some infamous carbuncle instability, the proposed multi-point RS seems insensitive to it.

Keywords: Riemann solver; Finite volume schemes on unstructured meshes; 3D supersonic; Lagrangian nodal solver; hydrodynamics.

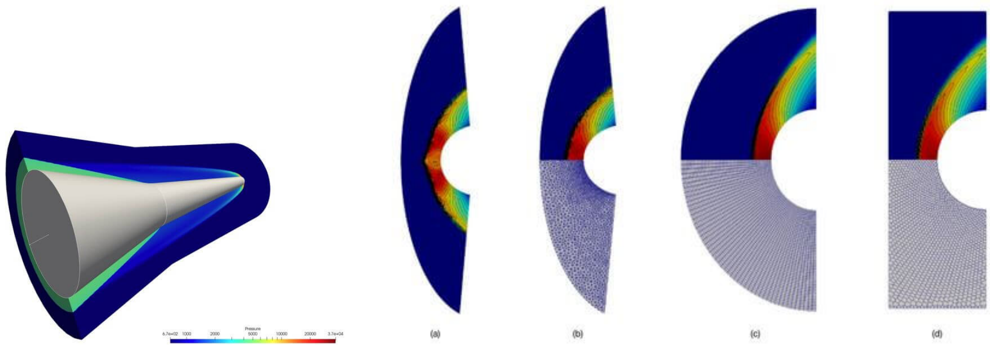


Figure 1: Left: 3D cone flare. Right: half cylinder at Mach 20 for different meshes. The two-point scheme (left most panel) presents some carbuncle phenomenon. The current scheme seems insensitive to it.

References

[1] G. Gallice. Positive and Entropy Stable Godunov-Type Schemes for Gas Dynamics and MHD Equs in Lagrangian or Eulerian Coordinates. *Numer. Math.*, 94(4), 2003

[2] A robust and contact resolving Riemann solver on unstructured mesh, Part I, and Part II. Z.Shen, W.Yan, G.Yuan *J. Comput. Phys.* 268 (2014) 432-455 and 456-484

[3] P.-H. Maire, R. Abgrall, J. Breil, and J. Ovadia. A cell-centered Lagrangian scheme for compressible flow problems. *SIAM J. Sci. Comput.*, 29(4):1781–1824, 2007.

[4] A. Chan, G. Gallice, R. Loubère, P.-H. Maire, Positivity preserving and entropy consistent approximate Riemann solvers. *Comput. & Fluids*, submitted 2021.

[5] A. Chan, G. Gallice, R. Loubère, P.-H. Maire, Revisiting Riemann solvers in the Lagrangian and Eulerian frameworks and their associated numerical methods in 2D, *in preparation* 2021.

Approximating viscosity solutions of the Euler equations

M. Lukáčová-Medvid'ová‡
‡ Institute of Mathematics, Johannes Gutenberg University Mainz
(lukacova@uni-mainz.de)

ABSTRACT

As suggested by the numerical experiments of Elling [2] the "wild" solutions of the Euler equations, obtained through the abstract approach of convex integration, may be physically relevant. In view of this fact, the Euler system being a model of ideal fluid should be viewed in a broader context as an asymptotic limit of more complex systems describing real fluids including the viscosity effects. Thus, following the original idea of DiPerna and Majda [1], we identify a viscosity solution of the Euler system with a parametrized family of probability measures generated by solutions of the Navier-Stokes system in the vanishing viscosity limit.

In the present talk we will report on our recent results on statistical (S-)convergence of the finite volume solutions of the Euler system of gas dynamics, see [3, 4]. The concept of S-convergence is based on averaging in the spirit of Strong Law of Large Numbers. Applying S-convergence, the vanishing viscosity solutions of the Euler system will be studied. We show how to efficiently compute a viscosity solution of the Euler system as the S-limit of numerical solutions obtained by the Viscosity Finite Volume method. Theoretical results will be illustrated by numerical simulations.

This work has been done in collaboration with E. Feireisl (Prague), B. She (Prague) and S. Schneider (Mainz).

Keywords: dissipative weak solutions, Euler equations, vanishing viscosity method

References

[1] R. J. DiPerna, A. Majda: Concentrations in regularizations for 2-d incompressible flow. *Comm. Pure and Appl. Math.* 40:301–345, 1987.

[2] V. Elling: Nonuniqueness of entropy solutions and the carbuncle phenomenon. *Hyperbolic problems: theory, numerics and applications. I*, p. 375–382. Yokohama Publ., Yokohama, 2006.

[3] E. Feireisl, M. Lukáčová-Medvid'ová, S. Schneider, B. She: Approximating viscosity solutions of the Euler system, *arXiv:2102.07876*, 2021.

[4] E. Feireisl, M. Lukáčová-Medvid'ová, H. Mizerová, B. She: *Numerical Analysis of Compressible Fluid Flows*, Springer-Verlag, 2021.

We gratefully acknowledge the support of SFB/TRR 146 "Multiscale Simulation Methods for Soft Matter Systems" as well as the Gutenberg Research College of the University Mainz.

Adaptive High Order Well Balanced Compact Approximate Method for Systems of Balance law

E. Macca†, **C. Parés**‡ **and G. Russo**†
† Department of Mathematics and Computer Science, University of Catania
(emanuele.macca@phd.unict.it, russo@dmi.unict.it)
‡ Departamento de Análisis Matemático, Universidad de Malága (pares@uma.es)

ABSTRACT

A new family of well-balanced high-order shock-capturing finite difference numerical methods for systems of balance law is presented. These methods, called Well-Balanced Adaptive Compact Approximation Taylor (WBACAT) schemes, use centered (2p + 1)-point stencils, where p may take values in 1, 2, . . . , P according to a family of smoothness indicators in the stencils. The methods are an extension of the Adaptive Compact Approximate Taylor (ACAT) methods introduced in Carrillo, Macca, Parés, Russo and Zorío (2020) [1]-[2] to systems of balance law. The expression of ACAT methods for 1D and 2D systems of conservative law and its well-balance extension to systems of balance laws will be presented together with their applications several linear and nonlinear problems.

Keywords: High order fully-discrete schemes; High order reconstruction techniques; Finite difference schemes; Well-balanced methods.

References

[1] H. Carrillo and C. Parés *Compact approximate Taylor methods for systems of conservation laws* , Journal of Scientific Computing, 80 (2019), pp. 1832-1866.

[2] H. Carrillo, E. M., C. Parés, G. Russo and D. Zorío, *An order-adaptive Compact Approximate Taylor method for systems of conservation law*, Journal of Computation Scientific, (2021).

GPU-accelerated Meso-resolved Simulation of Detonation Waves in Multiphase Energetic Materials

XiaoCheng Mi[†‡], **Louisa Michael**[‡], **and Nikolaos Nikiforakis**[‡]
[†] Department of Mechanical Engineering, McGill University (`xiaocheng.mi@mail.mcgill.ca`)
[‡] Cavendish Laboratory, Department of Physics, University of Cambridge (`lm355@cam.ac.uk`, `nn10005@cam.ac.uk`)

ABSTRACT

The challenge in computationally simulating detonation phenomena of multiphase energetic materials is rooted in the multiscale nature of the reactive media. To numerically capture an entire shock-to-detonation initiation (SDT) process in an energetic material with a large number of explicitly resolved mesoscale heterogeneities has long been considered infeasible. These computations are made amenable through the implementation of a diffuse-interface approach to treat multiphase flows—*MiNi16* formulation[1]—and a CUDA-based parallel computing on general-purpose graphic processing units (GPGPUs).[2] Under a Godunov-type numerical scheme, CUDA kernel functions were implemented to parallelize the most compute-intensive parts within the solver for the reactive Euler equations, including computation of fluxes, reactive source term integration, and the determination of time step length. The key physical behavior of an SDT process in a complex mixture of liquid nitromethane (NM) and glass micro-balloons (modeled as air-filled bubbles) has been successfully captured by these GPU-accelerated simulations.

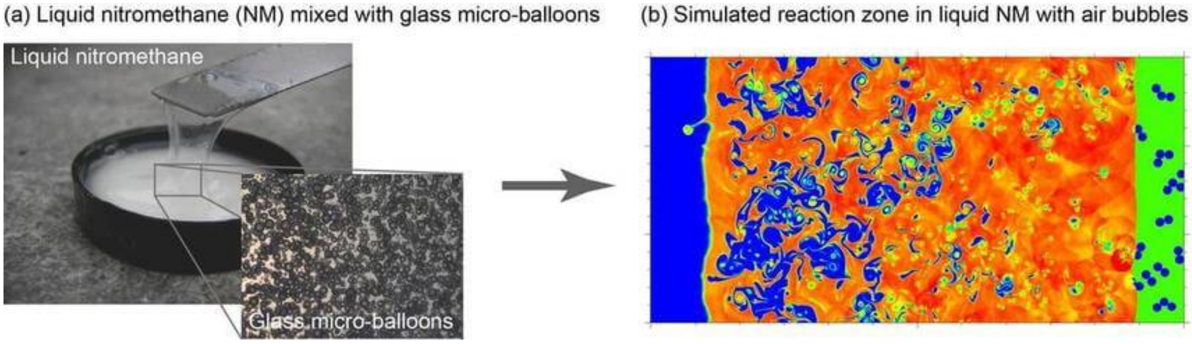


Figure 1: (a) A sample mixture of PMMA-gelled nitromethane and glass micro-balloons (GMBs). (b) Sample simulation result of the reaction-zone structure of a detonation in liquid NM with air-filled bubbles.

Keywords: Compressible Euler equations; Multiphase flow model; GPU computing; Combustion; Shock compression of condensed-phase matter.

References

[1] L. Michael and N. Nikiforakis. "A hybrid formulation for the numerical simulation of condensed phase explosives", *Journal of Computational Physics*, 316, pp. 193–217, 2016.

[2] X.C. Mi, L. Michael, E. Ioannou, N. Nikiforakis, A.J. Higgins, and H.D. Ng "Meso-resolved simulations of shock-to-detonation transition in nitromethane with air-filled cavities", *Journal of Applied Physics*, 125(24), p.245901.

Multi-physics simulations of lightning strikes on elastoplastic substrates

S. Millmore[†], **N. Nikiforakis**[†]

[†] Cavendish Laboratory, Department of Physics, University of Cambridge (`stm31@cam.ac.uk`, `nn10005@cam.ac.uk`)

ABSTRACT

A novel mathematical model and a corresponding algorithm for capturing the elastoplastic, electromagnetic and thermal response of aerospace materials due to their interaction with a plasma arc under lightning strike conditions is presented. This approach avoids the requirement for a one-way coupled co-simulation approach, in which arc evolution is passed to a separate materials model, by solving the entire system on a single computational mesh, allowing for two-way, non-linear interaction between materials. The magnetohydrodynamic and elastoplastic systems of equations are solved simultaneously, with the dynamic communication between these systems achieved through Riemann problem-based ghost fluid methods. To allow for this coupling, all equations are written in the same hyperbolic conservation law form, which further allows for their solution using shock capturing finite volume methods. The complete model is validated against experimental measurements for a variety of substrate materials, and demonstrates that the evolution of the arc is correctly captured. It is then shown that this model can simulate temperature-dependent response of the elastoplastic substrate as a result of energy deposition from a lightning strike.

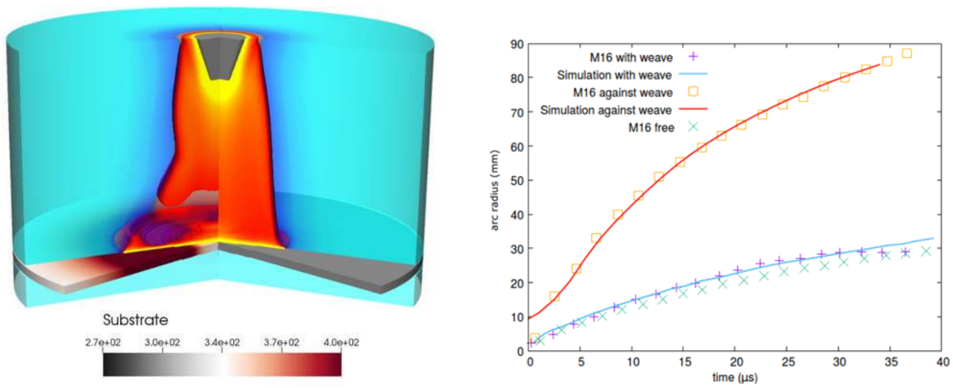


Figure 1: Left: Temperature of arc and substrate for attachment to carbon composite (left half) and aluminium (right half). Right: Validation of the expansion of the arc radius for attachment to carbon composite parallel and perpendicular to the weave direction.

Keywords: multi-physics; lightning; elastoplastic; multi-material; ghost fluid methods.

References

[1] S. Millmore and N. Nikiforakis, "Multi-physics simulations of lightning strike on elastoplastic substrates", *Journal of Computational Physics*, 405, 109142 (2020)

[2] F. Träuble, S. Millmore, and N. Nikiforakis, "An improved equation of state for air plasma simulations", *Physics of Fluids*, 33, 036112 (2021)

Hyperbolic equations in computational haemodynamics: models, numerics and physiology

L. O. Müller†
† Department of Mathematics, University of Trento (`lucas.muller@unitn.it`)

ABSTRACT

Prof. Toro began to work on problems related to computational haemodynamics around 2010, with his first work on the subject published in 2012 [1]. Since then, he has engaged in the study of physiological and patho-physiological states of the cardiovascular system (and related body fluids) from a modelling point of view. In doing that, Prof. Toro and his collaborators have contributed to this research field with works on mathematical models, numerical methods and physiology, adopting always a *hyperbolic approach* to the problem. In this talk I will present an overview of some results obtained during the last 10 years by Prof. Toro, collaborators and myself, focusing on how theoretical knowledge on hyperbolic PDEs and their discretization has contributed to the state of the art in the field of computational haemodynamics. I will also briefly describe exemplary physiological findings and illustrate where work in hyperbolic PDE theory and numerics is needed for further advancement of our (suprisingly limited) knowledge on the functioning of the cardiovascular system.

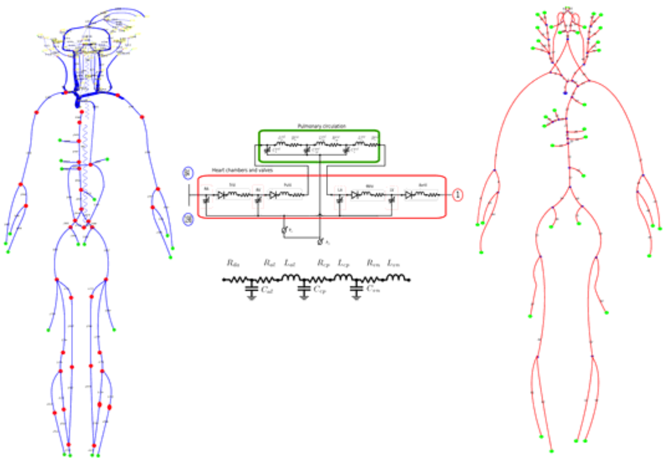


Figure 1: First version of the global model for the human circulation published by Prof. Toro and myself in [2]. Each artery (red) and vein (blue) shown in these networks represents a one-dimensional domain where hyperbolic PDEs are used to describe blood flow.

Keywords: First order hyperbolic PDEs; ADER schemes; Blood flow equations.

References

[1] E. F. Toro and A. Siviglia. "Simplified blood flow model with discontinuous vessel properties: Analysis and exact solutions". In In: D. Ambrosi, A. Quarteroni and G. Rozza (Eds.) *Modeling of Physiological Flows. MS&A - Modeling, Simulation and Applications*, vol 5. Springer, Milano, pp. 19–39, 2012.

[2] L. O. Müller and E. F. Toro. "A global multiscale mathematical model for the human circulation with emphasis on the venous system", *International Journal for Numerical Methods in Biomedical Engineering*, 30, pp. 681–725, 2014.

The Riemann Problem in the Sharp Interface Approximation of Two-phase Flow with Evaporation

C.-D. Munz†, **S. Jöns**†
† Institute for Aerodynamics and Gas Dynamics, University of Stuttgart
(munz@iag.uni-stuttgart.de, jens@iag.uni-stuttgart.de)

ABSTRACT

The numerical modelling of evaporation in a two-phase fluid is difficult within the continuum description because the equation of state is non-convex. In a sharp interface approach, we approximate the phase transition as a discontinuity. The coupling of the bulk phases at the evaporation front is based on the solution of the two-phase Riemann problem. We introduce an additional wave, an evaporation wave, at which appropriate jump conditions are prescribed. The jump conditions for this under-compressive shock wave are imposed such that they satisfy a kinetic relation, taking into account local non-equilibrium thermodynamics and the effect of heat conduction. The solution procedure for the exact solution is presented as well as an approximate Riemann solver based on the HLLC methodology.

We show numerical results of a flow solver that uses the solution of the two-phase Riemann solution within a sharp interface approximation based on a ghost fluid approach. A validation of this approach is performed for an equation of state that is obtained from molecular dynamics data for a truncated and shifted Lennard-Jones potential. By this, a highly accurate equation of state for the macroscopic equations is available, while molecular dynamics simulations are very efficient and allow long time and large space scales and allows a one-to-one correspondence.

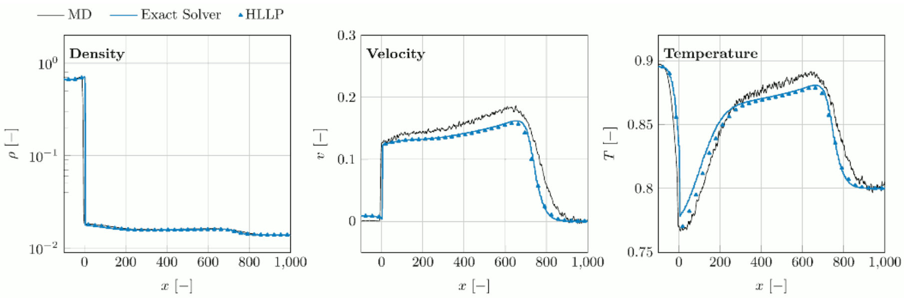


Figure 1: Comparison of macroscopic and molecular dynamics approximate solutions.

Keywords: Two-phase Riemann problem; sharp interface; ghost-cell approach.

References

[1] T. Hitz, M. Heinen, J. Vrabec, C.-D. Munz. “Comparison of Macro- and Microscopic Solutions of the Riemann Problem I. Supercritical Shock Tube and Expansion into Vacuum”. *Journal of Computational Physics*, 402, 2020.

[2] T. Hitz, S. Jöns, M. Heinen, J. Vrabec, C.-D. Munz. “Comparison of Macro- and Microscopic Solutions of the Riemann Problem II. Two-phase Shock Tube”, *Journal of Computational Physics*, 429, 2021.

The Authors acknowledge the financial support by the German Research Foundation (DFG) through the Project SFB-TRR 75 and the EXC 2075-390740016.

Computational multiphysics for interacting states of matter under extreme conditions

N. Nikiforakis[†], **L. Michael**[†] **and S.T. Millmore**[†]
[†] Cavendish Laboratory, Department of Physics, University of Cambridge
(nn10005@cam.ac.uk)

ABSTRACT

This work is concerned with the numerical simulation of problems arising from fundamental academic research or from industrial applications that involve an arbitrary combination of materials and states of matter interacting under extreme conditions. Examples include metal jet formation by means of an explosively-initiated shock wave, and the effects of lightning strike on an aircraft wing.

To this end, we consider formulations of the governing equations for four states of matter as an inhomogeneous hyperbolic system, augmented by elliptic operators to describe local as well as global processes. Communication between interacting materials and states of matter is facilitated by means of discontinuous or diffuse interface methods, which allow for large deformations, sliding and void opening. Appropriate numerical schemes that respect the mathematical properties of the underlying equations have been implemented in a highly parallelised hierarchical adaptive mesh refinement platform.

The approach is therefore suitable for large, multi-dimensional simulations that feature many complex materials and physical processes interacting over multiple time and space scales. Furthermore, because it allows for the generation of new interfaces, it naturally facilitates the simulation of high-strain rate fracture, which is accommodated by augmenting the model to include ductile damage.

Deployment of this approach will be demonstrated by means of a number of case-studies and with reference to talks in this conference by other members of our lab, which will elaborate in depth on certain aspects of this work.

During the course of the presentation tribute will be paid to Professor E.F. Toro's research, whose numerical schemes and Riemann solvers underpin nearly every element of this work.

Keywords: Computational multiphysics, material interface methods, extreme states of matter.

References

[1] T. Wallis, P.T. Barton and N. Nikiforakis. "A Flux-enriched Godunov Method for Multi-material Problems with Interface Slide and Void Opening". *Journal of Computational Physics*, 442, 110499, 2021.

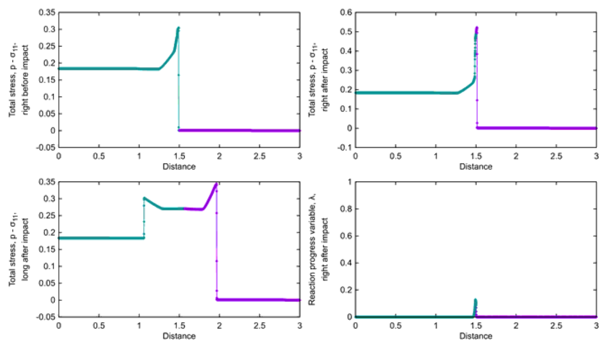
[2] L. Michael, S.T. Millmore and N. Nikiforakis. "A multi-physics methodology for four-states of matter", *Communications on Applied Mathematics and Computation*, 2 pp 487–514, 2019.

A unified multi-phase and multi-material formulation for combustion modelling

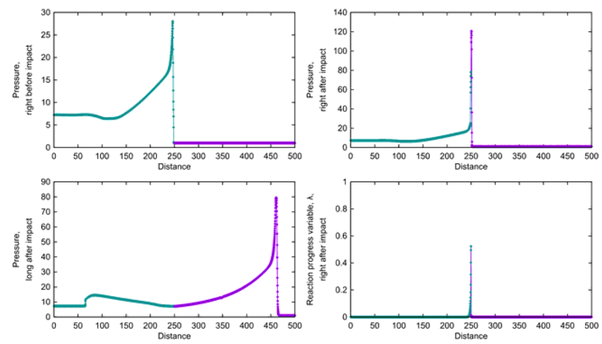
M. Nikodemou[†], **L. Michael**[†] **and N. Nikiforakis**[†]
[†] Laboratory for Scientific Computing, Cavendish Laboratory, Department of Physics, University of Cambridge, UK
(mn441@cam.ac.uk, louisa.michael@boeing.com, nn10005@cam.ac.uk)

ABSTRACT

The motivation of this work is to produce an integrated formulation for material response (e.g. elastoplastic, viscous, viscoplastic etc.) due to detonation wave loading. Here, we focus on elastoplastic structural response. In particular, we are interested to capture miscible and immiscible behaviour within condensed-phase explosives arising from the co-existence of a reactive carrier mixture of miscible materials, and several material interfaces due to the presence of immiscible impurities such as particles or cavities. The dynamic and thermodynamic evolution of the explosive is communicated to one or more inert confiners through their shared interfaces, which may undergo severe topological change. We also wish to consider elastic and plastic structural response of the confiners, rather than make a hydrodynamic assumption for their behaviour. Previous work by these authors has met these requirements by means of the simultaneous solution of appropriate systems of equations for the behaviour of the condensed-phase explosive and the elastoplastic behaviour of the confiners. To that end, both systems were written in the same mathematical form as a system of inhomogeneous hyperbolic partial differential equations which were solved on the same discrete space using the same algorithms, as opposed to coupling fluid and solid algorithms (co-simulation). In the present work, we employ a single system of partial differential equations (PDEs) proposed by Peshkov and Romenski, which is able to account for different states of matter by means of generalising the concept of distortion tensors beyond solids. We amalgamate that formulation with a single system of PDEs which meets the requirement of co-existing miscible and immiscible explosive mixtures. We present the mathematical derivation and construct appropriate algorithms for its solution. The resulting model is validated against exact solutions for several use-cases, including mechanically- and thermally-induced, inviscid and viscous detonations. Results indicate that the model can accurately simulate a very broad range of problems involving the nonlinear interaction between reactive and inert materials within a single framework.



(a) Mechanically-induced detonation in LX-17 interacting with elastoplastic copper



(b) Thermally-induced detonation in viscous gas interacting with water

Keywords: Condensed phase explosives; Multi-phase; Multi-material.

Convergence of DG Schemes for the Euler Equations via Dissipative Weak Solutions

Mária Lukáčová-Medvid'ová[†] **and Philipp Öffner**[†]
[†] Institute of Mathematics, Johannes-Gutenberg University Mainz, Germany
(`lukacova@mathematik.uni-mainz.de`, `poeffner@uni-mainz.de`)

ABSTRACT

Since the Cauchy problem for the complete Euler system is in general ill-posed in the class of admissible entropy weak solutions, one searches for alternatives. The concept of dissipative weak solutions seems to be quite promising to analyze this system analytically and numerically. In [1], the authors have studied the convergence of a class of entropy dissipative finite volume schemes for the barotropic and complete compressible Euler equations in the multidimensional case and proved suitable stability and consistency properties to ensure convergence of their FV schemes to a weak dissipative solution. In a series of papers, the theory has been further developed for several (classical) FV schemes (of maximum order two) and have been tested numerically, [2, 3]. In this talk, we focus on high-order finite element based methods, in particular on specific discontinuous Galerkin scheme and prove its convergence to a dissipative weak solution in multidimensional case. To this end it is crucial that structure preserving properties, such as positivity preservation and entropy inequality hold. We show how to ensure them and demonstrate the convergence of our multidimensional high-order DG scheme. In numerical simulations, we verify our theoretical findings, Figure 1.

M.L.-M. and P.O. gratefully acknowledge support of the Gutenberg Research College, JGU Mainz.

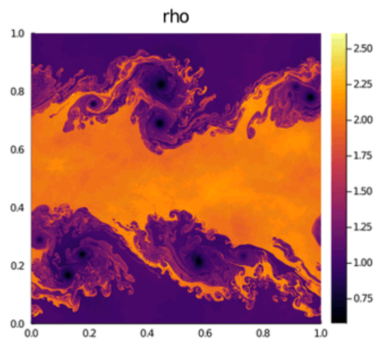


Figure 1: Kelvin-Helmholz instability.

Keywords: Euler equations; Dissipative weak (measure-valued) solutions; Convergence; Discontinuous Galerkin schemes; Entropy inequality.

References

[1] E. Feireisl, M. Lukáčová-Medvid'ová and H. Mizerová . "Convergence of finite volume schemes for the Euler equations via dissipative measure-valued solutions", *Foundations of Computational Mathematics*, 20 (4) pp. 1–44, 2019.

[2] E. Feireisl, M. Lukáčová-Medvid'ová and H. Mizerová. "A finite volume scheme for the Euler system inspired by the two velocities approach", *Numerische Mathematik*, 144(1), pp. 89–132, 2020.

[3] E. Feireisl, M. Lukáčová-Medvid'ová, B. She and Y. Wang. "Computing oscillatory solutions of the Euler system via $\mathcal{K}$-convergence", *Mathematical Models and Methods in Applied Sciences*, pp. 1–40, 2021.

An efficient and accurate implicit DG solver for the incompressible Navier-Stokes equations

G. Orlando[†], **A. Della Rocca**[‡], **P. F. Barbante**[†], **L. Bonaventura**[†], **N. Parolini**[†]

[†] Dipartimento di Matematica, Politecnico di Milano Piazza Leonardo Da Vinci 32, 20133 Milano, Italy (giuseppe.orlando@polimi.it, paolo.barbante@polimi.it, luca.bonaventura@polimi.it, nicola.parolini@polimi.it)

[‡] Tenova S. p. A., Global R&D Via Albareto 31, 16153 Genova, Italy (alessandro.dellarocca@tenova.com)

ABSTRACT

The efficient numerical solution of the Navier-Stokes equations is one the most relevant goals of Computational Fluid Dynamics. We propose an efficient, accurate and robust implicit solver for the Navier-Stokes equations, based on a low order DG spatial discretization and on the TR-BDF2 method for time discretization in order to combine, on the one hand, accurate and flexible discontinuous finite element discretizations, and on the other hand, efficient and unconditionally stable time discretization. The parallel implementation of the proposed method in the framework of the deal. II software package, based on a matrix-free approach, allows for accurate and efficient adaptive simulations in complex geometries, which makes the proposed solver attractive and easily accessible for large scale industrial applications. It is worthwhile to remark that time discretizations of Navier-Stokes equations based on accurate implicit solvers have been already proposed in a number of papers; however, we claim that the combination employed constitutes an optimal combination for the development a low order h-adaptive flow solver. We have shown that the method has superior accuracy and efficiency with respect to some well known alternative schemes on a number of classical benchmarks. Possible extensions to compressible flows will be also discussed.

Keywords: Discontinuous Galerkin method; implicit methods; incompressible flows; Navier-Stokes equations.

References

[1] G. Tumolo and L. Bonaventura. "A semi-implicit, semi-Lagrangian discontinuous Galerkin framework for adaptive numerical weather prediction: SISL-Dg framework for adaptive numerical weather predictions". *Quarterly Journal of the Royal Meteorological Society*, 141, pp. 2582–2601, 2015.

[2] L. Bonaventura and A. Della Rocca. "Unconditionally strong stability preserving extensions of the TR-BDF2 method", *Journal of Scientific Computing*, 70, pp. 859-895. 2017.

Well-balanced high-order finite difference WENO methods for systems of balance laws

<u>C. Parés</u>[†], C. Parés-Pulido[‡]
[†] Universidad de Málaga. (pares@uma.es)
[‡] ETH Zürich. (carlos.pares-pulido@sam.math.ethz.ch)

ABSTRACT

This work focuses on the design of high-order well-balanced finite-difference weighted essentially non-oscillatory methods to solve general 1d systems of balance laws

$$U_t + F(U)_x = S(U)H_x,$$

where $U(x,t)$ takes value in $\Omega \subset R^N$, $F : \Omega \to R^N$ is the flux function; $S : \Omega \to R^N$; and H is a known function from $R \to R$ (possibly the identity function $H(x) = x$). The system is supposed to be hyperbolic, i.e. the Jacobian $J(U)$ of the flux function is assumed to have N different real eigenvalues.

Two different families are introduced: while the methods in the first one preserve every stationary solution, those in the second family only preserve a given set of stationary solutions that depend on some parameters. The accuracy, well-balancedness, and conservation properties of the methods are discussed, as well as their application to systems with singular source terms. The strategy is applied to derive third and fifth order well-balanced methods for a linear scalar balance law, Burgers' equation with a nonlinear source term, and for the shallow water model. In particular, numerical methods that preserve every stationary solution or only water at rest equilibria are derived for the latter.

Keywords: High order reconstruction techniques; Finite volume schemes; Well-balanced methods; Systems of balance laws.

References

[1] C Parés, C Parés-Pulido. " Well-balanced high-order finite difference methods for systems of balance laws", *Journal of Computational Physics*, 425, 109880A, 2021.

[2] C.W.-Shu. "High Order Weighted Essentially Nonoscillatory Schemes for Convection Dominated Problems", *SIAM Reviews*, 51(1), pp. 82–126, 2009.

Hyperbolic models and numerical methods for the spatial spread of infectious diseases

L. Pareschi[†]
[†] Department of Mathematics and Computer Science, University of Ferrara, Italy
(lorenzo.pareschi@unife.it)

ABSTRACT

Standard compartmental epidemiological models account for the spread of viruses with respect to the temporal evolution of infection in the overall population. However, the importance of the spatial component is critical, especially when there is a need to consider spatially heterogeneous interventions. In this talk we survey some recent results on the development of multiscale hyperbolic transport models for the propagation of an epidemic phenomenon described by the spatial movement and interaction of a population. The division into commuters and non-commuters individuals avoids the instantaneous propagation of infection and mass migration effects typical of reaction-diffusion models based on a single population. Furthermore, because data on epidemic spread are generally highly heterogeneous and affected by large uncertainty, we convey statistical information to the problem, related to random inputs such as the initial amount of infected or other epidemic parameters. The resulting models are then solved by suitable numerical schemes which combine implicit-explicit (IMEX) time integrators with finite volume stochastic collocation methods. Applications to dynamics on networks and two-dimensional dynamics on realistic domains are considered in the case of the 'first wave' of COVID-19 in Italy.

Joint research with G. Albi (Verona), G. Bertaglia (Ferrara), W. Boscheri (Ferrara), G. Dimarco (Ferrara), L. Liu (Hong Kong), M. Zanella (Pavia), X. Zhu (Iowa).

Keywords: Hyperbolic equations, compartmental models, epidemic modeling, uncertainty quantification, IMEX methods, finite volume schemes, stochastic collocation methods

References

[1] Giacomo Albi, Lorenzo Pareschi, Mattia Zanella, *Control with uncertain data of socially structured compartmental epidemic models*, J. Math. Biol. 82, 63 (2021).

[2] Giulia Bertaglia, Lorenzo Pareschi, *Hyperbolic models for the spread of epidemics on networks: kinetic description and numerical methods*, ESAIM: Math. Mod. & Num. Anal. 55, (2021) 381-407

[3] Giulia Bertaglia, Lorenzo Pareschi, *Hyperbolic compartmental models for epidemic spread on networks with uncertain data: application to the emergence of Covid-19 in Italy*, preprint arXiv:2105.14258, (2021)

[4] Giulia Bertaglia, Walter Boscheri, Giacomo Dimarco, Lorenzo Pareschi, *Spatial spread of COVID-19 outbreak in Italy using multiscale kinetic transport equations with uncertainty*, preprint arXiv:2106.07262, (2021)

[5] Walter Boscheri, Giacomo Dimarco, Lorenzo Pareschi, *Modeling and simulating the spatial spread of an epidemic through multiscale kinetic equations*, Math. Mod. & Meth. in Appl. Scie. to appear, (2021)

The Author acknowledge the support by MIUR (Ministero dell'Istruzione, dell'Università e della Ricerca) PRIN 2017, project "*Innovative numerical methods for evolutionary partial differential equations and applications*", code 2017KKJP4X.

On the 1D steady states of the 1D Green-Naghdi equations

S. Noelle[†], **M. Parisot**[*], **T. Tscherpel**[‡]
[†] Institute for Geometry and Applied Mathematics, RWTH Aachen, 52056 Aachen, Germany (noelle@igpm.rwth-aachen.de)
[*] INRIA, Univ. Bordeaux, CNRS, Bordeaux INP, IMB, UMR 5251, 200 Avenue de la Vieille Tour, 33405 Talence cedex, France (martin.parisot@inria.fr)
[‡] Faculty of Mathematics, Bielefeld University, 33501 Bielefeld, Germany (ttscherpel@math.uni-bielefeld.de)

ABSTRACT

This work is devoted to the study of the steady states of the 1D Green-Naghdi model with bathymetry. The Green-Naghdi equations is an improved modeling of the shallow water equations used to describe dispersive waves by enforcing certain constraints. First, we will see that the strong steady solutions of the 1D Green-Naghdi model can be characterized using a Bernoulli principle by four quantities, namely the water depth, the horizontal discharge, the vertical velocity and the hydrodynamic pressure in one position. In [1], we proposed a class a boundary conditions and a numerical strategy to implement them At the inlet boundary it seems consistent with the shallow water test case to fix the horizontal discharge, the vertical velocity and the hydrodynamic pressure. At the outlet numerical evidences shows that the hydraulic head should be fixed instead of the water depth. This latter boundary condition does not directly enter in the class of boundary we proposed in [1]. To adapt our strategy we first need to construct a discret steady solution satisfying the same contraints as the unsteady scheme. Finally, a modification of the unsteady scheme is proposed that recovers the discret steady solution, i.e. it satisfies a well-balancing property.

Keywords: Shallow water flow; Dispersive equations; Well-balanced scheme;

References

[1] S. Noelle, M. Parisot, T. Tscherpel "A class of boundary conditions for time-discrete Green-Naghdi equations with bathymetry", (submitted).

Computational Aspects of the Unified Hyperbolic Formulation for Continuum Mechanics

I. Peshkov†
† Laboratory of Applied Mathematics, DICAM, University of Trento
(ilya.peshkov@unitn.it)

ABSTRACT

In this talk, we discuss some theoretical and computational aspects of the first-order hyperbolic formulation for continuum mechanics [1] that in a unified manner describes dissipative dynamics of viscous (Newtonian and non-Newtonian) heat conducting fluids, plastic solids and non-dissipative dynamics of ideal fluids and elastic solids. This formulation is a part of a bigger class of hyperbolic equations called Symmetric Hyperbolic Thermodynamically Compatible (SHTC) equations developed by Godunov and Romenski [2, 3].

Before appreciating the use of SHTC equations in practical applications, several computational challenges have to be addressed. Thus, usually SHTC equations have relaxation algebraic source terms which become stiff in the diffusive limit. Furthermore, depending on the context, several differential-type constraints must be imposed that can be stationary such as curl- or div-type involutions, or can be dynamic such as time evolution equations for torsion or the energy conservation law. All this imposes certain requirements to the discretization scheme such asymptotic-preserving and structure-preserving properties.

We shall briefly discuss these challenges, possible strategies to overcome them, and demonstrate some attractive capabilities of the SHTC equations via several numerical examples.

References

[1] M. Dumbser, I. Peshkov, E. Romenski, O. Zanotti. "High order ADER schemes for a unified first order hyperbolic formulation of continuum mechanics: Viscous heat-conducting fluids and elastic solids", *Journal of Computational Physics*, 314, pp. 824–862, 2016.

[2] S. Godunov. "An interesting class of quasilinear systems", *Dokl. Akad. Nauk SSSR*, 139(3), pp. 521–523, 1961.

[3] E. Romenski. "Hyperbolic systems of thermodynamically compatible conservation laws in continuum mechanics", *Mathematical and computer modelling*, 28(10), pp. 115–130, 1998.

This is a joint work with many friends and colleagues: Michael Dumbser, Evgeniy Romenski, Walter Boscheri, Simone Chiocchetti, and Saray Busto. The Authors acknowledge the financial support received from the Italian Ministry of Education, University and Research (MIUR) in the frame of the Departments of Excellence Initiative 2018–2022 attributed to DICAM of the University of Trento (grant L. 232/2016) and in the frame of the PRIN 2017 project Innovative numerical methods for evolutionary partial differential equations and applications. I.P. was also funded by a UniTN starting grant of the University of Trento.

Well-balanced methods for relativistic fluids on a Schwarzschild background

Ernesto Pimentel-García[†], **Carlos Parés**[†] **and Philippe G. LeFloch**[‡]
[†] Department of Applied Mathematics, University of Málaga, Spain (erpigar@uma.es)
[‡] Laboratoire Jacques-Louis Lions, Sorbonne Université, Paris, France

ABSTRACT

We are interested in the design of first, second and third order finite volume methods for computing relativistic fluid flows and apply it to study the global dynamics of a flow evolving around a (spherically symmetric) Schwarzschild black hole. We will consider both the relativistic Burgers-Schwarzschild model and the relativistic Euler-Schwarzschild model. Our purpose in this presentation is to design shock-capturing schemes that are high-order accurate and well-balanced in the sense that they preserve the spatially homogeneous solutions. These methods allow us to investigate the global asymptotic behavior of such flows and reach definite conclusions about the behavior of the mass density and velocity field. We build upon earlier investigations on this problem by LeFloch et al [2, 3, 4] and extend to the present problem the well-balanced methodology in Castro and Parés [1], in order to properly take the Schwarzschild curved geometry into account.

Keywords: Hyperbolic equation; Well-balanced method; Compressible fluid; Relativistic flow; Schwarzschild blackhole.

References

[1] M.J. Castro and C. Parés, Well-balanced high-order finite volume methods for systems of balance laws. *Journal of Scientific Computing*, 82(2):1–48, 2020.

[2] S. Dong and P.G. LeFloch, Convergence of the finite volume method on a Schwarzschild background. *ESAIM: Mathematical Modelling and Numerical Analysis*, 53(5):1459–1476, 2019.

[3] P.G. LeFloch and H. Makhlof, A geometry-preserving finite volume method for compressible fluids on Schwarzschild spacetime. *Communications in Computational Physics*, 15(3):827–852, 2014.

[4] P.G. LeFloch and S. Xiang, A numerical study of the relativistic Burgers and Euler equations on a Schwarzschild black hole exterior. *Applied Mathematics and Computational Science*, 13(2):271–301, 2018.

Traffic models,
or what we can learn coupling transport and source terms

G. Puppo[†]
[†] Department of Mathematics, University La Sapienza, Rome, Italy
(gabriella.puppo@uniroma1.it)

ABSTRACT

We will discuss kinetic and microscopic traffic models. The idea is to recover macroscopic equations from kinetic interactions. However, unlike classical gas dynamics, we have to cope with unexpected instabilities. And actually instabilities in the model reflect the fact that traffic phenomena do contain unstable waves. This prompts the need to consider backward forward diffusion models.

Geometrically Intrinsic Shallow Water Equations on Moving Surfaces

E. Bachini[†] **and M. Putti**[‡]
[†] Institute of Scientific Computing - TU-Dresden - Germany (elena.bachini@tu-dresden.de)
[‡] Dept. Mathematics "Tullio Levi-Civita" - University of Padua - Italy
(mario.putti@unipd.it)

ABSTRACT

Shallow water models of geophysical flows must be adapted to geometrical characteristics in the presence of a general bottom topography with non-negligible slopes and curvatures, such as a mountain landscape. In these applications it is important to consider bottoms that deform in time as a consequence of erosion and deposition processes. In these cases of time-dependent beds, a sound geometrical approach is of fundamental importance to develop robust modeling approaches.

Our starting point is the intrinsic shallow water (ISWE) model defined on the bottom surface developed in [1] for a fixed bed. This approach is extended to a time-dependent local coordinate system to derive ISWE on a moving bottom surface. The resulting hyperbolic system of balance equations contains a time-varying metric tensor that yields source terms proportional to the state variable but not to its derivatives.

ISWE is discretized by a 2nd order Discontinuous Galerkin method defined on the bottom surface. To this aim, we exploit our intrinsic finite element framework [2] to define appropriate linear basis functions and the DG linear and bilinear forms. We present some numerical experiments including cases with a fixed bottom and cases where the bottom evolution is prescribed.

Keywords: Surface PDEs; Intrinsic Shallow Water Equations; Intrinsic Discontinuous Galerkin; Hyperbolic PDEs on Evolving surfaces.

References

[1] E. Bachini and M. Putti. "Geometrically Intrinsic Modeling of Shallow Water Flows", *ESAIM: Mathematical Modelling and Numerical Analysis*, 54, pp. 2125–2157, 2020.

[2] E. Bachini, M. W. Farthing and M. Putti. "Intrinsic finite element method for advection-diffusion-reaction equations on surfaces", *Journal of Computational Physics*, 424, 109827, 2021.

A family of semi-implicit hybrid FV/FE methods for computational fluid dynamics using an efficient MPI parallel implementation

Laura Río-Martín[†], **Saray Busto**[‡] **and Michael Dumbser**[†]
[†] Laboratory of Applied Mathematics, DICAM, University of Trento
(laura.delrio@unitn.it, michael.dumbser@unitn.it)
[‡] Department of Applied Mathematics to Industrial Engineering,
Universidad Politécnica de Madrid (saray.busto@unitn.it)

ABSTRACT

In this talk, we review the semi-implicit hybrid finite volume/finite element (FV/FE) method for solving incompressible, weakly compressible, all Mach number flows and shallow water equations which has been widely described in [1, 2, 3, 4]. Although the numerical results presented in these references are obtained with a sequential implementation, we introduce the parallelization of this family of methods and, at the same time, we describe the implicitness of the viscous terms. The parallelization has been carried out using the message passing interface (MPI) standard to improve the computational efficiency of the scheme. The use of a parallel framework reduces the computational cost and allows us to run numerical simulations with more complex geometries and finer meshes involving millions of elements. The high-performance of the MPI implementation of the hybrid FV/FE approach is illustrated by using a speed-up analysis and comparing the obtained numerical results with some classical benchmarks shown in [1, 2, 3, 4] and with other numerical reference solutions.

Keywords: Massive parallelization; All Mach number flow solver; Projection method; Finite element method; Finite Volume method; Semi-implicit schemes on unstructured staggered meshes.

References

[1] S. Busto and M. Dumbser "A staggered semi-implicit hybrid FV/FE scheme for the shallow water equations at all Froude numbers", 2021 (Submitted)

[2] S. Busto, L. Río-Martín, E. Vázquez-Cendón and M. Dumbser "A semi-implicit hybrid FV/FE scheme for all Mach number flows on staggered unstructured meshes", *Applied Mathematics and Computation*, 402, pp. 126117, 2021.

[3] A. Bermúdez, S. Busto, M. Dumbser, J. L. Ferrín, L. Saavedra and E. Vázquez-Cendón "A staggered semi-implicit hybrid FV/FE projection method for weakly compressible flows", *Journal of Computational Physics*, 421, pp. 109743, 2020.

[4] S. Busto, J. L. Ferrín, E. F. Toro and M. E. Vázquez-Cendón "A projection hybrid high order FV/FE method for incompressible turbulent flows", *Journal of Comput. Phys.*, 353 (2018), pp. 169-192

The Authors acknowledge financial support from the Italian Ministry of Education, University and Research (MIUR) in the frame of the PRIN 2017 project. Furthermore, by Spanish MCIU under project MTM2017-86459-R.

Conservative Semi-Lagrangian methods for Kinetic Equations

G. Russo†, **S. Boscarino**†, **S. Y. Cho**† **and S.-B. Yun**‡
† Department of Mathematics and Computer Science, University of Catania, 95125 Catania, Italy (russo@dmi.unict.it)
‡ Department of Mathematics, Sungkyunkwan University, Suwon 440-746, Republic of Korea

ABSTRACT

In this talk, we overview a class of high order conservative semi-Lagrangian schemes for BGK model [1, 2]. The schemes are constructed by coupling the conservative non-oscillatory reconstruction [1] technique with a conservative treatment of the collision term, obtained by either a discrete Maxwellian [3] or by an L^2-minimization technique [4]. Due to the semi-Lagrangian nature, the time step is not restricted by a CFL-type condition, while the implicit treatment of the relaxation term based on Runge-Kutta or BDF time discretization enables us to avoid the stiffness problem coming from a small Knudsen number. Because of L-stability and exact conservation, the resulting scheme is asymptotic preserving for the underlying fluid dynamic limit. The method has been extended to the treatment of gas mixture [5]. Several test cases confirm the accuracy and robustness of the methods, and the AP property of the schemes. In general, such approaches use fixed velocity grids, and one must secure a sufficient number of grid points in phase space to resolve the structure of the distribution function. When dealing with high Mach number problems, where large variation of mean velocity and temperature are present in the domain under consideration, the computational cost and memory allocation requirements become prohibitively large. Local velocity grid methods have been developed to overcome such difficulty in the context of Eulerian based schemes [6, 7]. In this talk, we introduce a velocity adaption technique for the semi-Lagrangian scheme applied to the BGK model. The velocity grids will be set locally in time and space. We apply a weighted minimization approach to impose global conservation, generalizing the L^2-minimization technique introduced in [4]. We demonstrate the efficiency of the proposed scheme in several numerical examples.

References

[1] S. Y. Cho, S. Boscarino, G. Russo, S.-B. Yun. *Journal of Computational Physics*, 432, pp. 110159, 2021.

[2] S. Y. Cho, S. Boscarino, G. Russo, S.-B. Yun. *Journal of Computational Physics*, 436, pp. 110281, 2021.

[3] L. Mieussens. *Journal of Computational Physics*, 162, pp. 429-466, 2000.

[4] I. Gamba, S. H. Tharkabhushaman. *Journal of Computational Physics*, 228, pp. 2012-2036, 2009.

[5] S. Boscarino, S. Y. Cho, M. Groppi, and G. Russo. "BGK models for inert mixtures: comparison and applications". *arXiv:2102.12757* (submitted), 2021.

[6] S. Brull, L. Mieussens. *Journal of Computational Physics*, 266, pp. 22-46, 2014.

[7] F. Bernard, A.Iollo, G. Puppo. *Communications in Computational Physics*, 16, pp. 956-982, 2014.

All the authors would like to thank the Italian Ministry of Instruction, University and Research (MIUR) to support this research with funds coming from PRIN Project 2017 (No. 2017KKJP4X entitled Innovative numerical methods for evolutionary partial differential equations and applications). S. Y. Cho has been supported by ITN-ETN Horizon 2020 Project ModCompShock, Modeling and Computation on Shocks and Interfaces, Project Reference 642768. S.-B. Yun has been supported by Samsung Science and Technology Foundation under Project Number SSTF-BA1801-02.

Multidimensional approximate Riemann solvers for hyperbolic nonconservative systems

<u>Kleiton A. Schneider</u>[†], **Carlos Parés**[‡], **José M. Gallardo**[‡]
[†] Dept. of Mathematics, Federal University of Mato Grosso do Sul, Brazil
(`kleiton.schneider@ufms.br`)
[‡] Dept. Análisis Matemático, Estadística, y Matemática Aplicada, Universidad de Málaga,
Spain (`jmgallardo@uma.es, pares@uma.es`)

ABSTRACT

This work deals with the development of efficient incomplete multidimensional Riemann solvers for hyperbolic systems. We present a general strategy for constructing genuinely two-dimensional Riemann solvers, that can be applied for solving systems including source and coupling terms. Two-dimensional effects are taken into account through the approximate solutions of 2d Riemann problems arising at the vertices of the computational mesh.

First, a simple version of a well-balanced 2d HLL scheme is presented, which is later taken as basis to build a general class of incomplete Riemann solvers, the so-called Approximate Viscosity Matrix (AVM) schemes. The main benefit of the AVM strategy is the possibility to control the amount of numerical diffusion considered for each hyperbolic system at an affordable computational cost.

The presented numerical schemes are shown to be linearly L^∞-stable for a CFL number up to unity. On the other hand, our schemes can be used as building blocks for constructing high-order schemes. In this work, a second-order scheme is constructed by using a predictor-corrector MUSCL-Hancock procedure.

The performances of the proposed schemes have been tested with a number of challenging numerical experiments in one-layer and two-layer shallow water systems. The presence of the bottom topography and the coupling terms represent an additional difficulty, that has been solved by reformulating the problem within the path-conservative framework. Moreover, the schemes are well-balanced in the sense that they are able to preserve stationary solutions that are essentially 1d in the coordinate directions, including water at rest solutions for shallow water models. Finally, specially for problems presenting complex features, the 2d schemes have been shown to be more efficient than their projected 1d×1d counterparts.

Dispersive Moment Equations for Shallow Flow

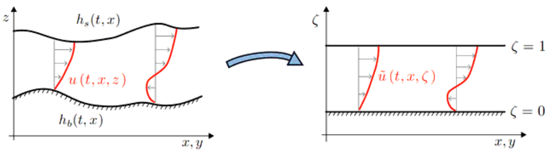
U. Scholz[†], **J. Kowalski**[‡] **and M. Torrilhon**[†]

[†] Applied and Computational Mathematics, RWTH Aachen (`mt@acom.rwth-aachen.de`, `scholz@acom.rwth-aachen.de`)

[‡] Abteilung Computational Geoscience, Georg-August-Universität Göttingen (`julia.kowalski@geo.uni-goettingen.de`)

ABSTRACT

Shallow flow models are used in situations where a flow's vertical extent is much smaller than its horizontal extent. These models exploit the shallowness by depth-averaging, which leads to the fact that they have lower computational costs than a corresponding vertically resolved free-surface flow. Quite naturally, the simpler formulation comes at the price of loosing vertical information, such as information on the velocity profile. However, in many realistic situations, an approximation of the velocity profile is needed. One approach to solve this problem is the usage of vertical moments as suggested by J. Kowalski and M. Torrilhon, in [1]. In this work, we suggest a new cascade of moment models, the Dispersive Shallow Moment Equations (DSM).

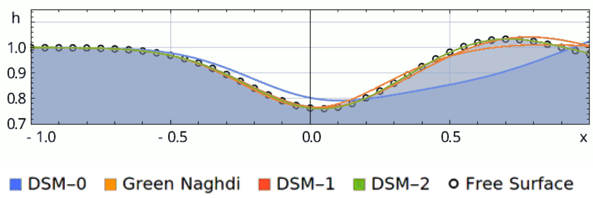


Ansatz

$$\begin{bmatrix} u(x,\zeta) \\ w(x,\zeta) \\ q(x,\zeta) \end{bmatrix} = \begin{bmatrix} u_m(x) \\ w_m(x) \\ q_m(x) \end{bmatrix} + \sum_{j=1}^{N} \begin{bmatrix} \alpha_j(x)\phi_j(\zeta) \\ \gamma_j(x)\phi_j(\zeta) \\ \kappa_j(x)\phi_j(\zeta) \end{bmatrix}$$

It again relies on a polynomial ansatz and subsequent Galerkin projection of the incompressible Euler equations for free surface flow in a transformed coordinate system where a new ζ-variable replaces the old z-variable. But it has the novelty of a non-hydrostatic pressure component q. Therefore, the equations are dispersive, similar to the well-known Green-Naghdi-Equations [2]. For a fixed height-to-depth ratio H the dispersion relation of the vertically resolved free surface flow is approximated up to arbitrary precision by the DSM models, depending on the degree of the polynomial ansatz N.

In order to show praticical relevance we set up a two-dimensional numerical test case. Results show that the DSM models can compete with the Green Naghdi system in terms of cost-effectiveness and are more accurate for higher degrees of the polynomial ansatz N.



Finally, we show that for a fixed polynomial degree N our equation systems converge to the classical shallow flow equations as we let the height-to-depth ratio H go to zero.

Keywords: Modelling, Shallow flow; Moment method; Green Naghdi equations.

References

[1] Julia Kowalski and Manuel Torrilhon. “Moment Approximations and Model Cascades for Shallow Flow”, *Commun. Comp. Phys.*, 25(3), pp. 669–702, 2019.

[2] Green, A. E. and Naghdi, P. M.. “A derivation of equations for wave propagation in water of variable depth”, *Journal of Fluid Mechanics*, 78(2), pp. 237–246, 1976.

One- and multi-dimensional CWENOZ reconstructions for implementing boundary conditions without ghost cells

M. Semplice[†], **E. Travaglia**[‡] **and G. Puppo**[⋆]
[†] Dipartimento di Scienza e Alta Tecnologia – Università dell'Insubria, Como
(`matteo.semplice@uninsubria.it`)
[‡] Dipartimento di Matematica – Università di Torino (`elena.travaglia@unito.it`)
[⋆] Dipartimento di Matematica – Università La Sapienza, Roma
(`gabriella.puppo@uniroma1.it`)

ABSTRACT

We address the issue of point value reconstructions from cell averages in the context of third order finite volume schemes, focusing in particular on the cells close to the boundaries of the domain. In fact, most techniques known in the literature (with the notable exception of [1] and related works) rely on the creation of ghost cells outside the boundary and on some form of extrapolation from the inside that, taking into account the boundary conditions, fills the ghost cells with appropriate values, so that a standard reconstruction can be applied also in boundary cells.

In [2], motivated by the difficulty of choosing appropriate boundary conditions at the internal nodes of a network, a different technique was explored that avoids the use of ghost cells, but instead employs for the boundary cells a different stencil, biased towards the interior of the domain.

Extending the approach of [2], which does not make use of ghost cells and relies on the adaptive-order CWENOZ reconstructions introduced in [3], we propose a more accurate reconstruction for the one-dimensional case and a two-dimensional one for Cartesian grids. In several numerical tests we compare the novel reconstruction with the standard approach using ghost cells.

Keywords: High order finite volume schemes; boundary conditions without ghost cells; hyperbolic systems; CWENOZ reconstruction; adaptive order reconstructions.

References

[1] Tan, S., Shu, C.W.: Inverse Lax-Wendroff procedure for numerical boundary conditions of conservation laws. J. Comput. Phys. **229**(21), 8144–8166 (2010).

[2] Naumann, A., Kolb, O., Semplice, M.: On a third order CWENO boundary treatment with application to networks of hyperbolic conservation laws. Appl. Math. & Comput. **325**, 252–270 (2018).

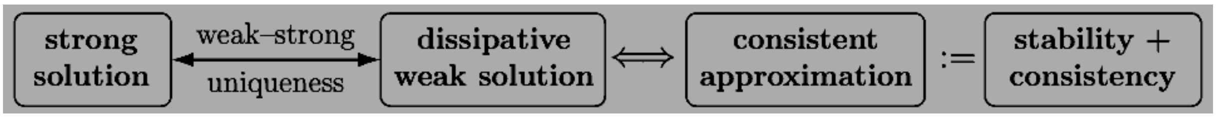
[3] Semplice, M., Visconti, G.: Efficient implementation of adaptive order reconstructions. J. Sci. Comput. **83**(1) (2020).

On convergence of numerical solutions for the compressible MHD system

Y. Li[†], **B. She**[‡]
[†] School of Mathematical Sciences, Anhui University. Email: (lynjum@163.com)
[‡] Institute of Mathematics of the Czech Academy of Sciences & Department of Mathematical Analysis, Charles University. Email: (she@math.cas.cz)

ABSTRACT

We study a general convergence theory for the analysis of numerical solutions to a magnetohydrodynamic system describing the time evolution of compressible, viscous, electrically conducting fluids in space dimension d $(= 2, 3)$. First, we introduce the concept of dissipative weak solutions and prove the weak–strong uniqueness property for dissipative weak solutions, meaning a dissipative weak solution coincides with a classical solution of the same problem. Next, we introduce the concept of consistent approximations and prove the convergence of consistent approximations towards the dissipative weak solution as well as the classical solution. Interpreting the consistent approximation as the energy stability and consistency of numerical solutions, we have built for the compressible MHD system a nonlinear variant of the celebrated Lax equivalence theorem. Finally, as an application of this theory, we show the convergence of some numerical methods.



Keywords: magnetohydrodynamic fluids; stability; convergence; dissipative weak solution; weak–strong uniqueness; consistent approximation;

References

[1] Y. Li and B. She. "On convergence of numerical solutions for the compressible MHD system with weakly divergence-free magnetic field", *Preprint*, 2021.

[2] Y. Li and B. She. "On convergence of numerical solutions for the compressible MHD system with exact divergence-free magnetic field", *Preprint*, 2021.

Development and application of a spectral/hp element, implicit compressible solver using the JFNK approach

S. Sherwin†, **Z.-G. Yan**†, **Y. Pan**†, **G. Lyu**† **and Peiró**†
† Department of Aeronautics, Imperial College London (s.sherwin@imperial.ac.uk, yanzhg@mail.ustc.edu.cn, yu.pan16@imperial.ac.uk, g.lyu19@imperial.ac.uk, j.peiro@imperial.ac.uk)

ABSTRACT

Advanced high-order methods using spectral/hp element discretization including Galerkin, discontinuous Galerkin (DG) and flux reconstruction (FR) formulations are gaining notable interest in both the academic and industrial sectors. The compact nature of the approach is not only attractive from the perspective of implementation on modern computational hardware but also provides a consistent geometric and spatially localized accuracy. These features make the methodology particularly suited for simulating complex geometry flows involving transitional and turbulent boundary layers which demand a high level of accuracy for the high-end engineering applications commonly arising in the aeronautical sector.

The paper will present our on-going effort towards the development of a spectral/hp element implicit compressible flow solver. The demands of handling "industrial strength" complex geometries at high Reynolds numbers necessarily lead to severe time step restrictions when using explicit time stepping approaches. We have therefore developed a Jacobian-Free-Newton-Krylov (JFNK) implicit solver [1] which makes use of the explicit technique but still requires suitable preconditioners that can be demanding from a memory footprint perspective. We have also explored robust approaches for selecting appropriate time steps [2], and investigated appropriate boundary conditions for matching the RANS outflow simulations within an embedded near wall spectral/hp element compressible solver. The methodology has been applied to practical cases such as the modelling of transitional flow past an aerofoil.

Keywords: Spectral/hp Element method; Implicit Solvers; Jacobian-Free Newton Krylov; Compressible flow.

References

[1] Z-G Yan, Y Pan, G. Castiglioni, K Hillewaert, J. Peiro, D. Moxey, S.Sherwin. "Nektar++: Design and implementation of an implicit, spectral/hp element, compressible flow solver using a Jacobian-free Newton Krylov approach". *Computers & Mathematics with Applications*, 81, pp 351-372, 2021.

[2] Y Pan, Z-G Yan, J. Peiro, S.Sherwin. "Development of a Balanced Adaptive Time-Stepping Strategy Based on an Implicit JFNK-DG Compressible Flow Solver", *Communications on Applied Mathematics and Computation*, 2021.

The authors acknowledge by EPSRC grant (EP/R029423/1) and Imperial College Research Computing Service (DOI: 10.14469/hpc/2232). Zhen-Guo Yan also acknowledges support from National Natural Science Foundation of China (Grant No. 11902344) and Ganlin Lyu, Spencer Sherwin also acknowledge supported by the Imperial-COMAC wing technology Centre.

A second-order well-balanced splitting scheme for the non-conservative Saint-Venant-Exner model

A. Siviglia[†], **D. Vanzo**[‡] **and E.F. Toro**[†]
[†] Laboratory of Applied Mathematics, DICAM, University of Trento, Italy
(annunziato.siviglia@unitn.it, eleuterio.toro@unitn.it)
[‡] Laboratory of Hydraulics, Hydrology and Glaciology VAW, ETH Zurich, Switzerland
(vanzo@vaw.baug.ethz.ch)

ABSTRACT

The purpose of this contribution is to present a novel flux splitting scheme for the non-conservative one-dimensional Saint-Venant-Exner equations describing the morphodynamics of shallow water systems. Our starting point is the flux vector splitting approach of [1]. We identify one subsystem of conservative equations (advection system) and one of non-conservative equations (pressure system), both having a very simple eigenstructure compared to the full system. The numerical scheme is obtained using a Godunov-type path-conservative scheme for the pressure system and a simple conservative Godunov method for the advection system and solved following a coupled synchronous strategy. The resulting first-order accurate method is extended to second order of accuracy in space and time via the ADER approach together with an AENO reconstruction technique. Accuracy, robustness and well-balanced properties of the resulting scheme are assessed through a carefully selected suite of test cases. The scheme is exceedingly simple, accurate and robust as the sophisticated Godunov methods. A distinctive feature of the novel scheme is its flexibility in the choice of the sediment transport closure formula, which makes it particularly attractive for scientific and engineering applications.

Keywords: Numerical morphodynamics; Saint-Venant-Exner model; Flux splitting; Finite Volume methods; Sediment transport; ADER method.

References

[1] E.F. Toro and M.E. Vázquez-Cendón. "Flux splitting schemes for the Euler equations", *Computers & Fluids*, 70, pp. 1–12, 2012.

AS acknowledges financial support from the Italian Ministry of Education, University and Research (MIUR) via the Departments of Excellence initiative 2018–2022 attributed to DICAM of the University of Trento (grant L. 232/2016).

Application of approximate dispersion-diffusion analyses to under-resolved Burgers turbulence using high resolution WENO and UWC schemes

P. Solán-Fustero*, **A. Navas-Montilla***§, **E. Ferrer**†‡, **J. Manzanero**†‡, **P. García-Navarro***

* Fluid Mechanics Department-Aragon Institute of Engineering Research (I3A), Universidad de Zaragoza, Spain. (`psolfus@unizar.es`)

§ Centro Universitario de la Defensa, Carretera de Huesca s/n, E-50090 Zaragoza, Spain.

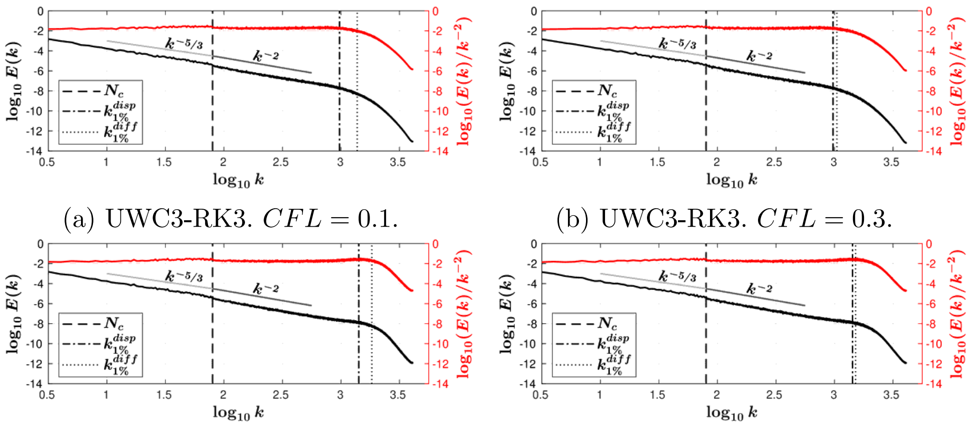
† ETSIAE-UPM - School of Aeronautics, Universidad Politécnica de Madrid.

‡ Center for Computational Simulation, Universidad Politécnica de Madrid.

ABSTRACT

This work presents a space-time approximate diffusion-dispersion analysis of high-order, finite volume Upwind Central (UWC) and Weighted Essentially Non-Oscillatory (WENO [1]) schemes. We perform a thorough study of the numerical errors to find a-priori guidelines for the computation of under-resolved turbulent flows. In particular, we study the 3-rd, 5-th and 7-th order UWC and WENO reconstructions in space, and 3-rd and 4-th order Runge-Kutta time integrators. To do so, we use the approximate von Neumann analysis for non-linear schemes introduced by Pirozzoli [2]. Moreover, we apply the "1% rule" for the dispersion-diffusion curves proposed by Moura et al. [3] to determine the range of wavenumbers that are accurately resolved by each scheme. The dispersion-diffusion errors estimated from these analyses agree with the numerical results for the forced Burgers' turbulence problem, which we use as a benchmark [4]. The cut-off wavenumbers defined by the "1% rule" are evidenced to serve as a good estimator of the beginning of the dissipation region of the energy cascade and they are shown to be associated to a similar level of dissipation, with independence of the scheme. It is concluded that both UWC and WENO schemes may be suitable schemes for iLES turbulence modelling, given their numerical dissipation level acting at the appropriate wavenumbers.

Keywords: Dispersion-diffusion analysis; von Neumann; High-order schemes; Weighted Essentially Non-Oscillatory WENO; Burgers' turbulence; implicit Large Eddy Simulation



(a) UWC3-RK3. $CFL = 0.1$.

(b) UWC3-RK3. $CFL = 0.3$.

(c) UWC5-RK3. $CFL = 0.1$.

(d) UWC5-RK3. $CFL = 0.3$.

Figure 1: Energy cascades for UWC schemes, with CFL=0.1, 0.3, 0.5 and 0.9.

References

[1] C.-W. Shu. High Order Weighted Essentially Nonoscillatory Schemes for Convection Dominated Problems, *SIAM Review*, vol. 51, n. 1, pp. 82–126, 2009.

[2] S. Pirozzoli. On the spectral properties of shock-capturing schemes, *J. Comput. Phys.*, 219, pp. 489–497, 2006.

[3] R. Moura, S. Sherwin and J. Peiró. Linear dispersion-diffusion analysis and its application to under-resolved turbulence simulations using discontinuous Galerkin spectral/hp methods, *J. Comput. Phys.*, 298, pp. 695–710, 2015.

[4] R. Maulik and O. San. Explicit and implicit LES closures for Burgers turbulence, *Journal of Computational and Applied Mathematics*, 327, pp. 12–40, 2018.

This work was funded by the Spanish Ministry of Science and Innovation under the research project PGC2018-094341- B-I00 (Solán-Fustero, Navas-Montilla and García-Navarro). This work has also been partially funded by Gobierno de Aragón through Fondo Social Europeo (T32-20R, Feder 2014-2020 "Construyendo Europa desde Aragón") (Solán-Fustero, Navas-Montilla and García-Navarro).

One-Dimensional Blood Flow with Discontinuous Properties and Transport: Mathematical Analysis and Numerical Schemes

A. Spilimbergo[†], **E.F. Toro**[‡] **and L.O. Müller**[†]

[†] Department of Mathematics, University of Trento, Via Sommarive 14 - 38123 Povo, Trento, Italy. (a.spilimbergo@unitn.it, lucas.muller@unitn.it)

[‡] Laboratory of applied Mathematics, University of Trento, Via Mesiano 77, 38123 Mesiano, Trento, Italy. (eleuterio.toro@unitn.it)

ABSTRACT

One-dimensional (1D) blood flow models have been extensively used to study wave propagation phenomena in arteries and, more recently, in veins. Here we focus on physical situations of medical interest in which certain properties that characterize compliant vessels change rapidly in space, for example after the insertion of stents in arteries or in veins due to a surgical procedure. We consider the one-dimensional blood flow model with discontinuous mechanical and geometrical properties, as well as passive scalar transport, proposed in [1], completing the mathematical analysis by providing new propositions and new proofs of relations valid across different waves. Next we consider a first order DOT Riemann solver, proposing an integration path that incorporates the passive scalar, completing the one proposed in [2], and proving the well-balanced properties of the resulting numerical scheme for stationary solutions. Finally we describe a novel and simple well-balanced, second order, non-linear numerical scheme to solve the equations under study; by using suitable test problems for which exact solutions are available, we assess the well-balanced properties of the scheme, its capacity to provide accurate solutions in challenging flow conditions and its accuracy [3].

Keywords: Blood flows; Riemann problem; Wave relations; Finite volume method; Well-balancing.

References

[1] E.F. Toro and A. Siviglia. “Flow in collapsible tubes with discontinuous mechanical properties: mathematical model and exact solutions” *Communications in Computational Physics*, 13(2), pp. 361-385, 2013

[2] L.O. Müller and E.F. Toro. “Well-balanced high-order solver for blood flow in networks of vessels with variable properties” *Int. J.Numer.Methods Biomed. Eng.*, 29:1388–1411, 2013

[3] A. Spilimbergo, E.F. Toro and L.O. Müller. “One-Dimensional Blood Flow with Discontinuous Properties and Transport: Mathematical Analysis and Numerical Schemes”, *Commun. Comput. Phys*, 29(3), pp. 649-697, 2021

An all-speed scheme for isentropic two phase flow

M. Lukáčová-Medvid'ová[†], G. Puppo[‡] and A. Thomann[†]
[†] Institute of Mathematics, Johannes-Gutenberg University Mainz, Germany
(lukacova@mathematik.uni-mainz.de, athomann@uni-mainz.de)
[‡] Institute of Mathematics , La Sapienza University Rome, Italy
(gabriella.puppo@uniroma1.it)

ABSTRACT

We are interested in the numerical simulation of liquid-gas mixtures, where the sound speed of the liquid phase is much faster than the one of the gas phase. If in addition, the material wave is much slower than the individual acoustic waves, the system can exhibit three different scales of wave speeds. In these regimes, which are characterized by small, potentially different, Mach numbers, using an explicit scheme requires a time step that scales with the smallest Mach number. At the same time, all waves are resolved, even though the main interest often lies only on the slow dynamics which would allow for a much larger time step. Therefore, we use an implicit-explicit (IMEX) approach whose CFL restriction only stems from the material wave and is independent of the Mach numbers.

In this talk, we present an all-speed scheme based on an isentropic hyperbolic two-pressure model given in [1, 2] for compressible flows. In contrast to the widely used Baer-Nunziato model, the model derived by Romenski et al. is given in a conservation-law form avoiding non-conservative products which makes it very attractive for the development of numerical schemes.

Keywords: Isentropic two-phase flow; Multi-scale equations; RS-IMEX; finite volume scheme;

References

[1] E. Romenski, E. F. Toro. " Compressible Two-Phase Flows: Two-Pressure Models and Numerical Methods", *Comput. Fluid Dyn. J.*, 13, pp. 403–416, 2004.

[2] E. Romenski, D. Drikakis and E. F. Toro. "Conservative Models and Numerical Methods for Compressible Two-Phase Flow", *J. Sci. Comput.*, 42, pp. 68–95, 2010.

M.L. and A.T. gratefully acknowledge the support of the Gutenberg Research College, JGU Mainz, and G.P. the support of PRIN 2017.

ALE-type discrete velocity scheme for kinetic equations as applied to rapid gas expansion problems

V.A. Titarev†, **A.A. Morozov**‡
† Federal Research Center "Computer Science and Control" of RAS, Russia
(`vladimir.titarev@frccsc.ru`, `titarev@mail.ru`)
‡ Kutateladze Institute of Thermophysics SB RAS, Novosibirsk, Russia (`morozov@itp.nsc.ru`)

ABSTRACT

Laser ablation of solids with nanosecond pulses of moderate intensity is widely used in various technologies [1]. Pulsed laser irradiation of solid targets leads to formation of a vapor cloud of the ablation products which then extends rapidly in the surrounding domain. Analysis of dynamics of the plume expansion is important for the control and monitoring of gas phase processes during laser ablation.

The theoretical analysis of the dynamics of laser-induced plume expansion is based mostly on numerical modeling, either using the direct simulation Monte Carlo (DSMC) method or by solving numerically the Boltzmann kinetic equation. The considered problem is a very demanding numerically as the flow is time-dependent with large gradients and contains rapid expansion into low-density region. The local Knudsen number varies greatly so that the Navier-Stokes equations are not valid.

The aim of the present work is to present an efficient numerical method to solve the kinetic equation with the S-model collision integral of E.M. Shakhov [2]. The new method is an arbitrary Lagrangian-Eulerian (ALE) [3] variant of the discrete velocity scheme [4]. The numerical studies are carried out for different levels of evaporation intensity and for both one-dimensional and axisymmetrical formulations. The accuracy and correctness of the new method is established by comparing results for the fixed-mesh approach for small output times and with the DSMC results for larger output times.

Keywords:

rarefied gas, pulsed evaporation, direct simulation Monte Carlo (DSMC), kinetic equation, moving mesh, arbitrary Lagrangian-Eulerian (ALE)

References

[1] D. Bauerle. *Laser Processing and Chemistry.* Springer, Berlin, 2011.

[2] E.M. Shakhov. Generalization of the Krook kinetic relaxation equation. *Fluid Dynamics*, 3(5):95–96, 1968.

[3] C.W. Hirt, A.A. Amsden, and J.L. Cook. An Arbitrary Lagrangian-Eulerian Computing Method for All Flow Speeds. *J. Comput. Phys.*, 14:227–253, 1974.

[4] V.A. Titarev. Application of model kinetic equations to hypersonic rarefied gas flows. *Computers & Fluids, Special issue "Nonlinear flow and transport"*, 169:62–70, 2018.

The Authors acknowledge the use of the shared high-performance facilities of the Joint Supercomputing Center of the Russian Academy of Sciences (http://www.jscc.ru/). The research was partially supported by the Russian Foundation for Basic Research (grant 19-08-01014).

Continuous Galerkin high order well-balanced discrete kinetic model for shallow water equations

D. Torlo[†], **M. Ricchiuto**[†]
[†] Team CARDAMOM, Inria Bordeaux - Sud-Ouest, France (`davide.torlo@inria.fr`)

ABSTRACT

Kinetic models describe many physical phenomena, inter alia Boltzmann equations, but can also be used to approximate with an artificial relaxation procedure other macroscopic models. We consider the kinetic model proposed by Aregba-Driollet and Natalini [1], and we modify it in order to approximate shallow water (SW) equations. The difference with the original model stands in the presence of the source term in the SW equations due to the effect of the bathymetry. Thus, the kinetic model [1] must be extended in order to include this term and to maintain the asymptotic convergence to the macroscopic limit of the SW problem.

To solve the equations with high order methods, we use an IMEX (implicit–explicit) discretization in time [2] to stabilize the relaxation and the friction terms, with DeC (deferred correction) [3] time integration, a high order iterative time integration technique, and RD (residual distribution) [4] space discretization, a finite–element based method.

The scheme proposed must verify many essential physical and numerical properties in order to guarantee the quality of the simulations. First of all, the scheme should be AP (asymptotic preserving), which means that in the relaxation limit, we will recast the macroscopic model of the shallow water equations. Then, we should guarantee the well–balancedness of the solution in the lake at rest case, where no oscillations should occur when the surface level of the water is constant and the speed is zero. Moreover, we want our scheme to guarantee positivity of water height everywhere in the domain, also close to wet and dry area.

We show some numerical tests to validate the quality of the scheme.

Keywords: Kinetic Model; Shallow Water; High order method; Well-balanced; IMEX method.

References

[1] Aregba-Driollet, D. and Natalini, R., *Discrete Kinetic Schemes for Systems of Conservation Laws.* Birkhauser Basel, Basel, 1999.

[2] Abgrall, R., Torlo, D. (2020). *High Order Asymptotic Preserving Deferred Correction Implicit-Explicit Schemes for Kinetic Models.* SIAM Journal on Scientific Computing, 42(3), B816-B845.

[3] A. Dutt, L. Greengard, and V. Rokhlin. *Spectral Deferred Correction Methods for Ordinary Differential Equations.* BIT Numerical Mathematics, 40(2):241–266, 2000.

[4] M. Ricchiuto and R. Abgrall. *Explicit Runge-Kutta residual distribution schemes for time dependent problems: Second order case.* Journal of Computational Physics, 229(16):5653–5691, 2010.

Some Models and Methods for Physiological Flows in Collapsible Conduits

Eleuterio Toro[†], **Annunziato Siviglia**[†] **and Lucas Müller**[‡]
[†] Laboratory of Applied Mathematics, DICAM, University of Trento, Italy.
(`eleuterio.toro@unitn.it; annunziato.siviglia@unitn.it`)
[‡] Department of Mathematics, University of Trento, Italy. (`lucas.muller@unitn.it`)

ABSTRACT

The growing trend to go for closed-loop, global models for the unified arterio-venous systemic and pulmonary human circulations seems unstoppable [1, 2]. However, the challenges are multiple, not least for the millions of vessels of highly different scales to be represented, but also for the physiological interaction of blood with other fluid compartments, such as cerebrospinal fluid, interstitial fluid and the vast network of lymphatics vessels and lymph nodes. Furthermore, the geometries of free-boundary conduits are complex and difficult to acquire with a degree of certainty. In this context fluid-structure interaction 3D models are an unrealistic proposition, if computational tractable tools for global simulations are searched for. The geometric multiscale approach involving 0-D models conformed by differential algebraic system (DAEs) and 1-D hyperbolic (or hyperbolised) models involving PDEs is a realist choice. In this talk we discuss work in progress on various formulations for 1D representations of compliant, variable parameter, elastic and viscoelastic conduits for fluid flow in arteries, veins and the spinal canal. The most general formulations involve systems in non-conservative form with geometric-type stiff source terms. Splitting schemes [3] to separate advection from pressure and viscous terms are explored. Such schemes look particularly attractive for parabolic viscoelastic models approximated as hyperbolic systems with stiff source terms [4]. These ideas are illustrated for arteries, veins and cerebrospinal fluid in the spinal canal and the optic nerve sheath. Numerical schemes are presented, as well numerical examples.

Keywords: Physiological flows; Arterial and venous flow; Cerebrospinal flow; Hyperbolized systems with stiff source terms; Splitting techniques; Advection and pressure systems; Path-conservative methods.

References

[1] L. O. Müller and E. F. Toro. “ A global multi-scale model for the human circulation with emphasis on the venous system”, *International Journal for Numerical Methods in Biomedical Engineering*, 30 (7), pp. 681-725, 2014.

[2] E. F. Toro, M. Celant, Q. Zhang, C. Contarino, N. Agarwal, A. A. Linninger and L. O. Müller. “ Cerebrospinal fluid dynamics coupled to the global circulation in holistic setting: mathematical models, numerical methods and applications”. *International Journal for Numerical Methods in Biomedical Engineering*, submitted, 2021.

[3] E. F. Toro and M. E. Vázquez-Cendón. “Flux splitting schemes for the Euler equations”. *Computers and Fluids*, 30 (7), Vol. 70, pp. 1-12, 2012.

[4] E. F. Toro and G. I. Montecinos. “Advection-diffusion-reaction equations: hyperbolisation and high-order ADER discretizations”. *SIAM Journal of Scientific Computing*, 36 (5), pp. A2423-A2457, 2014.

Boundary conditions for time-discrete Green–Naghdi equations

Tabea Tscherpel[*], **Sebastian Noelle**[†], **Martin Parisot**[‡]
[*] University of Bielefeld, Germany (ttscherpel@math.uni-bielefeld.de)
[†] RWTH Aachen, Germany (noelle@igpm.rwth-aachen.de)
[‡] Team CARDAMOM, Inria Bordeaux-Sud-Ouest, France (martin.parisot@inria.fr)

ABSTRACT

The Green–Naghdi model is derived from the incompressible free surface Euler equations and extends the shallow water equations by adding a hydro-dynamic pressure. For this reason it allows for dispersive waves and is used to describe wave phenomena in coastal areas and river flows.

This talk is concerned with the projection structure of the time-discrete Green-Naghdi equations on bounded domains including bathymetry. Due to the dispersive nature and the lack of hyperbolicity the understanding of suitable boundary conditions is still on a rudimentary level. Pressure correction methods - well-known for the incompressible Euler equations - have been introduced and analyzed for standard boundary conditions.

We use the projection structure to identify homogeneous and inhomogeneous boundary conditions for which well-posedness of the space-continuous correction step is available. On the fully discrete level we formulate a general approach to construct splitting schemes for a family of boundary conditions, in a way that a discrete projection property is satisfied. This allows us to design efficient and numerically robust schemes which are entropy-stable by construction. To illustrate the benefits and potential of our strategy numerical evidence is pro- vided for some simple cases. In particular, standard boundary conditions (wall, periodic) are included in our framework as well as some boundary conditions that are of practical interest and have not been considered in this way before (wave generation, fixed discharge).

This is joint work with Sebastian Noelle and Martin Parisot.

Evolution of a hybrid finite volume/finite element scheme for low-Mach number flows to all Mach number flows

A. Bermúdez†, S. Busto‡, M. Dumbser‡, J. L. Ferrín†, L. Río-Martín‡, L. Saavedra*, E.F. Toro‡, M.E. Vázquez-Cendón†

† Department of Applied Mathematics, Instituto de Matemáticas, ITMATI, University of Santiago de Compostela (`alfredo.bermudez@usc.es`, `joseluis.ferrin@usc.es`, `elena.vazquez.cendon@usc.es`)

‡ Laboratory of Applied Mathematics, DICAM, University of Trento (`saray.bustoulloa@unitn.it`, `laura.delrio@unitn.it`, `michael.dumbser@unitn.it`, `eleuterio.toro@unitn.it`)

* Departamento de Matemática Aplicada a la Ingeniería Aeroespacial, Universidad Politécnica de Madrid (`laura.saavedra@upm.es`)

ABSTRACT

At 2008, the research group in Mathematical Engineering, mat+i, at the University of Santiago de Compostela started working on finite volume methods for the simulation of environmental issues concerning the SIMULOX project. We have been working on the development of a numerical algorithm for the resolution of 3D Euler and Navier-Stokes equations. A hybrid projection finite volume/finite element method is employed making use of unstructured staggered grids. To attain second order of accuracy ADER methodology is employed [1]. We obtained the first results in 2011, at the same time that the first conference to honor E.F. Toro at Santiago de Compostela took place. During these ten years, the research group mat+i joint with the team of the University of Trento, work with to improve the numerical method in order to achive accuracy and computational efficiency, to obtain the methods studied in [2] and [3], as we can share at the talk.

Keywords: Finite volume schemes on unstructured meshes; hybrid finite volume/finite element scheme;

References

[1] S. Busto, J.L. Ferrín, E.F. Toro, M.E. Vázquez-Cendón. "A projection hybrid high order finite volume/finite element method for incompressible turbulent flows", *Journal of Computational Physics* 353 pp. 169-192, 2018.

[2] A. Bermúdez, S. Busto, M. Dumbser, J. L. Ferrín, L. Saavedra, Vázquez-Cendón. "A staggered semi-implicit hybrid FV/FE projection method for weakly compressible flows", *Journal of Computational Physics* 421(109743), 2020.

[3] S. Busto, L. Río-Martín, M.E. Vázquez-Cendón, M. Dumbser. "A semi-implicit hybrid finite volume/finite element scheme for all Mach number flows on staggered unstructured meshes", *Applied Mathematics and Computation* Applied Mathematics and Computation 402(126117), 2021.

This work was financially supported by the Italian Ministry of Education, University and Research (MIUR) in the frame-work of the PRIN 2017 project and via the Departments of Excellence Initiative 20182022 attributed to DICAM of the University of Trento (grant L. 232/2016). Furthermore, by Spanish MCIU under project MTM2017-86459-R and by FEDER and Xunta de Galicia funds under the ED431C 2017/60 project.

Fluid-structure interaction problems for blood flow in carotids

C. Vergara[†], **L. Bennati**[‡], **S. Pozzi**[*], **G. Gigante**[§], **P. Zunino**[*], **M. Domanin**[◇]

[†] LABS, Dipartimento di Chimica, Materiali e Ingegneria Chimica, Politecnico di Milano, Milan, Italy (`christian.vergara@polimi.it`)

[‡] Department of Surgery, Dentistry, Paediatrics and Gynaecology, University of Verona, Verona, Italy (`lorenzo.bennati@univr.it`)

[*] MOX, Dipartimento di Matematica, Politecnico di Milano, Milan, Italy (`silvia.pozzi@polimi.it`,`paolo.zunino@polimi.it`)

[§] Dipartimento di Ingegneria Gestionale, dell'Informazione e della Produzione, Università degli Studi di Bergamo, Dalmine (BG), Italy (`giacomo.gigante@unibg.it`)

[◇] Vascular Surgery Unit, IRCCS, Ospedale Maggiore Policlinico, Milan, Italy (`maurizio.domanin@unimi.it`)

ABSTRACT

Carotid arteries are a preferential site of atherosclerotic plaque formation. Blood dynamics plays a major role in determining the conditions that could promote both plaque progression and the damage due to plaque erosion. In this context, the numerical solution of the fluid-structure interaction (FSI) problem among blood, plaque and vessel wall could provide useful indications about such processes. To this aim, we first discuss the efficiency and stability of some partitioned algorithm for the solution of FSI in the context of hemodynamics, suitably designed for carotids. Second, we assess the issue of estimating the mechanical parameters (e.g. the plaque rigidity) from clinical measures. Then, we report results for a real dataset obtained by applying these methods to study the effect of different plaque typologies on plaque stability. Finally, we provide some preliminary results we obtained in the direction of modeling plaque progression. To this aim, we introduce a model composed by the FSI problem coupled with other partial differential equations describing at the macroscopic level the cellular processes leading to plaque formation. We propose a numerical method to solve this highly non-linear system of PDEs characterized by different time scales and we present some numerical results.

Keywords: Navier Stokes equations; Fluid-structure interaction; Loosely-coupled schemes; Human carotids; Plaque progression; Parameter estimation;

Quinpi: integrating conservation laws with CWENO implicit methods

G. Puppo[†], **M. Semplice**[‡] and **G. Visconti**[†]
[†] Dipartimento di Matematica, Sapienza Università di Roma
(gabriella.puppo@uniroma1.it, giuseppe.visconti@uniroma1.it)
[‡] Dipartimento di Scienza e Alta Tecnologia, Università degli Studi dell'Insubria
(matteo.semplice@uninsubria.it)

ABSTRACT

Many interesting applications of hyperbolic systems of equations are stiff, and require the time step to satisfy restrictive stability conditions. One way to avoid small time steps is to use implicit time integration. Implicit integration is quite straightforward for first order schemes. High order schemes instead need also to control spurious oscillations, which requires limiting in space and time also in the implicit case. We propose a framework to simplify considerably the application of high order non oscillatory schemes through the introduction of a low order implicit predictor, which is used both to set up the nonlinear weights of a standard high order space reconstruction, and to achieve limiting in time.

In this talk, we concentrate on the case of a third order scheme, based on DIRK integration in time and CWENO reconstruction in space. The numerical tests involve linear and nonlinear scalar conservation laws.

Keywords: Implicit schemes; Essentially non-oscillatory schemes; Finite volumes; WENO and CWENO reconstructions.

References

[1] G. Puppo, M. Semplice and G. Visconti, "Quinpi: integrating conservation laws with CWENO implicit methods", *Preprint arXiv:2102.00741*, (submitted), 2021.

[2] I. Cravero, G. Puppo, M. Semplice and G. Visconti, "CWENO: uniformly accurate reconstructions for balance laws", *Math. Comp.*, 87(312):1689-1719, 2018.

The Authors acknowledge support by MUR (Ministry of University and Research) through PRIN2017 project number 2017KKJP4X.

A Flux-enriched Godunov Method for Multi-material Problems with Interface Slide and Void Opening

Tim Wallis[†], **Philip T. Barton**[‡] **and Nikolaos Nikiforakis**[†]
[†] Department of Physics, University of Cambridge (`tnmw2@cam.ac.uk`)
[‡] AWE Aldermaston

ABSTRACT

This work outlines a new three-dimensional diffuse interface finite volume method for the simulation of multiple solid and fluid components featuring large deformations, sliding and void opening. To this end, an inhomogeneous hyperbolic partial differential system is evolved, which encompasses both multi-material fluid flow and multi-phase elastoplastic effects. The system is solved in conservative form, with the new material boundary conditions being mediated by means of a number of novel flux-modifiers and interface seeding routines. The method allows for slip boundary conditions across solid interfaces, material-void interaction, and interface separation. The method is designed to be straightforward to implement, inexpensive and highly parallelisable. This makes it suitable for use in large, multi-dimensional simulations that feature many complex materials and physical processes interacting over multiple levels of adaptive mesh refinement. Furthermore, the method allows for the generation of new interfaces in a conservative fashion and therefore naturally facilitates the simulation of high-strain rate fracture. Hence, the model is augmented to include ductile damage to allow for validation of the method against demanding physical experiments. The method is shown to give excellent agreement with both experiment and existing Eulerian interface tracking algorithms that employ sharp interface methods. This work is presented in Wallis *et al.* [1]. The underlying model also readily facilitates the extension to other multi-physics models. For example, Wallis *et al.* [2] extend the model to include reactive-fluids, demonstrating the ability to study the interaction of resolved detonation waves with elastoplastic structures.

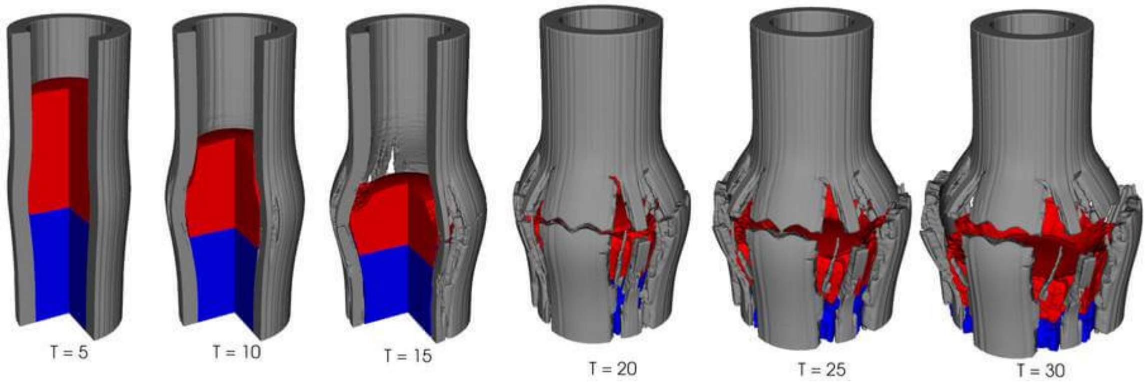


Figure 1: A 3D fracture test run with the model presented. The test depicts two polycarbonate blocks (red and blue) sliding inside a steel cylinder (grey) surrounded by void. As the blocks collide they push the cylinder outwards, eventually leading to fracture. Times shown are in μs.

Keywords: Multi-physics, Elastoplastic Solids, Diffuse interface, Fracture, Slide, Void opening.

References

[1] Tim Wallis and Philip T. Barton and Nikolaos Nikiforakis. "A Flux-enriched Godunov Method for Multi-material Problems with Interface Slide and Void Opening", *Journal of Computational Physics*, 2021. https://doi.org/10.1016/j.jcp.2021.110499

[2] Tim Wallis and Philip T. Barton and Nikolaos Nikiforakis. "A diffuse interface model of reactive-fluids and solid-dynamics", *Computers & Structures*, 2021. https://doi.org/10.1016/j.compstruc.2021.106578

Convergence of the Godunov Method for Multidimensional Compressible Euler Equations

M. Lukáčová-Medvid'ová† **and** **Y. Yuan**†
† Institute of Mathematics, Johannes Gutenberg-University Mainz (`lukacova@uni-mainz.de`, `yuhuyuan@uni-mainz.de`)

ABSTRACT

Recently developed concept of dissipative measure-valued (DMV) solution for compressible flows is a suitable tool to describe oscillations and singularities possibly developed in solutions of multidimensional hyperbolic conservation laws, and Euler equations in particular. Equipped with the concept of DMV we show the convergence of the first-order finite volume method based on the exact Riemann solver for the complete compressible Euler equations. Specifically, we derive the entropy inequality and weak BV estimates assuming only the uniform lower bound on density and upper bound on energy. We prove that this hypothesis is equivalent to the strict uniform convexity of the mathematical entropy. With the help of weak BV estimates we prove the consistency of numerical method. Passing to the limit, we show weak* convergence of numerical solutions to the DMV solution. If a limit of our numerical scheme is a weak or C^1 entropy solution, then the convergence is strong. Moreover, the corresponding compactness results for the Cesàro averages and first variance of numerical solutions are also obtained.

Several 2D experiments have been performed to confirm the results of theoretical analysis, including the spiral, Kelvin-Helmholtz and Richtmyer-Meshkov problems. In the Richtmyer-Meshkov test, we observe that single numerical solutions do not converge strongly, while the observable quantities (Cesàro averages and first variance) do converge strongly, which is consistent with the theoretical analysis.

Keywords: Compressible Euler equations; Finite volume method; Exact Riemann solver; Disspipative measure-valued solution; Convergence

References

[1] M. Lukáčová-Medviďová and Y. Yuan. "Convergence of first-order finite volume method based on exact Riemann solver for the complete compressible Euler equations", http://arxiv.org/abs/2105.02165, 2021.

This research was supported by the German Science Foundation (DFG) under the project TRR/SFB 146 multiscale simulation methods for soft matter systems. Y.Y. was supported by Sino-German (CSC-DAAD) Postdoc Scholarship Program.

Registered participants:

Rémi Abgrall
Nina Aguillon
Giacomo Albi
Elena Bachini
Francesco Bassi
Paul Bennett
Giulia Bertaglia
Walter Boscheri
Lukas Brencher
Cristian Brutto
Saray Busto
Celia Caballero-Cárdenas
Mauricio Caceres
Michele Giuliano Carlino
Manuel Castro
Cristóbal Castro
Morena Celant
Agnes Chan
Guoxian Chen
Alina Chertock
Simone Chiocchetti
Maria Chrysantou
Alessandro Coclite
Daniel Conde
Isabel Cordero-Carrión
Emiliano Cristiani
Alessia Del Grosso
Riccardo Dematté
Bruno Després
Firas Dhaouadi
Giacomo Dimarco
Minh-Hieu Do
Dimitris Drikakis
Michael Dumbser
Isabel Echeverribar
Maurizio Falcone
Francesco Fambri
Faezeh Farivar
Alexander Farmakalides
Davide Ferrari
Peter Frolkovič
Elena Gaburro
José M Gallardo
Gregor Gassner
Sergey Gavrilyuk
Stephan Gerster
Beatrice Ghitti
Irene Gómez-Bueno
Michael Groom
Maria Groppi
Yaman Güçlü
Arturo Hidalgo
Yannick Holle
Eleftherios Ioannou
Angelo Iollo
Armin Iske
Christian Klingenberg
Alexander Kurganov
Shannon Leakey
Raphaël Loubère
Alessia Lucca
Mária Lukáčová-Medvid'ová
Emmanuele Macca
Johannes Markert
James Mckenna
XiaoCheng Mi
Louisa Michael
Stephen Millmore
Lucas Müller
Claus-Dieter Munz
Sebastian Noelle
Nguyen Tri Nguyen
Nikolaos Nikiforakis
Maria Nikodemou
Philipp Öffner
Giuseppe Orlando
Carlos Parés
Lorenzo Pareschi
Martin Parisot
Ilya Peshkov
Ernesto Pimentel-García
Ivan Prusak
Gabriella Puppo
Mario Putti
Laura del Río-Martín
Giovanni Russo
Samuel Santos-Pérez
Kleiton A. Schneider
Ullika Scholz
Matteo Semplice
Bangwei She
Spencer Sherwin
Annunziato Siviglia
Pablo Solán-Fustero
Alessandra Spilimbergo
Maurizio Tavelli
Andrea Thomann
Vladimir Titarev
Wei Tong
Davide Torlo
Eleuterio Francisco Toro
Elena Travaglia
Tabea Tscherpel
Davide Vanzo
María Elena Vázquez-Cendón
Christian Vergara
Tim Wallis
Xue Wang
Sandra Warnecke
Ruifang Yan
Yuhuan Yuan
Yichi Zhang