

**AN INTERNATIONAL CONFERENCE
TO HONOUR
PROFESSOR E. F. TORO**

BOOK OF ABSTRACTS

**Santiago de Compostela
July, 4th-8th, 2011**

Numerical Methods for Hyperbolic Equations

Theory and Applications



EDITED BY
A. Bermúdez
L. Cea
E. Vázquez-Cendón

UNIVERSIDADE
DE SANTIAGO
DE COMPOSTELA

publicacións

Numerical Methods for Hyperbolic Equations

CURSOS E CONGRESOS DA
UNIVERSIDADE DE SANTIAGO DE COMPOSTELA
Nº 202

Numerical Methods
for Hyperbolic Equations
Theory and Applications
An International Conference
to Honour Professor E. F. Toro

Santiago de Compostela, July, 4th-8th 2011

BOOK OF ABSTRACTS

Edited by
A. BERMÚDEZ
L. CEA
E. VÁZQUEZ-CENDÓN

2011

UNIVERSIDADE DE SANTIAGO DE COMPOSTELA

Numerical methods for hyperbolic equations : theory and applications : an international conference to honor professor E. F. Toro, Santiago de Compostela, July 4-8, 2011 / edited by A Bermúdez, L. Cea, E. Vázquez-Cendón. — Santiago de Compostela : Universidade, Servizo de Publicacións e Intercambio Científico, 2011. — p. : cm. — (Cursos e congresos da Universidade de Santiago de Compostela ; 202). — D.L. . — ISBN 978-84-9887-713-7

1 Análise numérica. 2 Ecuacións hiperbólicas. I. Bermúdez de Castro, A., ed. II. Cea, L. ed. III. Vázquez-Cendón, E., ed. IV. Universidade de Santiago de Compostela. Servizo de Publicacións e Intercambio Científico, ed. V. Serie

519.6
517.956.3

© Universidade de Santiago de Compostela, 2011

Edita

Servizo de Publicacións e Intercambio Científico
Campus Vida
15782 Santiago de Compostela
www.usc.es/publicacions

ISBN 978-84-9887-713-7 (edición dixital .pdf)

Contents

Foreword	9
Invited Speakers	12
On the design of conservative, accurate and symmetrical cell-centered Lagrangian schemes in 2D cylindrical coordinates for compressible fluid flows.....	13
New Trends in Hyperbolic Problems in the Industry.....	14
Mathematical Theories of existence for 3D conservation laws and the nature of convergence in the large eddy simulation regime.	16
Numerical Relativistic Magnetohydrodynamics in Dynamical Spacetimes	17
Undercompressible shocks and moving phase boundaries	18
The initiation of detonations in reactive gases.....	19
Mathematical models for the cardiovascular system: analysis, numerical simulation, applications	21
Shock-capturing, past and future.....	22
The Riemann problem in computational science	23
Maximum-principle-satisfying and positivity-preserving high order discontinuous Galerkin and finite volume schemes for conservation laws.....	24
Minisymposia	26
Finite volume and discontinuous Galerkin schemes for stiff source term problems	27
Path-Conservative Schemes for Hyperbolic Source Term Problems.....	28
Entropy conservative and entropy stable schemes for nonconservative systems.....	30
A Runge-Kutta based Discontinuous Galerkin Method with Time Accurate Local Time Stepping	31
An efficient ADER scheme	33
A Unified Approach for High Order Finite–Volume and Discontinuous Galerkin Schemes for PDE with Stiff Source Terms	35
Accuracy enhancement of discontinuous Galerkin methods for stiff source terms	37
Novel shock-capturing schemes for Discontinuous Galerkin methods.....	39
Numerical Solution of Viscous and Thermally Conductive Gaseous Flows Via Hyperbolic Maximum Entropy Moment Closures	41
Methods and models for biomedical problems	43
A fully adaptive numerical approximation for a two-dimensional epidemic model with nonlinear cross-diffusion.....	44
Modeling and simulation of the hydrodynamics-biology coupling in a raceway.	45

A simple model of filtration and macromolecule transport through microvascular walls	47
Assessment of numerical methods for blood flow in veins.....	49
Reduced models for the Fluid-Structure Interaction problem.....	52
Multiphase flow and porous media	53
Dynamics of submerged gravitational granular flows.....	54
Finite Volume Method for Three-Dimensional Particle-Laden Free-Surface Flow	56
Numerical simulation of a bubble rising in still liquids: determination of the instability transition modes	58
Numerical simulation of three-dimensional transient variably saturated flow.....	60
An all-speed asymptotic preserving scheme for the low Mach number limit of the Euler equations.....	62
A New Scheme for Non-Classical Waves with TVB Proof for Scalar 1D-Case	64
Modelling Microbial Chemotactic Waves in Saturated Porous Media using Adaptive Mesh Refinement.	66
Numerical simulation of transport equations in Liquid Composite Molding Processes.....	68
Mathematical modeling of heat transfer during quenching process.....	70
Conservative formulation for compressible fluid flow through elastic porous media.....	72
On the coupling of compressible and incompressible fluids.....	74
Numerical methods in astrophysics.....	76
Numerical methods for the hyperbolic evolution equations in the Fully Constrained Formalism of Einstein equations.	77
SPH versus AMR.....	79
An Approximate Harten-Lax-van Leer Riemann Solver for Relativistic Magnetohydrodynamics.....	80
MUSTA schemes in magnetohydrodynamics and neutrino transfer: application to core-collapse supernovae and gamma-ray bursts	82
MHD-kinetic Modeling of Partially Ionized Plasma Flows: Solar Wind Interaction with the Interstellar Medium	84
ON THE RIEMANN PROBLEM IN RELATIVISTIC HD AND MHD.....	86
Recent advances in the numerical computation of environmental conservation laws with source terms.....	87
Asymptotic behaviour of godunov-type numerical models with mobile bed with adaptation term.	88
Finite volume discretisation of depth averaged scalar transport equations coupled to shallow water models.	90
A finite volume/duality method for Bingham viscoplastic flow	92
A numerical model for global climate: effect of the latent heat of fusion.....	94
Residual Distribution for Shallow Water Flows	96

Modelling of Flood Problems based on a Two-Dimensional Well-Balanced Wave Propagation Algorithm with Bed Efflux /Influx Source Terms Including a Coupled Pipe Network Solver.....	98
A Large Time Step Upwind Scheme for the Shallow Water Equations with source terms	100
Augmented Roe's approaches for Riemann problems including source terms: definition of stability region with application to the shallow water equations with rigid and deformable bed.....	102
Balancing source terms revisited in depth-averaged Smoothed Particle Hydrodynamics (SPH)	104
Flooding simulations with subgrids	106
Seismology and geophysics modelling	107
Dynamics of gravitational instabilities on Earth and on Mars	108
Two-phase models for debris flows	109
Meshless methods for lava flow modeling and simulation	111
Finite volume schemes for balance laws on time-dependent surfaces	113
Finite volume schemes for nonlinear dispersive wave equations	115
Tsunami modelling using high order finite volume schemes on GPUs	116
On multilayer shallow water systems	118
Parallel Sessions	120
High Order Methods for Hyperbolic Conservtion Laws	121
Comparison among different high order time integration methods useful for explicit schemes.....	122
High resolution schemes: TVD region its dependence on smoothness parameter.....	124
A numerical treatment of wet/dry zones in Well-Balanced Hybrid Schemes for Shallow Water Flow .	126
On the Analysis of a Solver for Generalized Riemann Problems by Asymptotic Expansion.....	128
CFL-Number-dependent TVD-Limiters	130
Arbitrary High Order Schemes for Transport Problems.	132
An Five-Equation Model Based ALE Method for Compressible Multifluid Fluids.	134
Efficient deterministic modelling of three-dimensional rarefied gas flows.....	136
High order stochastic finite volume methods for the numerical solution of hyperbolic equations with random initial data	138
Discontinuous-in-space explicit Runge-Kutta residual distribution schemes for time-dependent problems	140
Numerical Methods for Reactive Flows	142
Numerical simulation of a pulverized coal jet	143
Chapman-Jouguet conditions with temperature and velocity disequilibrium	145
Multi-phase simulation of Ammonium Nitrate Emulsion Detonations	147
A Flux Vector Splitting Scheme for the Euler Equations	149

On the use of Moving Least Squares for pressure discretization in low Mach number flows	151
Shallow Water Flows	153
Approximation of the Navier-Stokes system by multilayer models	154
Modeling of 3D phenomena in curved channels. Suitability of a 3D shallow waters approximation.	156
Porosity versus resolved approach in 2D shallow water models. Experimental validation.	158
A WENO scheme for the integral form of contravariant shallow water equations.....	160
A two-layer tidal-exchange flow.....	162
Numerical modelling of two-dimensional morphodynamics in gravel bed rivers.....	164
Poster Sessions	166
Noninvasive Monitoring of Hepatic Damage from Hepatitis C Virus Infection	167
Extension of fourthorder nonoscillatory central schemes to sediment transport equations	169
First approach to the application of operational 4DVAR Data Assimilation to the Regional Ocean Model System.....	171
Vascular network hemodynamics: Application to the study of Arteriovenous Malformations	173
Changes in buoyancy-driven instabilities by using a reaction-diffusion system.....	175
A Mean Gradient Method to Solve Shallow Flows.	177
Upwind and central schemes for nonhomogeneous hyperbolic conservation laws recognizing stationary solutions.....	179
Miscible Viscous Fingering on Reactive Interface	181
Discontinuous Galerkin Methods for Vlasov-Poisson system.....	182

This volume contains the abstracts of the papers presented at the International Conference on Numerical Methods for Hyperbolic Equations: Theory and Applications held in the Faculty of Mathematics of the University of Santiago de Compostela, Spain, from 4th to 8th July 2011. The conference was organized to honour Professor Toro in the month of his 65th birthday.

We would like to address our warmest thanks to the invited speakers: Juan Cheng (Institute of Applied Physics and Computational Mathematics, China), Frédéric Coquel (UPMC Paris 06, CNRS, France), James Glimm (University of Stony Brook, USA), José María Ibañez (Department of Astronomy and Astrophysics University of Valencia, Spain), Philippe LeFloch (University of Paris 6, France), Amable Liñan (Polytechnical University of Madrid, Spain), Alfio Quarteroni (Swiss Federal Institute of Technology, Lausanne, Switzerland), Philip L. Roe (University of Michigan, USA), Eleuterio Francisco Toro (University of Trento, Italy) and Chi-Wang Shu (Brown University, USA).

More than 70 abstracts were accepted for presentation at the following minisymposia:

- Recent advances in the numerical computation of environmental conservation laws with source terms.
- Multiphase flow and porous media.
- Numerical methods in astrophysics.
- Seismology and geophysics modelling.
- Finite volume and discontinuous Galerkin schemes for stiff source term problems.
- Methods and models for biomedical problems.

and parallel sessions:

- High order methods for hyperbolic conservation laws.
- Numerical methods for reactive flows and acoustics.
- Shallow water flows.

We think that all contributions are a valuable state of the art of the most recent research in the topic of numerical methods for hyperbolic equations providing the reader with the latest developments concerning the mathematical aspects and the applications of this active field of mathematics.

We would like to thank all the participants for the attendance and for their valuable contributions. Special thanks to the minisymposium organizers who made a large contribution to the conference.

Finally, the editors gratefully acknowledge the work of the Scientific Committee of the Conference and the financial support of the following Spanish organisations, which are the official sponsors of both the International Conference and the previous Short Course:

- CESGA (The Supercomputing Center of Galicia)
- i-MATH (Ingenio MATHEMATICA, nodo CESGA)
- Ministry of Science and Innovation of the Government of Spain
- Red Mathematica Consulting & Computing de Galicia
- RSME (The Royal Spanish Mathematical Society)
- SEMA (Spanish Society of Applied Mathematics)
- Santander Bank
- Springer
- USC (University of Santiago de Compostela)
- Xacobeo Galicia

A. Bermúdez, L. Cea and E. Vázquez-Cendón

Santiago de Compostela, Spain, July 2011

Numerical Methods for Hyperbolic Equations: Theory and Applications

An international conference to honour Professor E.F. Toro
in the month of his 65th birthday

Scientific Committee

Honour president: Peter Lax, Courant Institute of Mathematical Sciences, New York, USA.

- **Rémi Abgrall**, INRIA and Institut Polytechnique de Bordeaux, France.
- **Blanca Ayuso**, Centre de Recerca Matemàtica CRM, Barcelona, Spain.
- **Alfredo Bermúdez**, Department of Applied Mathematics, University of Santiago de Compostela, Spain.
- **Luis Cea**, GEAMA, Water and Environmental Engineering Group. Hydraulics Division. University of A Coruña, Spain.
- **Tomás Chacón Rebollo**, Dpto. Ecuaciones Diferenciales y Análisis Numérico. University of Sevilla, Spain.
- **Juan Cheng**, Institute of Applied Physics and Computational Mathematics, China.
- **Frédéric Coquel**, Centre de Mathématiques Appliquées, UPMC Paris 06, CNRS, France.
- **Rosa Donat**, Applied Mathematics Department. University of Valencia, Spain.
- **Michael Dumbser**, Laboratory of Applied Mathematics, University of Trento, Italy.
- **Enrique D. Fernández-Nieto**, Dpto. Matemática Aplicada I, Universidad de Sevilla, Spain.
- **Antonio Font**, Department of Astronomy and Astrophysics, University of Valencia, Spain.
- **Pilar García-Navarro**, Fluid Mechanics, CPS. University of Zaragoza, Spain.
- **James Glimm**, Department of Applied Mathematics and Statistics. University at Stony Brook, USA.
- **Arturo Hidalgo**, Departamento de Matemática Aplicada y Métodos Informáticos. Polytechnical University of Madrid, Spain.
- **José María Ibañez**, Department of Astronomy and Astrophysics University of Valencia, Spain.
- **Smadar Karni**, Department of Mathematics. University of Michigan, USA.
- **Philippe LeFloch**, Laboratoire Jacques-Louis Lions. University of Paris 6, France.
- **Amable Liñan**, Polytechnical University of Madrid, Spain.
- **Nikos Nikiforakis**, Director for Academic Programmes Centre for Scientific Computing, University of Cambridge, UK.
- **Carlos Parés**, Dpto. Análisis Matemático. University of Málaga, Spain.
- **Jerónimo Puertas**, GEAMA, Water and Environmental Engineering Group. Hydraulics Division. University of A Coruña, Spain.
- **Alfio Quarteroni**, Swiss Federal Institute of Technology, Lausanne, Switzerland.
- **Philip L. Roe**, Department of Aerospace Engineering. University of Michigan, USA.
- **Chi-Wang Shu**, Division of Applied Mathematics, Brown University, USA.
- **Eleuterio F. Toro**, Laboratory of Applied Mathematics. University of Trento, Italy.
- **Elena Vázquez-Cendón**, Department of Applied Mathematics, University of Santiago de Compostela, Spain. (Coordinator)

Numerical Methods for Hyperbolic Equations: Theory and Applications

An international conference to honour Professor E.F. Toro
in the month of his 65th birthday

Invited Speakers

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

On the design of conservative, accurate and symmetrical cell-centered Lagrangian schemes in 2D cylindrical coordinates for compressible fluid flows

Juan Cheng¹

Abstract The Lagrangian method is widely used in many fields for multi-material flow simulations due to its distinguished advantage in capturing material interfaces automatically. In this talk, we present our recent research on the design of the cell-centered control volume Lagrangian schemes for solving Euler equations of compressible gas dynamics which can keep a few desirable properties such as conservation, spherical symmetry and non-oscillatory behavior in two-dimensional cylindrical coordinates. The schemes can preserve one-dimensional spherical symmetry in a two-dimensional cylindrical geometry when computed on an equal-angle-zoned initial grid. Unlike many previous area weighted schemes that possess the spherical symmetry property, our schemes are discretized on the true volume and they can preserve the conservation property for all the conserved variables including density, momentum and total energy. Several two dimensional numerical examples in cylindrical coordinates are presented to demonstrate the performance of the schemes in terms of symmetry, accuracy and non-oscillatory properties.

This is a joint work with Chi-Wang Shu.

Keywords: Lagrangian scheme, Symmetry preservation, Conservative, Compressible flow, 2D cylindrical coordinates

References

- [1] J. CHENG AND C.-W. SHU. A cell-centered Lagrangian scheme with the preservation of symmetry and conservation properties for compressible fluid flows in two-dimensional cylindrical geometry. *Journal of Computational Physics* **229**(19), 7191-7206, 2010.
- [2] J. CHENG AND C.-W. SHU. Improvement on spherical symmetry in two-dimensional cylindrical coordinates for a class of control volume Lagrangian schemes. *Communications in Computational Physics*, to appear.

¹Institute of Applied Physics and Computational Mathematics,
P.O. Box 8009-14, Beijing 100088, China.
cheng_juan@iapcm.ac.cn

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

New Trends in Hyperbolic Problems in the Industry

Frédéric Coquel¹

Invited speaker

Abstract The present lecture gives an overview of some results obtained by the author with various collaborators on the mathematical coupling of nonlinear hyperbolic PDEs. Such an issue is motivated by the modeling of complex industrial setups under the form of a partition of components. The partition under consideration results from the fact that the phenomena taking place in a given component can be characterized by a typical range of physical time and space scales. To the resulting hierarchy of characteristic scales is associated a corresponding hierarchy of PDEs models that are naturally arranged from the coarsest to the nest ones according to the characteristic scales they are supposed to resolve. The large size of the complete device generally prevents from modeling each component using the finest model, the CPU effort would be indeed too large. The simulation of the whole operating system therefore requires to solve a Cauchy problem for a collection of hyperbolic PDEs formulated on a given partition of the physical domain, separated by interfaces at which transient exchange conditions have to be prescribed. For simplicity, we will only address the case of given fixed interfaces, whose locations are decided *ab initio* by the user.

We will present two complementary mathematical formalisms for handling the coupling of hyperbolic systems. Each supports the property that it is possible to prescribe infinitely many distinct coupling conditions. The first framework relies on the use of infinitely thin interfaces that allows to model the coupling on the ground of coupled boundary conditions. Such conditions are formulated so as to promote some continuity properties for the unknown across the coupling interface, the choice of which is left to the user. Importantly, resonance phenomena may occur at the expense of loosing the expected continuity properties and generally yields multiple discontinuous solutions. This rises the natural question of designing suitable selection criteria.

The second framework intends to tackle this issue. Roughly speaking, it consists in modeling the existence of the coupling interfaces under the form of standing waves for an augmented PDEs model which is set over the entire physical domain. Each standing wave is designed so as to exhibit a complete set of Riemann invariants (away from resonance) in agreement with the expected continuity properties in the coupled solution. The coupling problem thus takes the form of a standard Cauchy problem which can in turn support various regularization mechanisms. In a single space dimension, we adopt the viscous regularization *à la Dafermos*. Existence of self-similar weak solutions for the coupling of two hyperbolic systems can be obtained under fairly general conditions. However, in the limit of a vanishing viscosity, the

lack of uniqueness of resonant solutions can be still observed. We will explain in which sense multiplicity in the resonant solutions sounds natural. This will naturally lead us to promote another regularization mechanism based on thickened coupling interfaces resulting from the use of smooth transition profiles in between two domains. The proposed framework naturally allows for the definition of multidimensional and multicomponent couplings with possible recovering. Within this frame, existence and uniqueness of the coupled solution is proved in the setting of scalar conservations laws thanks to a convenient well-balanced finite volume strategy in several space variables. Numerical illustrations will be given all along the course of the lecture.

¹Centre de Mathématiques Appliquées,
Ecole Polytechnique and INSMI-CNRS,
France
`frederic.coquel@cmap.polytechnique.fr`

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Mathematical Theories of existence for 3D conservation laws and the nature of convergence in the large eddy simulation regime.

James Glimm¹

Keywords: 3D Conservation Laws, Convergence Large Eddy Simulations J. Glimm

Invited speaker

Abstract The Euler equations are fundamental as a model for simulations of Navier-Stokes turbulence in the high Reynolds number regime. Practical large eddy simulations (LES) for this regime resolve some but not all of the turbulent length scales. Existence theories, of necessity incomplete, for the Euler equations, suggest that solutions will not be worse than Young measures. Here we take this possibility seriously, and examine how this point of view will impact convergence for numerical simulations.

¹Applied Mathematics and Statistics
Stony Brook University
Math Tower, Stony Brook, NY 11794
glimm@ams.sunysb.edu
<http://www.ams.sunysb.edu/~glimm>

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Numerical Relativistic Magnetohydrodynamics in Dynamical Spacetimes

José Ma. Ibáñez¹

Invited speaker

Abstract The most spectacular events in the sky detected by current astronomical observatories (ground or space-based) involve matter evolving in the presence of strong gravitational fields. In these astrophysical scenarios, matter can reach velocities near the speed of light, and the gravitational field (background or generated by the flow itself) can be so strong that a description in terms of Einstein's theory of General Relativity is necessary. Some examples of these scenarios are: the stellar core collapse (as a mechanism of hydrodynamical supernovae), the relativistic outflows released from collapsars or from the remnants of coalescing binary neutron stars (as a models of formation of gamma-ray bursts), the relativistic jets associated with active galactic nuclei or microquasars, etc. To improve our knowledge of the physical nature of these energetic phenomena, modern Astrophysics is demanding the development of robust algorithms and numerical multidimensional relativistic magnetohydrodynamical (MHD) codes. These codes have to include the coupling to an Einstein solver. The main goal of this talk is to review the present status of the field of Numerical Relativity extended to the inclusion of matter fields, i.e., the Numerical Relativistic MHD in a dynamical space-time. I will summarize the present status of the efforts in numerically solving Einstein equations in the presence of matter and I will give a general overview about some of the many astrophysical applications.

¹Department of Astronomy and Astrophysics
University of Valencia

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Undercompressible shocks and moving phase boundaries

Philippe LeFloch¹

Invited speaker

Abstract Regularization-sensitive patterns often arise in continuum physics, especially in complex fluid flows which may contain undercompressive shock waves and moving phase boundaries. I will discuss solutions to nonlinear hyperbolic systems of conservation laws in the regime where small-scale effects (like viscosity, capillarity) turn out to be essential to the selection and propagation of (nonclassical) shocks. The concept of a kinetic relation provides the proper tool in order to characterize the dynamics of nonclassical waves; the kinetic relation can be determined by a higher-order regularization that takes into account additional physics. A general theory of the kinetic relation has been developed by the author and his collaborators in the past fifteen years. In the present lecture I will present results about the Riemann problem and numerical strategies adapted to handle nonclassical shocks.

References

- [1] B.T. HAYES AND P.G. LEFLOCH. Nonclassical shocks and kinetic relations. Finite difference schemes. *SIAM J. Numer. Anal.* **35** 2169–2194, 1998.
- [2] P.G. LEFLOCH. *Hyperbolic Systems of Conservation Laws. The theory of classical and nonclassical shock waves*. Lectures in Mathematics, ETH Zurich, Birkhauser, 2002.
- [3] P.G. LEFLOCH AND M. SHEARER. Nonclassical Riemann solvers with nucleation. *Proc. Royal Soc. Edinburgh* **134A**, 941–964, 2004.
- [4] P.G. LEFLOCH. Kinetic relations for undercompressive shock waves. *Physical, mathematical, and numerical issues*, in “Nonlinear Partial Differential Equations and Hyperbolic Wave Phenomena”, *Contemporary Mathematics*, **526**, Amer. Math. Soc., Providence, RI, pp. 237–272, 2010.
- [5] P.G. LEFLOCH AND M. MOHAMADIAN. Why many shock wave theories are necessary. Fourth-order models, kinetic functions, and equivalent equations. *J. Comput. Phys.* **227**, 4162–4189, 2008.

¹Laboratoire Jacques-Louis Lions &
Centre National de la Recherche Scientifique
Université Pierre et Marie Curie (Paris 6)
<http://philippelefloch.wordpress.com>

The initiation of detonations in reactive gases

A. Liñán¹

Keywords: Overdriven detonations, reactive blast waves, Chapman-Jouguet detonation

Invited speaker

Abstract The presentation will be devoted to the analysis of the initiation, by external, concentrated, energy sources in gaseous reactive mixtures, of cylindrical and spherical self-moving detonation waves.

These waves are led by a shock wave, moving radially outwards at supersonic speeds, and heating the mixture so that the reaction is ignited behind the shock wave, and goes to completion in a layer of thickness l_r large compared with the thickness of the shock wave; whose radius coincides with the radius $r_d(t)$ of the detonation wave when we use the reactive blast wave model (corresponding to the limit, $l_d/r_d \rightarrow 0$, of fast reaction rates).

The energy E_0 of the external source defines a characteristic value r_c of the detonation radius, for which the chemical energy released by the detonation wave is of the order of E_0 . For concentrated external energy sources, with a radius r_h small compared with r_c (and a deposition time t_h small enough), a strongly heated core, of radius $r_e(t)$, is formed; this has a gas density very small compared with the initial density, a very high temperature and near-uniform pressure. This heated core pushes the gas, so that it moves outwards with an inviscid radial motion in the region, $r_e < r < r_d$, bounded outside by an “overdriven” detonation wave.

The overdriven detonation is reached from behind by expansion waves, represented by the acoustic characteristics C^+ , that attenuate the detonation velocity D ; in such a way that at a finite time t_0 (after the initiation of the external heating), when the detonation radius is $r_d = r_0$ (of the order of r_c) the detonation velocity has decreased to the Chapman-Jouguet value D_{CJ} .

For $t > t_0$ the detonation surface, moving with the constant velocity D_{CJ} , is no longer reached by any of the characteristics (the three characteristics originate at the detonation and are left behind the front). The last characteristic surface C_0^+ that at $t = t_0$ reached, tangentially, the detonation wave, at $t > t_0$, is left behind the wave, and bounds, together with the detonation front, a shell of reacted gases with a self-similar motion, independent of the external deposition energy and of the deposition time. (On the other hand these determine r_0 and t_0 and also the non self-similar motion of the reacted gases bounded externally by C_0^+).

Outside the characteristic surface C_0^+ , the flow structure coincides with the Zeldovich-Taylor self-similar structure behind a Chapman-Jouguet detonation of constant velocity D_{CJ} , originating at $r = 0$ at $t = t_0 - r_0/D_{CJ}$. (The self-similar Z-T structure is reached everywhere for large values of t/t_0).

We shall also indicate in the presentation how finite reaction rate effects represented by the small, but non-negligible, value of the ratio l_r/r_0 at $t = t_0$ determines, together with the deposition time, the minimum energy needed to initiate a self-moving Chapman-Jouguet detonation.

(This is joint work with Vadim Kourdiumov (CIEMAT) and Antonio Sánchez (Universidad Carlos III))

¹Escuela de Ingeniera Aeronutica
 Universidad Politécnica de Madrid, Spain
 amable.linan@upm.es

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Mathematical models for the cardiovascular system: analysis, numerical simulation, applications

Alfio Quarteroni^{1,2}

Keywords: Circulatory system, mathematical modelling, finite elements, applications

Invited speaker

Abstract The role of mathematics in understanding and simulating fluid dynamics and biochemical processes in the physiological and pathological functioning of the human cardiovascular system is becoming more and more crucial. These phenomena are indeed correlated with the origin of some major cardiovascular pathologies, and influence the efficacy of the treatments to heal the arteries from their diseases.

Mathematical models allow the description of the complex fluid-structure interaction which govern the artery wall deformation under the pressure pulse. Moreover, appropriate reduction strategies can be devised to allow for an effective description of the interaction between large, 3D components, and small 1D branches of the circulatory system, as well as the transfer of drugs and chemicals between the arterial lumen and the vessel wall.

This presentation will address some of these issues and a few representative applications.

¹CMCS - MATHICSE

Ecole Polytechnique Federale de Lausanne,
Lausanne, Switzerland

²MOX-Politecnico di Milano

Milano

alfio.quarteroni@epfl.ch

<http://cmcs.epfl.ch/people/quarteroni>

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

SHOCK-CAPTURING, PAST AND FUTURE

P. Roe¹

Invited speaker

Abstract The battle between advocates of shock-capturing and shock-fitting was decisively won by the capturers in the 1970s. Perhaps it was a Pyrrhic victory.

Shock-capturing methods feature a number of anomalies that stubbornly resist removal. These include the carbuncle phenomenon, the slowly-moving shock phenomenon, the start-up problem, the wall-heating problem, and ambiguity in position. It will be argued that all of these are related, and that they are linked to each other both by the nonlinear nature of the Hugoniot curve for most physically-interesting systems, and by the generation of numerical entropy.

We will describe some unsuccessful cures, and speculate on the possibility either of an eventual cure, or perhaps the revival of fitting methods.

¹Department of Aerospace Engineering
University of Michigan, USA
philroe@umich.edu
<http://aerospace.engin.umich.edu/people/faculty/roe/>

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

The Riemann problem in computational science

E.F. Toro¹

Keywords: Riemann problem, hyperbolic systems

Invited speaker

Abstract I start by defining the classical Riemann problem for evolutionary partial differential equations as the simplest Cauchy problem with piece-wise constant initial conditions, giving examples for hyperbolic systems and for the diffusion equation. A brief historical account of the Riemann problem for hyperbolic systems is then given, before reviewing the various applications of the Riemann problem solution, with particular emphasis on the design of a wide variety of numerical methods for solving the relevant general initial-boundary value problems. Some recent generalizations of the classical Riemann problem are discussed, in which the initial condition is piece-wise smooth and the equations include source terms.

References

- [1] J GLIMM. Solution in the large for non linear hyperbolic systems of equations. *Comm. Pure and Appl. Math.* **18**, pp 697-715, 1965.
- [2] E GODLEWSKI AND P A RAVIART. *Numerical approximation of hyperbolic systems of conservation laws. Applied mathematical sciences.* **118**. Springer-Verlag, New York, 1996.
- [3] R J LEVEQUE. *Finite volume methods for hyperbolic problems*. Cambridge University Press, 2002.
- [4] E F TORO. *Riemann problems and numerical methods for fluid dynamics*. Third edition. Springer-Verlag, 2009.
- [5] E F TORO. *Shock-capturing methods for free surface shallow flows*. John Wiley and Sons. 2001.

¹Laboratory of Applied Mathematics
Department of Civil and Environmental Engineering
University of Trento
38050 Mesiano di Povo, Trento (TN), ITALY
toroe@ing.unitn.it
<http://www.ing.unitn.it/~toroe/>

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Maximum-principle-satisfying and positivity-preserving high order discontinuous Galerkin and finite volume schemes for conservation laws

Chi-Wang Shu¹

Keywords: Maximum principle, positivity preserving, finite volume scheme, discontinuous Galerkin method

Abstract We construct uniformly high order accurate discontinuous Galerkin (DG) and weighted essentially non-oscillatory (WENO) finite volume (FV) schemes satisfying a strict maximum principle for scalar conservation laws and passive convection in incompressible flows, and positivity preserving for density and pressure for compressible Euler equations. A general framework (for arbitrary order of accuracy) is established to construct a limiter for the DG or FV method with first order Euler forward time discretization solving one dimensional scalar conservation laws. Strong stability preserving (SSP) high order time discretizations will keep the maximum principle and make the scheme uniformly high order in space and time. One remarkable property of this approach is that it is straightforward to extend the method to two and higher dimensions. The same limiter can be shown to preserve the maximum principle for the DG or FV scheme solving two-dimensional incompressible Euler equations in the vorticity stream-function formulation, or any passive convection equation with an incompressible velocity field. A suitable generalization results in a high order DG or FV scheme satisfying positivity preserving property for density and pressure for compressible Euler equations. We will emphasize recent developments including arbitrary equations of state, source terms, integral terms and shallow water equations. Numerical tests demonstrating the good performance of the scheme will be reported. This is a joint work with Xiangxiong Zhang.

References

- [1] R. ZHANG, M. ZHANG AND C.-W. SHU. High order positivity-preserving finite volume WENO schemes for a hierarchical size-structured population model. *Journal of Computational and Applied Mathematics*, submitted.
- [2] Y. XING, X. ZHANG AND C.-W. SHU. Positivity preserving high order well balanced discontinuous Galerkin methods for the shallow water equations. *Advances in Water Resources*, **33**, 1476–1493, 2010.
- [3] X. ZHANG AND C.-W. SHU. A genuinely high order total variation diminishing scheme for one-dimensional scalar conservation laws. *SIAM Journal on Numerical Analysis*, **48**, 772–795, 2010.

- [4] X. ZHANG AND C.-W. SHU. On maximum-principle-satisfying high order schemes for scalar conservation laws. *Journal of Computational Physics*, **229**, 3091–3120, 2010.
- [5] X. ZHANG AND C.-W. SHU. On positivity preserving high order discontinuous Galerkin schemes for compressible Euler equations on rectangular meshes. *Journal of Computational Physics*, **229**, 8918–8934, 2010.
- [6] X. ZHANG AND C.-W. SHU. Positivity-preserving high order discontinuous Galerkin schemes for compressible Euler equations with source terms. *Journal of Computational Physics* (Accepted for publication), 2010.
- [7] X. ZHANG, Y. XIA AND C.-W. SHU. Maximum-principle-satisfying and positivity-preserving high order discontinuous Galerkin schemes for conservation laws on triangular meshes. *Journal of Scientific Computing*, submitted.

¹Division of Applied Mathematics
Brown University
Providence, RI 02912, USA
shu@dam.brown.edu
<http://www.dam.brown.edu/people/shu/>

Numerical Methods for Hyperbolic Equations: Theory and Applications

An international conference to honour Professor E.F. Toro
in the month of his 65th birthday

Minisymposia

Numerical Methods for Hyperbolic Equations: Theory and Applications

An international conference to honour Professor E.F. Toro
in the month of his 65th birthday

Minisymposia

Finite volume and discontinuous
Galerkin schemes for stiff source
term problems

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Path-Conservative Schemes for Hyperbolic Source Term Problems.

M.L. Muñoz-Ruiz¹ C. Parés²

Keywords: Path-Conservative Schemes, Hyperbolic Systems of Balance Laws and Hyperbolic Nonconservative Systems

Minisymposia: Finite volume and discontinuous Galerkin schemes for stiff source term problems

Abstract This work is concerned with the numerical approximation of Cauchy problems for one-dimensional systems of conservation laws with source terms or balance laws

$$U_t + F(U)_x = S(U) \sigma_x, \quad (1)$$

where $U(x, t) \in \Omega$, being Ω an open convex set of $\mathbf{R}^N$, $\sigma(x)$ is a known function from $\mathbf{R}$ to $\mathbf{R}$, F is a regular function from Ω to $\mathbf{R}^N$, and S is also a function from Ω to $\mathbf{R}^N$.

It is well known that standard methods that solve correctly systems of conservation laws can fail in solving (1), specially when approaching equilibria or near to equilibria solutions. In the context of shallow water equations Bermúdez and Vázquez-Cendón introduced in [1] the condition called *C-property*: a scheme is said to satisfy this condition if it solves correctly the steady-state solutions corresponding to water at rest. This idea of constructing numerical schemes that preserve some equilibria, which are called in general *well-balanced* schemes, has been studied by many authors. The design of numerical schemes with good properties for problems of the form (1) is a very active front of research, as PDE systems of this nature arise in many flow models.

Here, we will study systems (1) as a particular case of the more general systems

$$W_t + A(W)W_x = 0, \quad (2)$$

where the unknown $W(x, t)$ takes values on an open convex set and A is a smooth matrix function. This approach will allow us the discretization of the system as a whole, making thus easier to balance correctly the numerical flux and the source terms.

The presence of the nonconservative products $A(W)W_x$ in (2) makes difficult both the definition of weak solutions and the design of appropriate numerical schemes. We use the theory introduced by Dal Maso, LeFloch and Murat in [3] in order to define the concept of weak solutions. This theory is based on the prescription of a family of paths in the phases space. The introduction of a family of paths not only gives a way to properly define the concept of weak solutions of nonconservative systems: a theoretical framework for the design of finite difference methods for systems of the form (2) has been introduced in [4]. This framework is based on the concept of

path-conservative numerical scheme, which is a generalization of the usual concept of conservative method for conservation laws.

In [2] it was shown that, for general nonconservative systems a rather strong convergence assumption is needed to prove a Lax-Wendroff type convergence theorem. In this work we prove that, for systems of balance laws (1) with smooth enough σ , a Lax-Wendroff type result can be proved for path-conservative numerical schemes assuming the same hypotheses of convergence of the classical theorem for conservative problems, as the numerical results shown in [2] seemed to suggest. Together with the hypothesis concerning the convergence of the approximations, some minimal assumptions both on the chosen family of paths and on the numerical scheme are required, so that if σ is constant the usual notion of weak solution and a conservative numerical scheme are recovered.

In general, the assumptions on the family of paths required to prove the Lax-Wendroff theorem are not enough to ensure the well-balanced property. In this work we also present some general results showing that, in order to obtain well-balanced schemes, some further assumptions have to be imposed to the family of paths, concerning the relationship between the paths and the integral curves of the linearly degenerate field of the system associated to the null eigenvalue.

References

- [1] A. BERMÚDEZ, M.E. VÁZQUEZ. Upwind methods for hyperbolic conservation laws with source terms. *Computers and Fluids* **23**, 1049–1071, 1994.
- [2] M.J. CASTRO, P.G. LEFLOCH, M.L. MUÑOZ-RUIZ, C. PARÉS. Why many theories of shock waves are necessary: Convergence error in formally path-consistent schemes. *J. Comput. Phys.* **227**, 8107–8129, 2008.
- [3] G. DAL MASO, P.G. LEFLOCH, F. MURAT. Definition and weak stability of nonconservative products. *J. Math. Pures Appl.* **74**, 483–548, 1995.
- [4] C. PARÉS. Numerical methods for nonconservative hyperbolic systems: a theoretical framework. *SIAM J. Numer. Anal.*, **44**, 300–321, 2006.
- [5] E.F. TORO. *Riemann solvers and numerical methods for fluid dynamics. A practical introduction*. Springer-Verlag, 2009.

¹Departamento de Matemática Aplicada
 Universidad de Málaga
 E.T.S.I. Telecomunicación. 29071 Málaga, Spain
 munoz@anamat.cie.uma.es

²Departamento de Análisis Matemático
 Universidad de Málaga
 Facultad de Matemáticas. 29071 Málaga, Spain
 pares@anamat.cie.uma.es

Entropy conservative and entropy stable schemes for nonconservative systems

Ulrik Skre Fjordholm¹

Many hyperbolic systems with source terms can easily be written as nonconservative hyperbolic systems. While the theory and numerics for hyperbolic conservation laws has been in development for over half a century, numerics for nonconservative systems is still in its infancy, and only in recent years have the fundamental difficulties with the latter been recognized. Both types of systems exhibit discontinuous solutions, but their analysis differ greatly in several aspects: While conservation laws are easily interpreted within the theory of distributions, nonconservative systems cannot be, due to the nonconservative product. While solutions to conservative systems are largely invariant with respect to viscous terms, nonconservative systems are highly sensitive to such regularizing terms. Therefore, the definition of an entropy solution – a vanishing viscosity solution – makes sense only after choosing regularization operators, which mainly come from underlying physical principles. This information must be incorporated in numerical schemes, either in the choice of a family of paths (see [1]), or through the explicit addition of discrete viscous terms. In this talk we present a take on the latter approach. Given a nonconservative system with an entropy pair, we first design *entropy conservative*, *path conservative* finite volume schemes. These are schemes that add *no* numerical diffusion, and are thus faithful to the inviscid system. The construction of such schemes is a generalization of Tadmor’s work on entropy conservative schemes for conservation laws [2]. We then add explicit discretizations of physical viscous terms to obtain an entropy stable scheme. Examples are presented for a selection of nonconservative hyperbolic systems.

This is joint work with Manuel Castro, Siddhartha Mishra and Carlos Parés.

References

- [1] CASTRO, M. J., LEFLOCH, P. G., MUÑOZ-RUIZ, M. L. AND PARÉS, C. Why many theories of shock waves are necessary: Convergence error in formally path-consistent schemes. *J. Comput. Phys.*, **227**, 8107–8129, 2008.
- [2] TADMOR, E. The numerical viscosity of entropy stable schemes for systems of conservation laws. I. *Mathematics of Computation*, **49**, 91-103, 1987.

¹Seminar for Applied Mathematics, ETH Zürich
Rämistrasse 101, 8092 Zürich, Switzerland
ulrikf@sam.math.ethz.ch

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

A Runge-Kutta based Discontinuous Galerkin Method with Time Accurate Local Time Stepping

Claus-Dieter Munz¹, Florian Hindenlang¹, Gregor Gassner¹

Keywords: Discontinuous Galerkin, time accuracy, Runge-Kutta, optimal locality

Minisymposia: Finite volume and discontinuous Galerkin schemes for stiff source term problems

Abstract

An explicit one-step time discretization for discontinuous Galerkin schemes applied to advection-diffusion equations is presented. The main idea is based on a predictor corrector approach which was proposed in [1]. The interesting feature is that the predictor is local and takes only into account the time evolution of the data within each grid cell. In this paper, we focus ourselves to a continuous extension Runge-Kutta schemes which was described in [2]. The advantage of the predictor corrector formulation is that the time evolution is done in one step and the data of the direct neighbors are needed only. Hence, the proposed discontinuous Galerkin scheme has the optimal locality within the whole time step. This is the basis to introduce a timeconsistent local time stepping in a way such that every grid cell may run with its own optimal time step as given by the local stability restriction [3]. The time accuracy and the efficiency of the local time stepping is shown for linear and non-linear problems. A direct numerical simulation of the aeroacoustics of a natural gas injector shows the efficiency of the presented methodology, being well suited for unsteady advection dominated problems with adaptive schemes.

References

- [1] G. GASSNER, M. DUMBSER, F. HINDENLANG, C.-D. MUNZ. Explicit one-step time discretizations for discontinuous Galerkin and finite volume schemes based on local predictors. *Journal of Computational Physics*, In Press, 2010.
- [2] B. OWREN, M. ZENNARO. Derivation of efficient continuous explicit Runge-Kutta methods. *SIAM J. Sci. Stat. Comput.*, **13**,1488–1501, 1992.
- [3] F. LÖRCHER AND G. GASSNER AND C.-D. MUNZ. A discontinuous Galerkin scheme based on a space-time expansion. I. Inviscid compressible flow in one space dimension. *J. Sci. Comp.*, **32**(2),175–199 , 2007.

¹Institute of Aerodynamics and Gas Dynamics
Universitaet Stuttgart
Faculty of Aerospace Engineering and Geodesy, 70569 Stuttgart, Germany
`munz@iag.uni-stuttgart.de`

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

An efficient ADER scheme

Junbo Cheng¹ Weijun Tang² Ming Yu³

Keywords: ADER scheme, reconstruction of high-order spatial derivatives and high order

Minisymposia: Finite volume and discontinuous Galerkin schemes for stiff source term problems

Abstract ADER scheme is very popular because the numerical results of the ADER scheme have very high order of accuracy in both time and space in all dimensions when the ADER scheme is used to discretize the time-dependent linear or non-linear conservation laws. In order to obtain the high accuracy in time and space, it is necessary to approximate the pointwise values and the spatial derivatives up to $(r-1)$ -th order with high order of accuracy from the cell averages for the ADER scheme of r -th order of accuracy. The motivation of this paper is to introduce how to approximate efficiently the high-order spatial derivatives with high order of accuracy.

In the classical scheme, the $(r-1)$ -th order reconstruction polynomials must be used for the r -th order ADER scheme. However, the highest order of reconstruction on a fixed point is up to $(2r-3)$ -th wherever the solution is sufficiently smooth if using the WENO reconstruction. So we think it is not necessary to use the $(r-1)$ -th order reconstruction polynomials for the r -th order ADER scheme. It is enough to only use $\frac{r+1}{2}$ -th order polynomials to achieve the r -th order ADER scheme. If we just use $\frac{r+1}{2}$ -th order polynomials, how to compute the high-order spatial derivatives?

We use repeated reconstruction methods to compute the high-order spatial derivatives which are showed as follows. Given the left and right boundary values $((\frac{\partial^{n-1}u}{\partial x^{n-1}})^L)_{i+\frac{1}{2}}$ and $((\frac{\partial^{n-1}u}{\partial x^{n-1}})^R)_{i-\frac{1}{2}}$ of cell i , we will use the following methods to compute the cell average of $\frac{\partial^n u}{\partial x^n}$:

$$(\frac{\partial^n u}{\partial x^n})_i = \frac{1}{\Delta x} [(\frac{\partial^{n-1}u}{\partial x^{n-1}})^L_{i+\frac{1}{2}} - (\frac{\partial^{n-1}u}{\partial x^{n-1}})^R_{i-\frac{1}{2}}]. \quad (1)$$

If the variable u is sufficiently smooth and the reconstructed distribution function $f(x)$ of the variable u from the cell average is the polynomial with r -th degree, we can prove that

$$(\frac{\partial^n u}{\partial x^n})_i = \frac{1}{\Delta x} \int_{x_{i-\frac{1}{2}}}^{x_{i+\frac{1}{2}}} \frac{\partial^n u}{\partial x^n} dx + O(\Delta x^k), \quad (2)$$

Where $k = 2r - 2$ if $n \leq r$, $k = 2r - 2 - (n - r)$ if $n > r$.

If we use the $(2r-1)$ -th order WENO reconstruction methods to approximate the left and right boundary values of $\frac{\partial^n u(x)}{\partial x^n}$ of cell i from $(\frac{\partial^n u}{\partial x^n})_i$, the local truncation errors of the left and right boundary values are

$$(\frac{\partial^n u}{\partial x^n})_{i\pm\frac{1}{2}}^{L,R} = (\frac{\partial^n u(x)}{\partial x^n})(x_{i\pm\frac{1}{2}}) + O(\Delta x^k). \quad (3)$$

We solve the following linear advection equation:

$$PDE : \partial_t q + \partial_x q = 0, \quad (4)$$

$$IC : q(x, t = 0) = q_0(x). \quad (5)$$

The numerical solution is computed with periodic boundary condition in the spatial domain $0 \leq x \leq 1$. We compute the PDE at the output time $t = 1$ and courant number coefficient is 0.9. Table 1 and Table 2 show the convergence study for fifth-order ADER scheme with piece-wise parabolic reconstruction and we can find the numerical results achieves the desired accuracy.

Table 1: errors of 5-th order ADER scheme using the 5-th WENO reconstruction with $q_0(x) = \sin(2\pi x)$

Total number of grid	L_1 error	Order
50	1.608552288829834E-6	-
100	4.488081727149761E-8	5.16
200	1.350276059002703E-9	5.05
400	4.165486974880975E-11	5.01

Table 2: errors of 5-th order ADER scheme using the 5-th WENO reconstruction with $q_0(x) = \sin^4(\pi x)$

Total number of grid	L_1 error	Order
50	5.808261762126144E-5	-
100	2.676289832073615E-6	4.44
200	3.077777759161518E-8	6.44
400	3.745013304027228E-10	6.36
800	6.896507975139292E-12	5.76

We also use the ADER scheme using the repeated reconstruction methods for computing the high-order spatial derivatives to compute the problem which initial condition is the square wave and find the results are very good.

Conclusion: We use the repeated reconstruction methods for computing the high-order spatial derivatives and the results are good.

¹The institute of applied physical and computational mathematics

Beijing, China

cheng_junbo@iapcm.ac.cn

The project is supported by Science and Technology Foundation of Academy of Engineering Physics of China (No. 2009A09004)

A Unified Approach for High Order Finite-Volume and Discontinuous Galerkin Schemes for PDE with Stiff Source Terms

M. Dumbser¹

Keywords: hyperbolic balance laws, stiff source terms, unstructured meshes, local space-time DG predictor-corrector scheme, $P_N P_M$ schemes

Minisymposia: Finite volume and discontinuous Galerkin schemes for stiff source term problems

Abstract In this talk we present a new unified approach of general $P_N P_M$ schemes on unstructured meshes in two and three space dimensions for the solution of hyperbolic balance laws with stiff source terms. The new $P_N P_M$ approach uses piecewise polynomials u_h of degree N to represent the data in each cell. For the computation of fluxes and source terms, another set of piecewise polynomials w_h of degree $M \geq N$ is used, which is computed from the underlying polynomials u_h using a reconstruction or recovery operator. The $P_N P_M$ method contains classical high order finite volume schemes ($N = 0$) and high order discontinuous Galerkin (DG) finite element methods ($N = M$) just as two particular special cases of a more general class of numerical schemes. Our method uses a novel high order accurate one-step time discretization, based on a local space-time discontinuous Galerkin predictor, which is also able to solve PDE with stiff source terms. We show that our method is asymptotic preserving for a linear model system. We also present results for non-conservative PDE and for PDE systems with higher order derivatives.

We will show applications to several hyperbolic balance laws with stiff source terms, including the relaxation system of Jin and Xin, the resistive relativistic MHD equations, the Baer-Nunziato model for compressible multi-phase flows as well as the chemically reacting compressible Navier-Stokes equations.

References

- [1] M. Dumbser, D. Balsara, E.F. Toro, and C.D. Munz. A unified framework for the construction of one-step finite-volume and discontinuous Galerkin schemes. *Journal of Computational Physics*, 227:8209–8253, 2008.
- [2] M. Dumbser, C. Enaux, and E.F. Toro. Finite volume schemes of very high order of accuracy for stiff hyperbolic balance laws. *Journal of Computational Physics*, 227:3971–4001, 2008.

- [3] M. Dumbser, A. Hidalgo, M. Castro, C. Pars and E.F. Toro. FORCE Schemes on Unstructured Meshes II: Non-Conservative Hyperbolic Systems. *Computer Methods in Applied Mechanics and Engineering*, 199:625–647, 2010.
- [4] M. Dumbser and O. Zanotti. Very high order PNPM schemes on unstructured meshes for the resistive relativistic mhd equations. *Journal of Computational Physics*, 228:6991–7006, 2009.
- [5] A. Hidalgo and M. Dumbser. ADER Schemes for Nonlinear Systems of Stiff Advection-Diffusion-Reaction Equations. *Journal of Scientific Computing*, DOI: 10.1007/s10915-010-9426-6, in press.

¹Laboratory of Applied Mathematics
University of Trento
Via Mesiano, 77
I-38123 Trento (TN), Italy
michael.dumbser@ing.unitn.it

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Accuracy enhancement of discontinuous Galerkin methods for stiff source terms

J.K. Ryan,¹ L. Ji¹ and Y. Xu²

Keywords: Numerical Methods, Hyperbolic Equations, discontinuous Galerkin, Accuracy Enhancement

Minisymposia: Finite volume and discontinuous Galerkin schemes for stiff source term problems

Abstract Discontinuous Galerkin (DG) methods exhibit "hidden accuracy" that makes the superconvergence of this method an increasingly popular topic to address. Previous work has implemented a convolution kernel approach that allows us to improve the order of accuracy from $k+1$ to order $2k+m$ for time-dependent linear convection-diffusion equations, where k is the highest degree polynomial used in the approximation and m depends upon the choice of the numerical flux. Additionally, we are able to obtain similar convergence for smooth solutions to nonlinear hyperbolic equations [3, 4]. In this talk, we discuss how this accuracy enhancing technique may be applied to linear hyperbolic equations with stiff source terms,

$$u_t + \sum_{i=1}^d a_i u_{x_i} + a_o u = \psi(u) \quad (\mathbf{x}, t) \in \Omega \times (0, T], \quad (1)$$

$$u(\mathbf{x}, 0) = u_0(\mathbf{x}). \quad (2)$$

For simplicity, we always consider the domain Ω to have unit length in each coordinate direction.

This convolution kernel approach to accuracy enhancement utilizes information contained in the negative-order norm. Until now there has been relatively little work on error estimates in the negative-order norm, which is related to capturing the superconvergent points. The local postprocessing technique that makes use of the information contained in the negative-order norm was originally developed by Bramble and Schatz [1] in the context of continuous finite element methods for elliptic problems. They demonstrated that it is possible to construct a better approximation by convoluting the finite element solution with a local averaging operator in the neighborhood of a point x , where the convergence in the negative-order norms was higher than L^2 norm. Cockburn, Luskin, Shu, and Süli [2] established a framework to apply this technique to linear hyperbolic equations in the context of the Discontinuous Galerkin methods. Numerical experiments showed that the post-processing had a positive impact on nonlinear hyperbolic equations. Furthermore, this technique is labelled as a smoothness-increasing accuracy-conserving (SIAC) filter by Ryan et al. and was extended to nonuniform meshes, higher-order derivatives, and as a filtering technique to improve the visualization of streamlines [5, 6, 7].

In this talk, we discuss the accuracy enhancing capabilities of this SIAC filter for DG solutions to hyperbolic equations with smooth source terms. We discuss theoretical and computational results and the possibilities for accuracy enhancement.

References

- [1] J.H. Bramble and A.H. Schatz, Higher order local accuracy by averaging in the finite element method, *Mathematics of Computation*, **31**, 94-111, 1977.
- [2] B. Cockburn, M. Luskin, C.-W. Shu, E. Süli, Enhanced accuracy by post-processing for finite element methods for hyperbolic equations, *Mathematics of Computation*, **72**, 577-606, 2003.
- [3] L. Ji, Y. Xu, and J.K. Ryan, Accuracy Enhancement for the linear convection-diffusion equation in multiple dimensions. Submitted.
- [4] L. Ji, Y. Xu, and J.K. Ryan, Negative Order Norm Estimates for Nonlinear Hyperbolic Conservation Laws. In preparation.
- [5] J.K. Ryan and B. Cockburn, Local Derivative Post-processing for the discontinuous Galerkin method, *Journal of Computational Physics*, **228**, 8642 - 8664, 2009.
- [6] J.K. Ryan, C.-W. Shu, and H. Atkins, Extension of a post-processing technique for the discontinuous Galerkin method for hyperbolic equations with application to an aeroacoustic problem, *SIAM Journal on Scientific Computing*, **26**, 821-843, 2005.
- [7] M. Steffan, S. Curtis, R.M. Kirby, and J.K. Ryan, Investigation of smoothness enhancing accuracy-conserving filters for improving streamline integration through discontinuous fields, *IEEE-TVCG*, **14**, 680-692, 2008.

¹Delft Institute of Applied Mathematics
Delft University of Technology
Mekelweg 4, 2628 CD Delft, Netherlands
{j.k.ryan,L.Ji}@tudelft.nl
<http://ta.twi.tudelft.nl/NW/users/ryan/>

²Department of Mathematics
University of Science and Technology of China
Hefei, Anhui 230026, P.R. China
yxu@ustc.edu.cn

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Novel shock-capturing schemes for Discontinuous Galerkin methods

E.Casoni¹ J.Peraire² A.Huerta¹

Keywords: shock-capturing; high-order; artificial diffusion; discontinuous functions

Minisymposia: Finite volume and discontinuous Galerkin schemes for stiff source term problems.

Parallel Session: High order methods for hyperbolic conservation laws

Abstract In the last decades, discontinuous Galerkin (DG) methods have centered the attention of many researches. The success of DG in hyperbolic problems has driven many studies for nonlinear conservation laws and convection-dominated problems. Among all the advantages of DG, their inherent stability and local conservation properties are relevant. Moreover, DG methods are naturally suited for high-order approximations. Actually, in recent years it has been shown that convection-dominated problems are no longer restricted to low-order elements. In fact, highly accurate numerical models for High-Fidelity predictions in CFD are necessary.

Two shock-capturing techniques are presented and discussed. First, a novel and simpler strategy based on discontinuous basis functions is presented. Second, a most commonly approach based on the introduction of artificial viscosity is also presented. The former incorporates a set of basis functions which may vary from piecewise constant to a continuous representation of the solution within every element. Non-constant dissipation proportional to the smoothness of the solution is introduced inside the DG element. This technique allows the shock to be contained within a single element, obtaining shock thickness less than the element size, specially for large high-order elements, [1]. The latter induces a shock width of order $\mathcal{O}(h^k)$ with $1 \leq k \leq p$, which improves the width of common artificial diffusion techniques of order $\mathcal{O}(h/p)$, see for instance [2]. The study extends the accurate one-dimensional artificial viscosity approach proposed in the earlier work [3] to triangular multidimensional meshes. Both approaches prove the efficiency of large high-order DG elements in the presence of shocks.

References

- [1] E. CASONI, J. PERAIRE, AND A. HUERTA. A simple shock-capturing technique for high-order discontinuous Galerkin methods. *In preparation*
- [2] P.-O. PERSSON AND J. PERAIRE. Sub-cell shock capturing for discontinuous Galerkin methods. *Proc. of the 45th AIAA Aerospace Sciences Meeting and Exhibit. AIAA-2006-0112*. Reno, Nevada.

- [3] E. CASONI, J. PERAIRE, AND A. HUERTA. One-Dimensional Shock- Capturing for High-Order Discontinuous Galerkin Methods. *vol. 14 of ECCOMAS Multidisciplinary Jubilee Symposium. Computational Methods in Applied Sciences*. Springer Netherlands, 2009.

¹Laboratori de Calcul Numeric (LaCàN)
Departament de Matemàtica Aplicada 3
Universitat Politècnica de Catalunya-BarcelonaTech, Spain
{eva.casoni, antonio.huerta}@upc.edu
<http://www-lacan.upc.edu/>

²Aerospace Computational Design Laboratory
Massachusetts Institute of Technology, U.S.A.
peraire@mit.edu
<http://www.acdl.mit.edu/>

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Numerical Solution of Viscous and Thermally Conductive Gaseous Flows Via Hyperbolic Maximum Entropy Moment Closures

C. P. T. Groth¹

Minisymposia: Finite volume and discontinuous Galerkin schemes for stiff source term problems

Abstract The numerical solution of the first-order hyperbolic systems of weak conservation equations that result from maximum-entropy moment closures for the Boltzmann equation of gas-kinetic theory is considered. The maximum-entropy closures provide hyperbolic transport equations for an extended set of fluid-dynamic properties that, unlike conventional continuum descriptions, can account for the evolution of gases both in and significantly far from local thermodynamic equilibrium. The latter may be encountered in micron-scale flows; highly rarefied flows; and flows with extreme gradients, as are encountered within shock waves. The use hyperbolic moment equations for numerical prediction of gaseous flows can also present several numerical advantages. Although treatment is required for in many cases stiff source terms, the first-order nature of moment equations allows for the prediction of viscous or heat-transfer effects without the need for the computation of second derivatives. This is in contrast to the continuum-limit Navier-Stokes equations which rely on elliptic operators for the prediction of these phenomena. Numerical solutions to first-order hyperbolic equations are less sensitive to grid irregularities which often result from adaptive-mesh-refinement or embedded-boundary techniques. The lower requirements on the order of the derivatives that must be approximated during a computation also means that numerical schemes for moment equations can gain an extra order of spatial accuracy for the same reconstruction stencil when compared to equations requiring the use of second-order derivatives. In general, numerical methods have been devised that are well-suited for solution of hyperbolic conservation laws (e.g., Godunov-type finite-volume or discontinuous-Galerkin schemes) while other methods have been developed that are more appropriate for the solution of elliptic or parabolic systems. However, the development of schemes which handle the different natures of terms in conservation equations of mixed type with equal elegance have proven to be somewhat more difficult to develop. For all these reasons, the purely hyperbolic nature of moment equations is highly desirable.

In this study, a parallel, implicit, adaptive mesh refinement (AMR), Godunov-type, upwind, finite-volume scheme is described for the solution of the 10-moment equations with stiff source terms of the Gaussian closure and applied to the prediction of a range of three-dimensional, non-equilibrium, micro-scale, gaseous flows. The Gaussian maximum-entropy closure and provides a fully-realisable and strictly

hyperbolic description of non-equilibrium gaseous flows that is valid from the near-equilibrium or continuum limit, through the transition regime, up to the near-collisionless or free-molecular flow limit. Although this somewhat simplified mathematical model does not incorporate the effects of heat transfer, it has been shown to accurately describe non-equilibrium momentum transport for a range of micro-scale flows and is very representative of other higher-order closures. The proposed upwind scheme makes use of Riemann-solver-based flux functions and limited linear reconstruction to provide accurate and monotonic solutions, even in the presence of large solution gradients and/or under-resolved solution content on three-dimensional, multi-block, body-fitted, hexahedral mesh. A block-based AMR strategy is adopted herein to allow for local refinement of the computational mesh. A rather effective and scalable parallel implicit time-marching scheme based on a Jacobian-free inexact Newton-Krylov-Schwarz (NKS) approach with additive Schwarz preconditioning and domain partitioning following from the multi-block AMR mesh is used for the solution of the non-linear ordinary-differential equations, including stiff source terms, that result from the application of the upwind spatial discretization procedure. Details are given of the Gaussian closure, its recovery of the continuum limit, and the proposed finite-volume method. Numerical results for several canonical flow problems demonstrate the robustness of the parallel implicit, finite-volume scheme for accurately predicting fully three-dimensional, non-equilibrium, gaseous flow behaviour. Extension of these ideas to viscous thermally conducting flow is also considered by examining approximate numerical solutions to the hyperbolic system with stiff source terms of a higher-order maximum-entropy closure for a one-dimensional gas using the proposed finite-volume solution scheme.

Keywords: gas kinetic theory, moment closures, hyperbolic treatments of viscous and thermally conductive, finite-volume methods, adaptive mesh refinement (AMR)

Mathematics Subject Classification 2000: 76M10, 82B40, 35L60

¹University of Toronto Institute for Aerospace Studies
4925 Dufferin Street, Toronto, Ontario, M3H 5T6, Canada
groth@utias.utoronto.ca
<http://arrow.utias.utoronto.ca/~groth>

Numerical Methods for Hyperbolic Equations: Theory and Applications

An international conference to honour Professor E.F. Toro
in the month of his 65th birthday

Minisymposia

Methods and models for
biomedical problems

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

A fully adaptive numerical approximation for a two-dimensional epidemic model with nonlinear cross-diffusion

Stefan Berres¹ and Ricardo Ruiz-Baier²

Minisymposia: Methods and models for biomedical problems

Abstract An epidemic model is formulated by a reaction-diffusion system where the spatial pattern formation is driven by cross-diffusion. Whereas the reaction terms describe the local dynamics of susceptible and infected species, the diffusion terms account for the spatial distribution dynamics. For both self-diffusion and cross-diffusion nonlinear constitutive assumptions are suggested. To simulate the pattern formation two finite volume formulations are proposed, which employ a conservative and a non-conservative discretization, respectively. An efficient simulation is obtained by a fully adaptive multiresolution strategy. Numerical examples illustrate the impact of the cross-diffusion on the pattern formation.

Keywords: Epidemic model, reaction-diffusion equation, cross-diffusion, fully adaptive multiresolution, SIR model, Turing instability

Mathematics Subject Classification 2000: 35K55, 92D30, 65M08

References

- [1] S. BERRES, R. BÜRGER AND H. FRID, *Neumann problems for quasi-linear parabolic systems modeling polydisperse suspensions*, SIAM J. Math. Anal. 38 (2006) 557–573.
- [2] M. BENDAHMANE AND M. SEPÚLVEDA, *Convergence of a finite volume scheme for nonlocal reaction-diffusion systems modelling an epidemic disease*, Disc. Cont. Dynam. Syst. - Series B 11 (2009) 823–853.

¹Departamento de Ciencias Matemática y Física, Facultad de Ingeniería,
Universidad Católica de Temuco, Temuco, Chile.
`sberres@uct.cl`

²Modeling and Scientific Computing CMCS-MATHICSE-SB,
École Polytechnique Fédérale de Lausanne, CH-1015 Lausanne, Switzerland.
`ricardo.ruiz@epfl.ch`

Modeling and simulation of the hydrodynamics-biology coupling in a raceway.

A.-C. Boulanger^{1,2} J. Sainte-Marie^{2,3}

Keywords: Navier-Stokes equations, Free surface stratified flows, Multilayer system, Kinetic scheme, Droop model, Raceway, Hydrodynamics and biology coupling.

Minisymposia: Advanced numerical methods in biomathematics

Abstract The coupling between hydrodynamics and biology occurring in oceans, lakes and industrial bioreactors is complex to model and simulate. Here we concentrate on algae growth in aquaculture for biofuel production that is an acute optimization problem for industrials. In this context, the coupling between hydrodynamics and biology has mostly been studied – with simple hydrodynamics models – in photobioreactors [1] that are narrow closed tubes in which a turbulent flow transports the algae. We are rather interested in situations where the algae culture takes place in small circular ponds called raceways, used for the intensive outside culture of algae. As far as we know, such studies have only been carried out in simplified situations [2] that are not fully adapted to the problem we address.

The Navier-Stokes equations give a complete description of hydrodynamics. Nevertheless, they are computationally demanding. Thus, the shallow water assumption is often invoked when considering shallow depth flows, adopting an averaged description. But raceways where the flow is forced by a paddlewheel-like move, are highly heterogeneous along the depth dimension, regarding temperature, light intensity or nutrients availability. We can neither content ourselves with a model that assume uniformity along the vertical dimension nor with a superposition of shallow water models that would prevent exchanges of fluid between the layers. Therefore we use for hydrodynamics a multilayer Saint-Venant model that allows mass exchanges [4, 3].

For the biological part, we have to deal with a great amount of existing models in the literature. We choose to restrict ourselves to one particular species, the diatoms whose evolution is governed by three variables : the phytoplanktonic carbon X (of particular interest for biofuel production), the extracellular nutrients S , but also the intracellular nitrogen N , that can be seen as a nutrient storage pool in the algae. We then build an improved Droop model [5] that includes light effect on algae growth. Our biological system reads

$$\begin{aligned}\frac{\partial \rho X}{\partial t} + \nabla \cdot (\rho \underline{u} X) &= \nu X \Delta X + \rho(\mu(q, I)X - RX) \\ \frac{\partial \rho S}{\partial t} + \nabla \cdot (\rho \underline{u} S) &= \nu S \Delta S - \rho \lambda(S, q)X \\ \frac{\partial \rho N}{\partial t} + \nabla \cdot (\rho \underline{u} N) &= \nu N \Delta N + \rho(\lambda(S, q)X - RN),\end{aligned}$$

with $q = N/X$. ρ is the fluid density and $\mathbf{u}$ the water velocity field both coming from the Navier-Stokes system. I is the light intensity, R is a respiration coefficient, and the growth coefficient write ($Q_0, Q_l, K_s, \tilde{\mu}$ being constants):

$$\mu(q, I) = \tilde{\mu} \frac{I}{I + K_{sI} + \frac{I^2}{K_{iI}}} \left(1 - \frac{Q_0}{q}\right), \quad \lambda(S, q) = \bar{\lambda} \frac{S}{S + k_S} \left(1 - \frac{q}{Q_l}\right)$$

From these equations, we derive a multilayer system similar to what has been proposed in [4, 3]. A kinetic interpretation of the whole system will result in an efficient numerical scheme. The choice for the kinetic solver is encouraged by two main arguments. First of all, we show that we have good properties: conservativity, stability, maximum principle on the tracers. Second of all, since hyperbolicity of the hydrodynamics system has not been proven yet in the general case, eigenvalues are not accessible.

We enforce accuracy of our method by finding analytical stationary solutions for a specific biological model immersed in the hydrostatic Euler equations with free surface. Then we demonstrate through 2d and 3d numerical simulations that our approach is capable of discriminating between situations of moving water or calm pond, choosing an optimal water height or proposing initial conditions for the biological variables. Moreover, we exhibit that a posteriori treatment of our velocity fields can provide lagrangian trajectories of algae and the resulting light patterns, that we hope may improve the choices of more adapted biological systems.

References

- [1] L. Lardon, A. Helias, B. Sialve and O. Bernard. Life-Cycle Assessment of Biodiesel Production from Microalgae. *Environmental Science & Technology*, **43**(17), 6475–6481, 2009.
- [2] S.C. James and V. Boriah. Modeling algae growth in an open-channel raceway. *Journal of Computational Biology*, **7**, 895–906, 2010.
- [3] E. Audusse, M.-O. Bristeau, B. Perthame, and J. Sainte-Marie. A multilayer Saint-Venant system with mass exchanges for Shallow Water flows. Derivation and numerical validation. *ESAIM: M2AN*, **45**:169–200, 2011.
- [4] E. Audusse, M.-O. Bristeau, M. Pelanti, and J. Sainte-Marie. Approximation of the hydrostatic Navier-Stokes system for density stratified flows by a multilayer model. Kinetic interpretation and numerical validation. *Accepted for publication in J. Comp. Phys.*, doi: 10.1016/j.jcp.2011.01.042, 2011.
- [5] K. Lange and F.J. Oyarzun. The attractiveness of the Droop equations. *Mathematical Biosciences*, **111**(2), 261–278, 1993.

¹Jacques Louis-Lions Laboratory, University Pierre et Marie Curie
75005, Paris, France
anne-celine.boulanger@inria.fr

²INRIA Paris-Rocquencourt, France
jacques.sainte-marie@inria.fr

³Saint-Venant Laboratory, Chatou, France

A simple model of filtration and macromolecule transport through microvascular walls

L. Facchini¹ A. Bellin² E.F. Toro²

Keywords: filtration, macromolecule transport, microvascular walls

Minisymposia: Methods and models for biomedical problems

Abstract The walls of microvessels play an important role in maintaining the equilibrium between intravascular and extravascular fluid compartments. Under normal conditions the vessel walls are nearly impermeable to macromolecules, while lipophilic species and small hydrophilic substances are allowed to cross the walls and reach the surrounding tissue. Fluid flow and transport of dissolved molecules through the walls depends on permeability and diffusivity of the membrane composing the wall. Several alterations of these processes have been observed resulting in leaking of macromolecules, which is typically attributed to the reduction of oncotic pressure, or inflammatory processes altering the endothelial structure.

In this work we investigate the role of an increased blood pressure as the driving force for leaking of macromolecules. The microvessel wall is assumed to be composed of two layers with different permeability and porosity, as assumed in previous studies on fluid flow and macromolecules transport in heteroporous membranes. Variability in the porosity and tortuosity of the porous material leads to variations in dispersivity.

In this preliminary analysis we assume permeability, porosity and tortuosity to be constant within the single layer, but variable through the layers composing the microvessel wall. With this simplified, yet realistic, model we obtain closed-form steady-state solutions for the fluid flow and solute transport through the microvessel walls, which can be used for a preliminary analysis of leaking of macromolecules due to an increased blood pressure.

With this simple model we interpret experimental data available in the literature.

References

- [1] C.C. MICHEL, F.E. CURRY. Microvascular permeability. *Physiological Reviews* **79**(3), 703–761, 1999.
- [2] C.C. MICHEL, M.E. PHILLIPS. Steady-state fluid filtration at different capillary pressures in perfused frog mesenteric capillaries. *The Journal of Physiology* **388**, 421–435, 1987.

¹Department of Mathematics
University of Trento (Italy)
Via Sommarive 14
38123 Povo (TN), Italy

²Department of Civil and Environmental Engineering
University of Trento (Italy)
via Mesiano 77
38050 Mesiano di Povo (TN), Italy

Assessment of numerical methods for blood flow in veins

G. Montecinos¹ L. O. Müller¹ E. F. Toro¹

Minisymposia: Methods and models for biomedical problems

Abstract The aim of this work is to systematically assess a range of modern numerical methods for simulating the blood flow in veins. As a mathematical model we adopt the one-dimensional system of hyperbolic balance laws, as formulated in [1]. See also [2] and references therein. The unknowns of the problem are cross-sectional area, cross-sectional averaged velocity and pressure. A closure condition is required, as there are less equations than unknowns. The tube law is a closure condition that relates cross-sectional area, and thus the compliance of veins, to pressure. In this work we pay special attention to the treatment of source terms and junctions, see [3] and [4]. It is known that endothelial cells react to mechanical signals, which in turn determines their configuration and behaviour [5]. Hence the correct representation of wave propagation plays a major role. It is therefore useful to apply accurate numerical methods to solve the partial differential equations.

Here we implement and systematically assess a broad range of modern numerical methods to solve the adopted model system. We start with two second-order shock-capturing finite volume schemes, namely the WAF method [6] and the MUSCL-Hancock method [7]. We then proceed to higher order schemes in the frameworks of finite volumes and discontinuous Galerkin finite elements. In particular we assess the ADER approach [8, 9], a fully discrete, one-step scheme of arbitrary order of accuracy in both space and time. The ADER approach requires a non-linear spatial reconstruction step and a solver for the generalized (or high order) Riemann problem. Various available options for the generalized Riemann solver are implemented. See [10], [11] and [12]. In addition we also implement semi-discrete finite volume and DG schemes using TVD Runge-Kutta schemes [13], and the semi-implicit scheme of Casulli [14].

To conclude we present preliminary numerical results for a simplified venous network associated with the extracranial venous system.

Keywords: hemodynamics, collapsible tubes, high order methods

References

- [1] E. F. TORO AND A. SIVIGLIA. Flow in collapsible tubes with discontinuous mechanical properties: mathematical model and exact solutions. *Proc. Roy. Soc. London*, submitted (2011).

- [2] A. QUARTERONI AND M. TUVERI AND A. VENEZIANI. Computational vascular fluid dynamics: problems, models and methods. *Computing and Visualization in Science*. **2**, 163–197, 2000.
- [3] J. FULLANA AND S. ZALESKI. A branched one-dimensional model of vessel networks. *Journal of Fluid Mechanics*. **621**, 183–204, 2009.
- [4] E. MARCHANDISE AND P. FLAUD. Accurate modelling of unsteady flows in collapsible tubes. *Computer Methods in Biomechanics and Biomedical Engineering*. **13**, 279–290, 2010.
- [5] P. ZAMBONI, R. GALEOTTI, E. MENEGATTI, A. M. MALAGONI, S. GIANESINI, I. BARTOLOMEI, F. MASCOLI, AND F. SALVI. A prospective open-label study of endovascular treatment of chronic cerebrospinal venous insufficiency. *Journal of Vascular Surgery*. **50**, 1348–1358, 2009.
- [6] E. F. TORO. A Weighted Average Flux method for hyperbolic conservation laws. *Proceedings of the Royal Society of London. Series A. Mathematical and Physical Sciences*. **423**, 401–418, 1989.
- [7] B. VAN LEER Upwind-difference methods for aerodynamic problems governed by the Euler equations, in: Bjorn E. Engquist, Stanley Osher and Richard C. J. Somerville (Eds.), *Large-Scale Computations in Fluid Mechanics*, AMS, *Lectures in Applied Mathematics*. **22**, 327–336, 1985.
- [8] E. F. TORO, R. C. MILLINGTON AND L. A. M. NEJAD. Towards very high-order Godunov schemes. In Godunov Methods: Theory and Applications. Edited Review. E. F. Toro (Editor). *Kluwer Academic/Plenum Publishers. Conference in Honour of S K Godunov*. **1**, 897–902, New York, Boston and London, 2001.
- [9] T. SCHWARTZKOPFF, C. D. MUNZ, E. F. TORO AND R. C. MILLINGTON. The ADER approach in 2D. In Discrete Modelling and Discrete Algorithms in Continuum Mechanics. *Berlin, Germany. T Sonar (editor)*. **1**, pages 207–216, 2001.
- [10] V. A. TITAREV, E. F. TORO. ADER: Arbitrary High Order Godunov Approach. *Journal of Scientific Computing*. **17**, 609–618, 2002.
- [11] C. E. CASTRO AND E. F. TORO. Solvers for the high-order Riemann problem for hyperbolic balance laws. *Journal of Computational Physics*. **227**, 2481–2513, 2008.
- [12] M. DUMBSER AND E. F. ENAUX, C. TORO. Finite volume schemes of very high order of accuracy for stiff hyperbolic balance laws. *Journal of Computational Physics*. **227**, 3971–4001, 2008.
- [13] B. COCKBURN AND C. W. SHU. Runge-Kutta discontinuous Galerkin methods for convection dominated problems. *Journal of Scientific Computing*. **16**, 173–261, 2001.
- [14] V. CASULLI. A high-resolution wetting and drying algorithm for free-surface hydrodynamics. *Journal for Numerical Methods in Fluids*. **60**, 391–408, 2009.

¹Laboratory of Applied Mathematics
University of Trento
Faculty of Engineering. 38123 Trento, Italy
`lucas.mueller@ing.unitn.it`

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Reduced models for the Fluid-Structure Interaction problem

C. Vergara¹

Keywords: Finite Elements, absorbing conditions, fluid-structure interaction, one-dimensional models

Minisymposia: Methods and models for biomedical problems

Abstract The *fluid-structure interaction* (FSI) problem occurs whenever a fluid and a solid exchange energy through a FSI interface. The hyperbolic nature of the FSI problem allows to capture waves' propagation. This is the case, for example, of the blood flowing in the cardiovascular vessels, which inspires the present work. In that case, a pressure wave travels downstream the arterial tree. One of the major topic in FSI simulations in haemodynamics is the prescription of suitable boundary condition at the outlets, in order to avoid non-physiological wave reflections. With this aim, one classical solution is the coupling with a reduced one-dimensional (1D) model. The first goal of this work consists in studying the coupling between the FSI and the 1D problems, by means of defective Robin (resistance) boundary conditions.

The second goal of this work is related to the study of another reduced model for the FSI problem, given by considering a membrane structure instead of a thick one. In this context, an algorithm based on the solution of just one fluid problem at each time step has been introduced in [1]. Here we want to provide a comparison of the solutions obtained with a thick structure and with a membrane, in relation to the aims of the simulations and to the computational costs involved.

References

- [1] F. NOBILE AND C. VERGARA. An effective fluid-structure interaction formulation for vascular dynamics by generalized Robin conditions. *SIAM J Sc Comp*, **30**(2), 731–763, 2008.

¹Department of Information Engineering and Mathematical Methods
University of Bergamo
Viale Marconi 5, 24044 Dalmine (BG), Italy
`christian.vergara@unibg.it`

Numerical Methods for Hyperbolic Equations: Theory and Applications

An international conference to honour Professor E.F. Toro
in the month of his 65th birthday

Minisymposia

Multiphase flow and porous media

Dynamics of submerged gravitational granular flows

A. Armanini¹ M. Dumbser¹ E. Nucci¹

Keywords: debris flows, granular flows, uniform flow, numerical methods for differential-algebraic equations

Minisymposia: Multiphase flows and porous media

Abstract From a mathematical point of view, debris flows can be treated as gravitational granular flows in a water matrix. In absence of cohesive materials, the rheology of the liquid phase is Newtonian, while the rheology of the granular phase is more complex. In the gravitational flows analyzed in this paper, we are considering the motion of a granular flow flowing on an immobile bed composed by the same material, in equilibrium state. In this condition the presence of a stratification of the rheological mechanisms across the flow depth has been observed [3]. Near the free surface the granular flow is characterized by instantaneous contact among the particles (*collisional regime*), while near the immobile bed the granular flow is dominated by prolonged contacts among particles (*frictional regime*). In general, the tensor of the granular phase can be assumed to be the composition of two tensors, one, T_{ij}^{g-coll} , representative of the stresses exchanged with a collisional mechanism, one, T_{ij}^{g-fric} , representative of the stresses expressed by a frictional mechanism. The rheology of the *collisional regime* is well described by means of the *kinetic theory*, in which the granular flow is assimilated to a dense gas [4]. The rheology of the frictional regime, however, is still under debate and a persuasive formulation of this regime does not exist yet. Often, one assumes that frictional stress is Coulombian, but experimental evidence shows that this hypothesis is rather limitative [2]. A successful scheme of this kind was recently proposed by the GDR-Midi group [6], but this model is based on some parameters, the determination of which is doubtful and it does not contain a suitable formulation for the granular pressure. Armanini [1] proposes a reinterpretation of the model, as weighted average of a Coulombian stress (dependent on static friction angle at the bottom), and of a dynamic stress, represented by a dynamic friction angle. Besides in the same work also a relation for the granular pressure is given.

In the present paper, we are presenting a generalization of this model its numerical solution for two-dimensional, uniform flows. Under this hypothesis the system of differential equations that describes the flow is:

$$\begin{aligned} \frac{\partial \tau_{12}^g}{\partial x_2} &= -(1 + C\Delta)\rho_w g \frac{\partial z}{\partial x_1} \\ \frac{\partial p^g}{\partial x_2} &= -C\Delta\rho_w g \frac{\partial z}{\partial x_1} \\ \frac{\partial}{\partial x_2} \left(-f_4 \sqrt{\Theta} d_p \frac{\partial \Theta}{\partial y} \right) &= f_2 \sqrt{\Theta} d_p \left(\frac{\partial u_1}{\partial y} \right)^2 - f_5 \rho_s \frac{\Theta^{1.5}}{d_p} \end{aligned} \quad (1)$$

where d_p is the size of the particles; Θ the granular temperature; u^g the velocity of granular phase; $\Delta = (\rho_s - \rho_w)/\rho_w$; z is the vertical direction; c is the volume concentration of the granular phase; x_1 and x_2 are the longitudinal and the normal directions respectively, and f_1, f_2 and f_4 are functions of the concentration. With the following relationships:

$$\tau_{12}^g = f_2 \sqrt{\Theta} d_p \frac{\partial u_1^g}{\partial y} + p^g \frac{I_0^2}{I_0^2 + I^2} \tan \varphi^{fric} \quad \text{and} \quad p_{12}^g = f_1 \rho_s \Theta + p_{z=0}^g e^{-\frac{(c-c^*)^2}{A}} \quad (2)$$

where C^* is the solid concentration in the bed, and $I = d_p (\partial u_1^g / \partial y) / \sqrt{p_{12}^g / \rho_s}$, while I_0 and A are costants.

The system (1)-(2) can be written as a differential algebraic system of the general form

$$\frac{dE(Q)}{dy} = S(y, Q), \quad (3)$$

with two nonlinear vector functions $E(Q)$ and $S(Q)$. The boundary value problem (1)-(2) is solved on the basis of a shooting method which resolves the DAE (3) numerically. We studied two options: first, using a simple Crank–Nicholson type scheme of the form

$$\frac{E(Q_{n+1}) - E(Q_n)}{\Delta y} - \frac{1}{2} (S(y_n, Q_n) + S(y_{n+1}, Q_{n+1})) = 0, \quad (4)$$

which is solved for Q_{n+1} using a Newton method with an appropriate globalization strategy based on linesearch. As a second option we use a high order *continuous Galerkin* finite element method for the solution of (3), similar to the *discontinuous Galerkin* finite element scheme proposed in [7] for the solution of the ODE systems arising in boundary layer theory. The numerical results obtained from the model (1)-(2) are satisfactory compared with the experimental data.

References

- [1] A.Armanini, 2010, Palermo, *Proc. XXXII Conv. Naz. di Idr. e C.I.*
- [2] A.Armanini, M. Larcher, and L. Fraccarollo, *Ph. Rev. E*, **79**, 5, p.051306, 2009.
- [3] A. Armanini, H. Capart, L.Fraccarollo, M. Larcher, 2005, *J.F.M.*, **532**,269-319.
- [4] S.B. SAVAGE, *Advances in Applied Mechanics*, **24**:289-366, 1984.
- [5] N.F. Carnahan & K. Starling, *J. Chem. Phys.* 51:635-636 (1969).
- [6] GDR MiDi, 2004. On dense granular flows.*Eur. Phys. J. E* 14,341-365.
- [7] M. Dumbser, *Computers & Fluids*. 2010; **39**:60–76.

¹Department of Civil and Environmental Engineering
University of Trento, via Mesiano, 77, I-38123, Italy
aronne.armanini@unitn.it

Finite Volume Method for Three-Dimensional Particle-Laden Free-Surface Flow[†]

Patricio Bohorquez¹

Keywords: Polyhedral mesh, Volume of Fluid, Finite Volume Method, Thin Film Flow, Equilibrium Theory of Dense Suspension.

Minisymposia: Multiphase flow and porous media

Parallel Session: Shallow water flows

Abstract We present a Finite Volume Method (FVM) for three-dimensional, incompressible, free-surface flow in the presence of non-cohesive, uni-modal, sediment particles near equilibrium. The physical model is based on the mixture (or drift-flux) theory [1]. The continuity equations of the water, sediment and air phases are reformulated in terms of the mixture volumetric velocity $\vec{u}$,

$$\nabla \cdot \vec{u} = 0, \quad (1)$$

resulting in two scalar transport equations as

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\phi \vec{u}) + \nabla \cdot [\phi(1 - \phi) \vec{u}_{r\phi}] = 0, \quad \phi = \beta, \gamma \quad (2)$$

where t is time, $\vec{u}_{r\phi}$ denotes the slip velocity, β is the sediment-volumetric concentration and γ is the phase indicator function employed to capture the free surface. Equation (3) is solved by means of an explicit numerical scheme designed for the multi-dimensional advection equation [2]. The bulk momentum equation is formulated under the assumption of near equilibrium between water and sediment, i.e. $|\vec{u}_{r\beta}| \ll |\vec{u}|$. Under this assumption, the mixture momentum equation reads [3]

$$\frac{\partial \rho \vec{v}}{\partial t} + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot \bar{\bar{\tau}} - \vec{g} \cdot \vec{x} \nabla \rho, \quad (3)$$

where ρ is the mixture density, $\vec{v} := \vec{v}(\vec{u}, \vec{u}_{r\phi})$ is the mixture mass velocity, $\vec{g}$ is the acceleration due to gravity, and $\bar{\bar{\tau}}$ is the generalised stress tensor that accounts for the viscous stress tensor, the diffusion stresses due to the relative motion between phases and the non-isotropic part of the sediment stress tensor. The model reveals that the isotropic part of the bulk stress tensor, i.e. the mixture pressure p , plays the same role as for a single incompressible flow. Consequently, (1)–(3) are discretised and assembled into a block system using a Schur complement formulation [4].

[†]This work has been supported by the Spanish MCyT, Junta de Andalucía and European funds under Projects# DPI2008-06624-C02 and P07-TEP02693

We compare numerical results obtained both in structure and polyhedral meshes for two physical problems: vertical settlers in the presence of free surface [5] and particle-laden flows down a steep inclined plane [6]. With these examples, we show the capability of the numerical scheme to capture discontinuities in ϕ , see figure 1, and to obtain the pressure distribution and the three-dimensional velocity profile. Finally, we also assess the mesh topology influence on the numerical results.

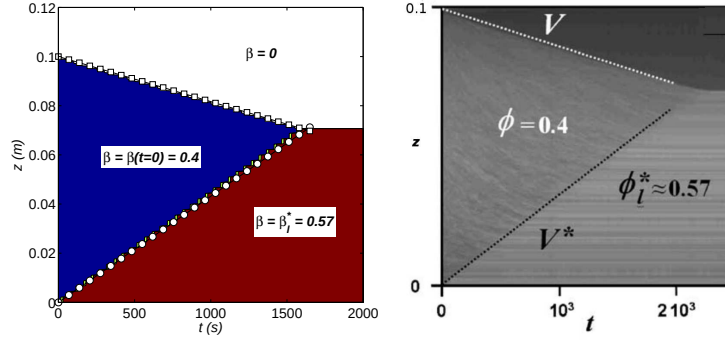


Figure 1: Numerical solution (left) for the volumetric concentration of sediment $\beta(z, t)$ in the experiments (right) by Snabre *et al.* [5].

References

- [1] M. ISHII AND T. HIBIKI. *Thermo-fluid dynamics of two-phase flow*. Springer, 2006.
- [2] S. T. ZALESK. Fully multidimensional flux-corrected transport algorithms for fluids. *J. Comput. Phys.* **31**(3), 335–362, 1979.
- [3] P. BOHORQUEZ. *Study and numerical simulation of sediment transport in free-surface flow*. Ph.D. Thesis, University of Malaga, 2008.
- [4] M. BENZI, G. H. GOLUB AND J. LIESEN. Numerical solution of saddle point problems. *Acta Numerica* **14**:1–137, 2005.
- [5] P. SNABRE, B. POULIGNY, C. METAYER AND F. NADAL. Size segregation and particle velocity fluctuations in settling concentrated suspensions. *Rheol. Acta* **48**:855–870, 2009.
- [6] T. WARD, C. WEY, R. GLIDDEN, A.E. HOSOI AND A.L. BERTOZZI. Experimental study of gravitation effects in the flow of a particle-laden thin film on an inclined plane. *Phys. Fluids* **21**(8): 083305, 2009.

¹Área de Mecánica de Fluidos, Departamento de Ingeniería Mecánica y Minera, Universidad de Jaén, Campus de las Lagunillas, 23071 Jaén, Spain
 patricio.bohorquez@ujaen.es <http://blogs.ujaen.es/prmedina>

Numerical simulation of a bubble rising in still liquids: determination of the instability transition modes[†]

J. C. Cano-Lozano,¹ P. Bohorquez,¹ C. Martínez-Bazán¹

Keywords: Bubble, Hydrodynamic Stability, Volume of Fluid, Finite Volume Method.

Minisymposia: Multiphase flow and porous media

Abstract In this work, we have investigated numerically the transition from straight to zigzag motion during the rise of a single gas bubble of diameter D in a pure-clear stagnant liquid [1], for the limiting case $\rho_g/\rho_l \ll 1$ and $\mu_g/\mu_l \ll 1$, where ρ is density, μ is dynamic viscosity, and subindices g and l denote gas and liquid phases, respectively. The transition is determined in terms of the Reynolds, $Re = \rho_l g^{1/2} D^{3/2}/\mu_l$, and Bond, $Bo = \rho_l g D^2/\sigma$, numbers as set of nondimensional, independent parameters governing the flow dynamics [2], in which g is the acceleration due to gravity and σ is the surface tension. Subsequently, the neutral curve for the onset of zigzag motion is characterized in the $\{Re, Bo\}$ -plane.

This type of flow has been previously studied numerically and experimentally [1, 2, 3]. From the numerical point of view, the terminal velocity and bubble shape are usually computed under the axisymmetric hypothesis, e.g. [2], though the real flow is both three-dimensional and unsteady. On the experimental side, e.g. [1], available data for the neutral curve cover a limited part of the $\{Re, Bo\}$ diagram because of the intrinsic difficulties to the current problem, see dashed line in figure 1(a). Although the neutral curve has been already determined considering a spheroidal shape for the bubble [3], its precise description is still an unresolved problem. Thus, in this work we characterise accurately the above mentioned transition combining several second-order finite volume methods, which have been implemented in open source software, discussing the advantages of each one. We show that the combination of Godunov momentum advection scheme and adaptive quadtree mesh refinement, available in Gerris flow solver [4], performs better than schemes implemented in the interFoam multiphase solver in OpenFOAM. Consequently, the axisymmetric basic flow is characterised by means of Gerris flow solver, see figure 1(b), and the evolution of the bubble shape, terminal velocity, pressure and velocity fields described as functions of Re and Bo in the vicinity of the neutral curve. The stability property of the background flow is subsequently determined by means of three-dimensional numerical simulations [5], showing the loss of the flow axisymmetry and the developing of two counter-rotating vortices, as illustrated in figure 1(c).

[†]This work has been supported by the Spanish MCyT, Junta de Andalucía and European funds under Projects# DPI2008-06624-C02 and P07-TEP02693

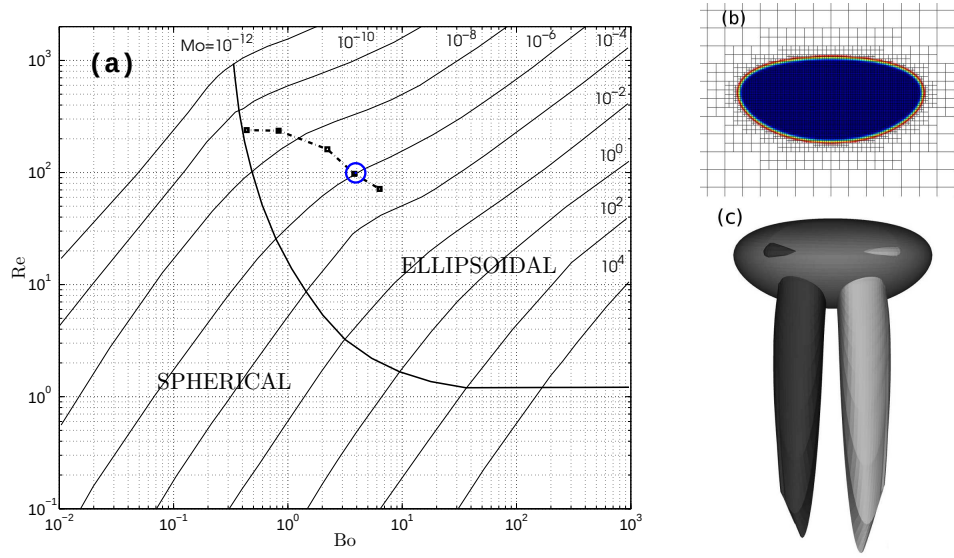


Figure 1: (a) Stability diagram showing the experimental neutral curve [1], dashed line, and our preliminary result, blue circle. Thin solid lines depict the Morton number, $Mo = g\mu_L^4 / \rho_L \sigma^3$, from 10^{-12} to 10^4 . (b) Axisymmetric bubble at $Re = 113.1$ and $Bo = 3.925$ simulated with Gerris flow solver. (c) Streamwise vorticity contours observed in the three-dimensional perturbed flow with axisymmetric bubble shape, corresponding with the circle depicted in (a).

References

- [1] R. ZENIT AND J. MAGNAUDET. Path instability of rising spheroidal air bubbles: A shape-controlled process. *Phys. FLuids* **20**, 061702, 2008.
- [2] J. HUA AND J. LOU. Numerical simulation of bubble rising in viscous liquid. *J. Comput. Phys.* **222**, 769–795, 2007.
- [3] J. MAGNAUDET AND G. MOUGIN Wake instability of a fixed spheroidal bubble. *J. Fluid Mech.* **572**, 311–337, 2007.
- [4] S. POPINET. An accurate adaptative solver for surface-tension-driven interfacial flows. *J. Comput. Phys.* **228**, 5838–5866, 2009.
- [5] P. BOHORQUEZ, E. SANMIGUEL-ROJAS, A. SEVILLA, J. I. JIMÉMEZ-GOZÁLEZ, C. MARTÍNEZ-BAZÁN Stability and dynamics of the laminar wake past a slender blunt-based axisymmetric body. *J. Fluid Mech.* (doi:10.1017/S0022112011000358).

¹Área de Mecánica de Fluidos, Departamento de Ingeniería Mecánica y Minera, Universidad de Jaén, Campus de las Lagunillas, 23071 Jaén, Spain
 jccano@ujaen.es <http://www.fluidsujen.es>

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Numerical simulation of three-dimensional transient variably saturated flow

D. Caviedes-Voulliéme, J. Murillo, P. García-Navarro¹

Keywords: Numerical Methods, Porous Media, Unsaturated Soils

Minisymposia: Multiphase flow and porous media

Abstract

Subsurface flow is an important part of hydrological processes, from infiltration during rain events to surface-subsurface flow interactions in floodplains and hillslopes. Since subsurface flow may occur in unsaturated and saturated conditions, models which are able to represent such conditions in a continuous fashion are necessary. Furthermore, adequate mass conservation in hydrological processes must account for the exchange of surface and subsurface water both in steady and unsteady cases. Hence, a model which allows to approximate the solution of transient variably saturated flow problems, aiming towards surface-subsurface interactions, has been developed.

Flow in variably saturated soils is described by Richards equation. This highly non-linear PDE requires in most cases the use of numerical methods for its solution. The proposed numerical model allows for the solution of the water content and pressure distribution in three dimensional domains by means of an implicit finite volume scheme which adequately conserves mass and represents wetting fronts both in time and space. The model has been developed progressively from one to three dimensions, testing it against an analytical solution of an infiltration problem and by means of test cases. In one dimensional form, several numerical schemes were tested to determine the best numerical approach and determine the response of the model to different configurations and parameters. Validation against the analytical solution shows that the model is able to correctly solve Richards' equation in one, two and three dimensions. Test case solutions show good behavior of the model in all dimensions against particular geometries and different initial and boundary conditions.

References

- [1] CELIA, M., BOULOUTAS, E., ZARBA, R. A General mass-conservative numerical solution for the unsaturated flow equation. *Water Resources Research* **26**(7), 1983-1946, 1990.

- [2] PHOON, K.-K., TAN, T.-S., CHONG, P.-C. Numerical simulation of Richards equation in partially saturated porous media: under-relaxation and mass balance. *Geotechnical and Geological Engineering* **25**(5), 525-541, 2007.
- [3] WARRICK, A., LOMEN, D., YATED, S.,. A generalized solution to infiltration. *Soil Science Society of America Journal* **49**(1), 34-38, 1985.

¹Fluid Mechanics Group
University of Zaragoza
María de Luna 3,
50018 Zaragoza, Spain
`daniel.caviedes@unizar.es`

An all-speed asymptotic preserving scheme for the low Mach number limit of the Euler equations

F. Cordier¹ P. Degond² A. Kumbaro¹

Keywords: Numerical Scheme, Asymptotic Preserving, Low Mach Number, Euler Equations, All Speed Scheme, Compressible Flows, Incompressible Flows.

Minisymposia: Multiphase flows and porous media.

Abstract We present an all-speed asymptotic preserving scheme for the low Mach limit of the full Euler equations.

In the framework of nuclear power plants safety studies, the numerical simulation of multiphase flows is a crucial issue. One of the difficulties is the existence of both compressible and incompressible flow regions, as the sound speed can be much higher in the pure liquid nearly incompressible flow regions than in the multiphase mixture regions. This is why a numerical method for the simulation of multiphase flows must apply equally to both compressible and incompressible flows. This type of scheme is referred to as an “all speed” scheme.

Standard compressible methods do not have this all-speed property and suffer from an important loss of accuracy, a deterioration of the convergence rate, and a timestep dramatically reduced along with the Mach number. As a result, low speed preconditioning techniques have been developed for the computation of nearly incompressible single phase models or two phase flow mixture models. Nevertheless, the divergence-free constraint on the velocity in the low Mach number regime is not respected and can produce robustness problems.

In the past few years, a new class of schemes, “asymptotic preserving” (AP) schemes, have been successfully applied to singular limit problems, as plasmas in the quasineutral limit [1] or Euler equations in a strong magnetic field [2]. Scaling the equations highlights the existence of a parameter $\varepsilon \in [0, O(1)]$, like the Mach number in the Euler equations. Asymptotic preserving schemes provide a consistent discretisation of both the $\varepsilon = O(1)$ equations and the limit system $\varepsilon = 0$.

All-speed asymptotic preserving schemes for the low Mach number limit have already been proposed for the isentropic Euler equations in [3] and [4]. Given the application of our work to thermal power plants, we start extending the work of [3] to the full Euler equations as they represent the convective part of the multiphase model HEM (homogeneous equilibrium model), including an energy equation. Our aim is to propose an all speed solver for general multifluid models.

The key idea of our AP scheme is a specific semi-implicit time-discretisation: the low Mach number stiff terms are divided in two parts, one part being treated explicitly and the other one implicitly. This semi-implicit discretisation is the key to the asymptotic preserving property. Moreover, it allows to be released from the

stringent CFL constraint appearing in the low Mach number regime: timesteps are based on the fluid velocity and not on the acoustic velocity.

The AP scheme also takes into account the change of nature of the Euler equations, hyperbolic in the compressible regime and hyperbolic/elliptic in the low Mach regime, via the resolution of an elliptic equation on the pressure. The construction of the elliptic equation differs from [3] as it involves the energy equation. The divergence-free constraint on the velocity is respected in the low Mach number regime.

The proposed AP scheme is conservative, adapted to a general equation of state, and able to compute steady or unsteady flows.

So far, very promising numerical results have been obtained from a second-order discretisation of the AP scheme. First, our scheme conserves the accurate shock capturing properties of the Riemann solvers. Then, to test the behaviour of the AP scheme in the low Mach regime, the relevant and well-known test-cases of the backwards facing step and the lid driven cavity have been computed. Our AP scheme is able to capture the circulation regions typical of the low Mach number regime, and the dimensions match the results found by other authors.

The numerical results show the good behaviour of our asymptotic preserving scheme in both compressible and nearly incompressible regions. Moreover, the idea can be extended to the multifluid model, which we intend to consider in the future.

References

- [1] DEGOND, P. AND LIU, J.G. AND VIGNAL, M.H. Analysis of an Asymptotic Preserving Scheme for the Euler–Poisson System in the Quasineutral Limit. *SIAM Journal on Numerical Analysis*, vol. 46, p. 1298, 2008.
- [2] DEGOND, P. AND DELUZET, F. AND SANGAM, A. AND VIGNAL, M.H. An Asymptotic Preserving Scheme for the Euler equations in a strong magnetic field. *Journal of Computational Physics*, vol. 228, number 10, pp. 3540–3558, 2009.
- [3] DEGOND, P. AND TANG, M. All speed scheme for the low mach number limit of the Isentropic Euler equations. *Preprint*, 2009.
- [4] HAACK, J. AND JIN, S. AND LIU, J.G. An all-speed asymptotic-preserving method for the isentropic Euler and Navier-Stokes equations. *Preprint*, 2010.

¹CEA-Saclay DEN, DM2S, SFME, LETR F-91191 Gif-sur-Yvette, France.
floraine.cordier@cea.fr, anela.kumbaro@cea.fr

²Institute of Mathematics of Toulouse, Universite Paul Sabatier, 118 route de Narbonne, 31062 Toulouse cedex, France. degond@mip.ups-tlse.fr

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

A New Scheme for Non-Classical Waves with TVB Proof for Scalar 1D-Case

P. Engel¹, C. Chalons² and C. Rohde¹

Keywords: Numerical Methods, Hyperbolic Equations, Non-Classical Shocks, TVB, convergence proof

Minisymposia: Multiphase flows and porous media.

Abstract Conservation laws are used in a wide range of applications. Some of these applications show special phenomena, which can be understood with the help of non-classical waves. For more informations on non-classical waves see for example [2].

One important application is multiphase flow. Let us for example look at a liquid droplet in vapour. The surface of the droplet becomes a non-classical shock in the mathematical model. It is well known, that the criterion of entropy solutions is not enough to get well-posedness in this setting. One has to require an additional algebraic equation, which each non-classical wave has to satisfy. Due to this fact normal finite volume schemes are not able to treat non-classical shocks.

For the sake of simplicity, we write down the model in the scalar one-dimensional case, which is also the case, where the results hold. We have

$$\begin{aligned} u_t + f(u)_x &= 0 && \text{in } \mathbb{R} \times (0, T), \\ u(t=0) &= u_0 && \text{in } \mathbb{R}, \end{aligned}$$

where $u: \mathbb{R} \times [0, T] \rightarrow \mathbb{R}$ is the unknown quantity and $f: \mathbb{R} \rightarrow \mathbb{R}$ is the flux by which u is transported. Since we are interested in non-classical waves, we assume, that f is for example concave-convex. At non-classical waves an additional algebraic relation, the so-called kinetic relation [1], has to hold

$$u_+(t) = \varphi(u_-(t)).$$

Here $u_-(t)$ and $u_+(t)$ are the left and right hand side trace values of $u(\cdot, t)$ at the non-classical wave. $\varphi: \mathbb{R} \rightarrow \mathbb{R}$ is the kinetic function.

In the talk we will introduce a new generalised finite volume scheme, which is able to treat non-classical shocks. The scheme will be written down in a more-dimensional setting for systems of conservation laws. After introducing the scheme, we will proof for the scalar one-dimensional case with only one phase boundary, that the scheme is TVB, and discuss further convergence properties of the scheme.

References

- [1] R. ABEYARATNE AND J. K. KNOWLES. Kinetic relations and the propagation of phase boundaries in solids. *Archive for Rational Mechanics and Analysis* **114**, 119–154, 1991.
- [2] P. G. LEFLOCH. Hyperbolic systems of conservation laws. The theory of classical and nonclassical shock waves. *Lectures in Mathematics ETH Zürich*, Birkhäuser Verlag, Basel, 2002.

¹Institut für Angewandte Analysis und numerische Simulation
University of Stuttgart
Pfaffenwaldring 57, D-70569 Stuttgart, Germany
{patrick.engel, christian.rhode}@mathematik.uni-stuttgart.de

²Université Paris Diderot Paris 7 & Laboratoire Jacques-Louis Lions
U.M.R. 7598, Boîte courrier 187, 75252 Paris Cedex 05, France
chalons@math.jussieu.fr

Modelling Microbial Chemotactic Waves in Saturated Porous Media using Adaptive Mesh Refinement.

S.A.E.G.Falle¹ N.F.Dudley-Ward² M.S.Olson³

Keywords: Numerical Methods, AMR, Porous media, Chemotaxis

Minisymposia: Multiphase flows and porous media

Abstract Microbial transport in groundwater can be strongly influenced by chemotaxis induced by chemical gradients and this can have a significant effect on the rate of degradation of groundwater contaminants ([1]). The main aim of this paper is to describe a method of simulating the propagation of a travelling microbial wave in a contaminated region and the resulting degradation of the contaminant. The presence of the chemotactic term and the relatively small bacterial diffusion means that the wave contains a very sharp wavefront (of the order of millimetres). We therefore use an upwind conservative numerical scheme to obtain accurate and numerically stable solutions. The accuracy of the method is verified by comparisons with an exact one dimensional solution of a simplified problem. The method is then used to simulate the propagation of a realistic chemotactic wave in one dimension. We then use Adaptive Mesh Refinement (AMR) to compute the propagation of chemotactic waves in two dimensions using the simplified model calibrated to give the same wavespeed as the full model. This technique makes it possible to study the propagation of such waves on scales of the order of kilometers, which is typical for contaminated aquifers. The figure shows the propagation of a microbial wave into an initially polygonal contaminated region.

Mathematics Subject Classification 2000: 68-04,76Z99,92B05

References

- [1] R.M. FORD R.W. HARVEY. Role of chemotaxis in the transport of bacteria through saturated porous media, *Advances in Water Resources* **30** (6-7), 1608-1617, 2007.

¹Department of Applied Mathematics, University of Leeds, Leeds LS2 9JT, United Kingdom.

²Groundwater Resources Research, PO Box 2, Kurow, New Zealand.

³Department of Civil, Architectural and Environmental Engineering, Drexel University, 3141 Chestnut Street, Philadelphia, PA 19104, United States

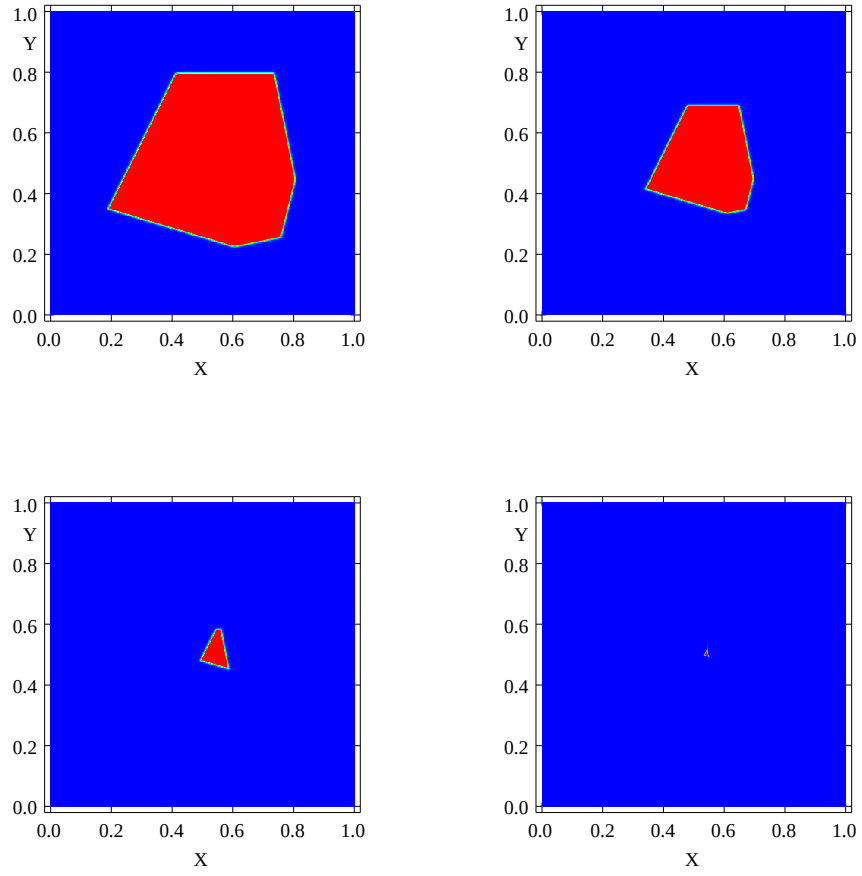


Figure 1: Cell density at $t = 0.5 \times 10^{-4}$, $t = 1.0 \times 10^{-4}$, $t = 1.5 \times 10^{-4}$ and $t = 2.0 \times 10^{-4}$ for an initially irregular polygonal region.

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Numerical simulation of transport equations in Liquid Composite Molding Processes[†]

J.A. García¹, Ll. Gascón¹, F. Chinesta

Keywords: Flux limiter, FEM, Liquid Composite Molding, Porous media

Mathematics Subject Classification 2000: 65M06, 65N06, 35L40

Minisymposia: Multiphase Flow and Porous Media

Abstract This work is concerned with the development and application of a numerical procedure for the simulation of the flow phenomena involved in composite manufacturing processes.

Liquid Composite Molding (LCM) is one of most important manufacturing processes to produce reinforced plastics. It is a process where a liquid resin is injected in a closed mold containing a dry fibrous reinforcement. The resin impregnation is usually modelled as a flow in porous media. The mathematical model of injection molding is a free boundary problem described by Darcy law and incompressible flow. An important task in the simulation of the mold filling process is the numerical treatment of the moving boundary defined by the liquid resin flow front. The numerical method used by the authors to solve this problem combines a discretization of an elliptic equation governing the pressure and velocity distributions and a transport hyperbolic equation governing the evolution of the resin volume fraction in a fixed mesh.

The incubation time is another important variable in the LCM process. It is the elapsed time since the reactives are mixed, which for each fluid particle depends on the path travelled throughout the mold, and is directly related with the resin curing conversion. The evolution of both, the volume fraction and the incubation time in the LCM processes, are given by linear advection equations. In general, the LCM process simulation involves an accurate treatment of the advection equation which governs the evolution of different fluid properties: fluid presence function, incubation time, temperature, degree of saturation, etc.

In [2] the authors describe a 2D numerical model for LCM simulations in which the volume fraction has been updated by using a new second-order scheme with flux limiters, that provide excellent results regardless of the position of the injection points. In [3] we propose a general strategy able to carry out an accurate field transport when an Eulerian technique, using a fixed mesh, is adopted. We redefine the flux limiters proposed in [2] in order to account for the numerical difficulties related to the integration of a variable, such as the incubation time, in fixed mesh simulations when it is not defined in the empty part of the mold. In order to obtain an accurate

[†]Financial support: Spanish Ministry of Science and Innovation (Project DPI2010-20333).

transport of the correspondent field in these cases, we have assumed that the fluid comes to an element only from its full filled neighbor elements. This condition has been introduced in the formulation of the schemes from the use of a parameter which has to be considered in all stages of the discretization. The numerical method minimizes the numerical diffusion and implicitly transports all fields with the flow front movement. We think this numerical technique can also be useful to update other properties governing by advection equations, by example, to simulate the degree of saturation in LCM filling process with void formation.

The reduction of voids formed during the resin filling process is a major issue in LCM processes [1]. Reducing the void content increases the quality of the composite and improves its mechanical properties. Most of modeling efforts on process simulation of mold filling has been focused on the single phase Darcys law, with resin as the only phase, ignoring the formation and transport of voids. Equations that describe the LCM filling process with void formation are based on a two phase flow model and lead to a coupled system of a nonlinear advection-diffusion equation for saturation and an elliptic equation for pressure and velocity. In general, the saturation equation is a non-linear advection-diffusion equation which includes the capillary pressure effect and it reduces to a purely advection transport equation when capillary effects are neglected. The hyperbolic nature of the saturation equation and its strong coupling through relative permeability represents a challenging numerical issue. In this work, a very accurate numerical approach, based on flux limiters, is proposed to solve this complex flow behavior. The implemented algorithm allows a numerical optimization of the injected flow rate which minimizes the micro/macroscopic void formation during mold filling. Some preliminary numerical results are presented here in order to validate the proposed mathematical model and the numerical scheme. This formulation opens up new opportunities to improve LCM flow simulations and optimize injection molds.

References

- [1] J. BREARD, A. SAOUAB, G. BOUQUET. Numerical simulation of void formation in LCM. *Composites Part A: Applied Science and Manufacturing*. **34**, 517-523, 2003.
- [2] J.A. GARCÍA, LL. GASCÓN, F. CHINESTA. A fixed mesh numerical method for modelling the flow in liquid composites molding processes using a volume of fluid technique. *Comput. Methods Appl. Mech. Engrg.* **192**, 877-893, 2003.
- [3] F. SÁNCHEZ, J.A. GARCÍA, LL. GASCÓN, F. CHINESTA. Towards an efficient numerical treatment of the transport problems in the Resin Transfer Molding Simulation. *Methods Appl. Mech. Engrg.* **196**, 2300-2312, 2007.

¹Instituto de Diseño y Fabricación
 Universidad Politécnica de Valencia
 Camino de Vera s/n, 46022 Valencia, Spain
 llgascon@mat.upv.es, jugarcia@mcm.upv.es

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Mathematical modeling of heat transfer during quenching process

D.N. Passarella¹ F. Varas¹ E.B. Martín²

Keywords: Quenching, multiphase flows, finite element method, drift-flux model, heat partition model

Minisymposia: Multiphase flow and porous media

Abstract In quenching by submerging the workpiece is cooled due to vaporization, convective flow and interaction of both mechanisms. The dynamics of these phenomena is very complex and the corresponding heat fluxes are strongly dependent on local flow variables such as velocity of fluid and vapor fraction. This local dependence may produce very different cooling rates along the piece, responsible for inappropriate metallurgical transformations, variability of material properties and residual stresses. In order to obtain an accurate description of cooling during quenching, a mathematical model of heat transfer is presented here. The model is based on the drift-flux mixture-model for multiphase flows [1], including an equation of conservation of energy for the liquid phase and specific boundary conditions that account for evaporation and presence of vapor phase on the surface of the piece [2]-[4]. The model was implemented on *Comsol Multiphysics* software. Generation of appropriate initial and boundary conditions, as well as numerical resolution details is briefly discussed. To test the model, a simple flow condition around a sphere is analyzed. The effect of vapor fraction on heat transfer is assessed. The presence of the typical vapor blanket and its collapse are recovered by the model, and its effect on the cooling rates on different parts of the piece is analyzed.

References

- [1] Ishii M and Hibiki T 2006 *Thermo-fluid dynamics of two-phase flow* (New York: Springer)
- [2] Krepper E, Koncar B and Egorov Y 2007 CFD modelling of subcooled boiling – Concept, validation and application to the fuel assembly design *Nuclear Engineering and Design* **237** 716-31
- [3] Kocar C and Sokmen C M 2009 A CFD model of two-phase flow in BWRs with results of sensitivity and uncertainty analysis *Nuclear Engineering and Design* **239** 1839-46
- [4] Koncar B and Tiselj I 2010 Influence of near-wall modelling on boiling flow simulation *Nuclear Engineering and Design* **240** 275-83

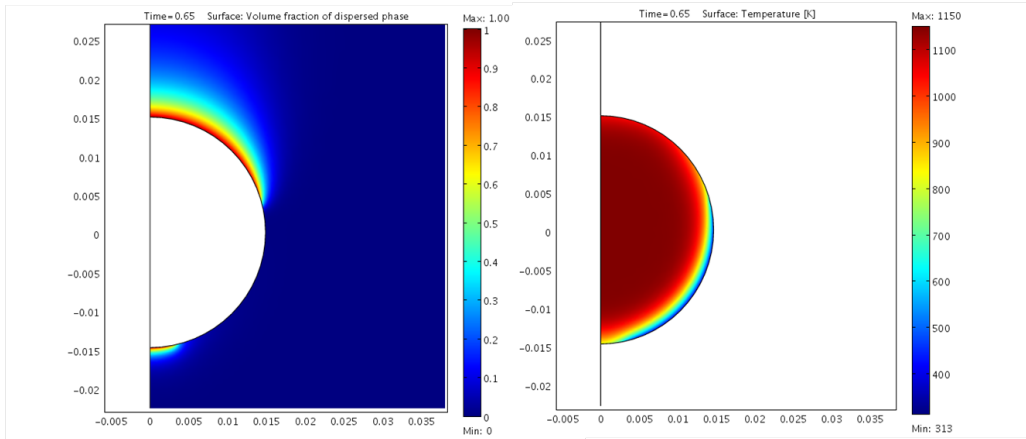


Figure 1: Vapor blanket (l) and piece temperature (r) at $t=0.65$ seg.

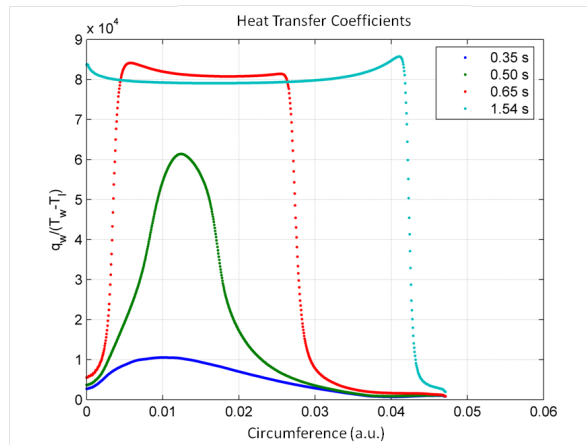


Figure 2: Variation of heat transfer coefficient along the circumference

¹Departamento de Matemática Aplicada II
E.T.S. de Ing. de Telecomunicación
Universidad de Vigo, Campus Marcosende, 36310 Vigo, España
diego@dma.uvigo.es
fvaras@uvigo.es

²Área de Mecánica de Fluidos
E.T.S. de Ing. Industriales
Universidad de Vigo, Campus Marcosende, 36310 Vigo, España
emortega@uvigo.es

Conservative formulation for compressible fluid flow through elastic porous media[†]

E. Romenski¹

Keywords: Hyperbolic systems of conservation laws, compressible fluid flow, elastic porous media

Minisymposia: Multiphase flow and porous media

Abstract The development of advanced computational modelling for fluid flow in elastic porous media is of interest in a number of scientific and engineering disciplines including geophysical applications. In recent decades a theories for multiphase flows and associated numerical algorithms have been developed and exploited for the needs of industry and environmental sciences. But up to now there is no conventional form of the model and its governing equations even for compressible two-phase flows. Several approaches exist also for the modelling of fluid flow in elastic media. The Biot equations [1] for small amplitude wave propagation in the saturated porous medium are widely used in geophysics but they are not applicable to the case of finite deformations. The phenomenological model based on the irreversible thermodynamics principles has been developed in [2] taking into account finite deformations, but its governing equations are not fully conservative and in addition have some disadvantages for the case of vanishing phase.

The main challenge in the multiphase flow modelling is associated with the formulation of a mathematical model that satisfies such important properties as hyperbolicity, fully conservative form of the governing equations and compatibility and consistency of the mathematical model with the thermodynamic laws. These properties provide a solid mathematical framework for a theory of different initial-boundary value problems and allows to develop highly accurate numerical methods.

Recently a hierarchy of conservative models for compressible two-phase flow and high-accuracy numerical methods for solving of corresponding governing equations have been proposed in [5, 4]. The development of these models are based on the theory of thermodynamically compatible hyperbolic systems of conservation laws [3].

Here we present an application of this theory for conservative formulation of isentropic flow of compressible fluid in the elastic porous media. Its core aspect is a phenomenological modelling of continuous media, where by using thermodynamic laws we determine a structure of the governing balance laws. In this context the mixture is supposed to be a continuum in which the multiphase character of flow is taken into account. The resulting system of PDEs is hyperbolic and all balance equations can be cast in conservation form.

[†]This work was supported by the RFBR grants 09-01-00221, 10-05-00233, 11-01-00147 and by the Presidium of Russian Academy of Sciences Programme of Fundamental Research No 2 (project 121)

As an alternative to the Biot theory, the governing equations for small amplitude wave propagation are derived from the general system of conservation laws. The dependence of the fast and slow pressure waves and shear wave on the porosity is studied. Some test problems are solved numerically.

References

- [1] M. A. BIOT. Theory of propagation of elastic waves in a fluid-saturated porous solid. I. Lowfrequency range. *J. Acoust. Soc. Am.* **28**(2), 168–178, 1956.
- [2] A. M. BLOKHIN AND V. N. DOROVSKY. Mathematical modeling in the theory of multivelocity continuum. Nova Science. New York, 1995.
- [3] S. K. GODUNOV AND E. I. ROMENSKII. Elements of continuum mechanics and conservation laws. Kluwer Academic/Plenum Publishers. New York, 2003.
- [4] E. ROMENSKI, A. D. RESNYANSKY AND E. F. TORO. Conservative hyperbolic formulation for compressible two-phase flow with different phase pressures and temperatures. *Q. Appl. Math.* **65**(2), 259-279, 2007.
- [5] E. ROMENSKI, D. DRIKAKIS AND E. F. TORO. Conservative models and numerical methods for compressible two-phase flow *J. Sci. Comput.* **42**(1), 68-95, 2010.

¹Sobolev Institute of Mathematics
 Novosibirsk 630090, Russia
 evrom@math.nsc.ru

On the coupling of compressible and incompressible fluids.

V. Schleper¹

Keywords: Multiphase flow, coupling conditions

Minisymposia: Multiphase flow and porous media

Abstract In some applications in multiphase flow, compressible effects cannot be neglected. In this case, one assumes in general, that the liquid phase as well as the gaseous phase are modelled by compressible equations such as the Euler equations

$$\begin{aligned}\partial_t \rho + \nabla \cdot (\rho \mathbf{v}) &= 0 \\ \partial_t (\rho \mathbf{v}) + \nabla \cdot (\mathbf{v} \otimes (\rho \mathbf{v})) + \nabla p_i &= 0 \\ \partial_t E + \nabla \cdot (\mathbf{v} (E + p_i)) &= 0,\end{aligned}\tag{1}$$

where as usual ρ denotes the density, $\mathbf{v}$ the velocity, $E = \rho e + \frac{1}{2} \rho |\mathbf{v}|^2$ the total energy and e the internal energy.

To model the different behaviours of the phases, each phase is described by a different equation of state $p_i(\rho, e)$, which accounts especially for the different compressibilities. In the literature (see e.g. [1, 2]), it is for instance common to use the Tait equation of state

$$p_{Tait}(\rho) = k_0 \left(\left(\frac{\rho}{\rho_0} \right)^\gamma - 1 \right) + p_0\tag{2}$$

to model the behaviour of very weakly compressible fluids such as liquids, where the constants are chosen to $\gamma \approx 7$ and $k_0 \approx 3000$. This leads to a very stiff pressure law and to a very large speed of sound in the liquid regions. Numerically, this results in severe time step restrictions due to the CFL-condition.

To circumvent this high restriction on the time step size, we propose a coupling of compressible and incompressible flows, where the flow in the liquid phase is given by the incompressible Euler equations

$$\begin{aligned}\nabla \cdot (\mathbf{v}) &= 0 \\ \partial_t \mathbf{v} + \nabla \cdot (\mathbf{v} \otimes \mathbf{v}) + \nabla p_i &= 0.\end{aligned}$$

We show that the interface conditions for the coupling of Euler and incompressible Euler equations in one space dimension are given by an ordinary differential equation and discuss how the interface conditions can be solved numerically.

Unlike the purely compressible case, where a one-dimensional Riemann solver is sufficient to solve numerically also problems in higher space dimension, this holds not true for the coupling of compressible and incompressible flows. We show why this is the case and discuss the necessary extensions of the interface conditions for higher space dimensions.

References

- [1] A. FERRARI, C.-D. MUNZ AND B. WEIGAND. A High Order Sharp-Interface Method with Local Time Stepping for Compressible Multiphase Flows. *Commun. Comput. Phys.* **9**, 205–230, 2010.
- [2] T. G. LIU, B. C. KHOO AND C. W. WANG. The ghost fluid method for compressible gas-water simulation. *J. Comput. Phys.* 204(1), 193–221, 2005.

¹University of Stuttgart
Institute of applied analysis and numerical simulation
Pfaffenwaldring 57
D-70569 Stuttgart
Germany
`veronika.schleper@mathematik.uni-stuttgart.de`
`http://www.ians.uni-stuttgart.de/am/Schleper/`

Numerical Methods for Hyperbolic Equations: Theory and Applications

An international conference to honour Professor E.F. Toro
in the month of his 65th birthday

Minisymposia

Numerical methods in
astrophysics

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Numerical methods for the hyperbolic evolution equations in the Fully Constrained Formalism of Einstein equations.

I. Cordero-Carrión¹ P. Cerdá-Durán¹ J.M. Ibáñez²

Keywords: numerical methods, hyperbolic equations, General Relativity

Minisymposia: Numerical methods in astrophysics, nuclear fusion and plasma physics

Abstract In this talk we will review the structure of the resulting coupled hyperbolic-elliptic set of equations in the Fully Constrained Formalism (FCF) [1] of Einstein equations. Transverse-traceless decomposition of metric tensors [2] make possible a reformulation of the equations, providing local uniqueness properties to the elliptic ones. The hydrodynamics equations have to be solved, coupled with Einstein equations, in presence of matter.

The gravitational radiation of the system is encoded in the hyperbolic equations of the formulation. If these equations are neglected, i.e., the gravitational radiation of the system is not taken into account, the so-called Conformally Flat Condition (CFC) [3, 4] of Einstein equations is recovered. This scheme is an approximation of Einstein equations used in many astrophysical applications in the last years [5]. It can be, therefore, desirable to extend this approximation to a fully relativistic formulation where very compact objects are involved in the astrophysical simulations (e.g., neutron stars or black holes).

The main objective of the talk is to present the numerical methods used in order to evolve numerically the hyperbolic system of Einstein equations, in particular implicit-explicit Runge-Kutta high order methods and finite differences grid. Stability and convergence of this system are studied. In current simulations, this system is influenced by the rest of variables, although the sources containing the ones of the hyperbolic system of Einstein equations are neglected in the rest of elliptic and hydrodynamic equations. We are then able to read the gravitational radiation of a system in which this gravitational radiation itself does not affect the evolution of the rest of equations.

The next step, after checking the good numerical behavior of the system, is the full implementation of the FCF in order to extract gravitational waves from astrophysical scenarios.

Mathematics Subject Classification 2000: 35L40, 35Q76, 65N06

References

- [1] S. BONAZZOLA, E. GOURGOULHON, P. GRANDCLÉMENT AND J. NOVAK. Constrained scheme for the Einstein equations based on the Dirac gauge and spherical coordinates. *Phys. Rev. D* **70**, 104007, 2004.
- [2] I. CORDERO-CARRIÓN, P. CERDÁ-DURÁN, H. DIMMELMEIER, J. L. JARAMILLO, J. NOVAK AND E. GOURGOULHON. Improved constrained scheme for the Einstein equations: An approach to the uniqueness issue. *Phys. Rev. D* **77**, 084007, 2008.
- [3] J. A. ISENBERG. Waveless Approximation Theories of Gravity. *Int. J. Mod. Phys. D* **17**, 265, 2008.
- [4] J. R. WILSON AND G. J. MATHEWS in *Frontiers in numerical relativity*, edited by C. R. Evans, L. S. Finn and D. W. Hobill (Cambridge University Press, Cambridge, 1989).
- [5] H. DIMMELMEIER, J. A. FONT AND E. MÜLLER. Relativistic simulations of rotational core collapse II. Collapse dynamics and gravitational radiation. *Astron. Astrophys.* **393**, 523, 2002.

¹Max-Planck-Institute for Astrophysics, Garching
Karl-Schwarzschild-Str. 1, D-85741, Garching, Germany

²Departamento de Astronomía y Astrofísica, Universidad de Valencia
C/ Dr. Moliner, 50, E-46100, Burjassot (Valencia), Spain

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

SPH versus AMR

S. A. E. G. Falle¹

Keywords: Adaptive Mesh Refinement, Smoothed Particle Hydrodynamics

Minisymposia: Numerical methods in astrophysics

Abstract The most widely used computational methods for astrophysical fluid dynamics are SPH (Smoothed Particle Hydrodynamics) and grid codes using AMR (Adaptive Mesh Refinement). The relative merits of these methods have been hotly disputed for a number of years, but in the end the only way to settle this is to compare them for a range of problems for which one has a good idea of the correct solution. I will discuss comparisons between the SPH code, SEREN, and the Godunov-type AMR code, MG for a number of test problems, such as Kelvin-Helmholtz, radiative shocks, moving polytropes and a simple model of the Santa Barbara cluster formation problem.

Although the Riemann problem is more than 100 years old, it still offers a fertile ground for research and surprises. I will review our present understanding of this problem both in relativistic hydrodynamics and in relativistic magnetohydrodynamics, highlighting some new developments such as the ideas of a hydrodynamical-boost in astrophysical jets or the "shock-detector" in numerical hydrodynamics.

¹Department of Applied Mathematics
University of Leeds, Leeds LS2 9JT, UK
`sam@maths.leeds.ac.uk`

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

An Approximate Harten-Lax-van Leer Riemann Solver for Relativistic Magnetohydrodynamics

A. Mignone¹

Keywords: hydrodynamics - MHD - Riemann problem - Conservation Law - relativity - shock waves - methods: numerical

Minisymposia: Numerical methods in astrophysics

Abstract An approximate Riemann solver for the equations of ideal relativistic magnetohydrodynamics is presented. The solver belongs to the so-called Harten-Lax-van Leer (HLL, [1]) family of solvers where an initial guess to the characteristic wave speeds is given without any knowledge a priori of the solution. Our proposed method of solution generalizes to the relativistic case the classical five-wave HLLD Riemann solver initially developed by Miyoshi & Kusano [6] for the equations of classical ideal magnetohydrodynamics. The solution to the Riemann problem is approximated by a five-wave pattern, comprising two outermost fast shocks, two rotational discontinuities and a contact surface in the middle, [3]. The proposed scheme is considerably more elaborate than in the classical case since the normal velocity is no longer constant across the rotational modes. Still, proper closure to the Rankine-Hugoniot jump conditions can be obtained by solving a single nonlinear scalar equation in the total pressure variable which, for the chosen configuration, has to be constant over the whole Riemann fan. The accuracy of the new Riemann solver is validated against one-dimensional tests and multidimensional applications. It is shown that our new solver considerably improves over the popular Harten-Lax-van Leer solver and the similar, less accurate, HLLC schemes [5, 4, 2].

References

- [1] A. HARTEN, P.D. LAX, B. VAN LEER. On upstream differencing and Godunov-type schemes for hyperbolic conservation laws. *SIAM Rev.* **25**(1), 35-61, 1983
- [2] V. HONKKILA, P. JANHUNEN. HLLC solver for ideal relativistic MHD. *J. Comput. Phys.* **22**(2), 643-656, 2007
- [3] A. MIGNONE, M. UGLIANO, G. BODO. A five-wave Harten-Lax-van Leer Riemann solver for relativistic magnetohydrodynamics. *Mon. Not. R. Astron. Soc.* **393**, 1141-1156, 2009.
- [4] A. MIGNONE, G. BODO. An HLLC Riemann solver for relativistic flows - II. Magnetohydrodynamics. *Mon. Not. R. Astron. Soc.* **368**, 1040-1054, 2006.

- [5] A. MIGNONE, G. BODO. An HLLC Riemann solver for relativistic flows - I. Hydrodynamics. *Mon. Not. R. Astron. Soc.* **364**, 126-136, 2005.
- [6] T. MIYOSHI, K. KUSANO. A multi-state HLL approximate Riemann solver for ideal magnetohydrodynamics. *J. Comput. Phys.* **208**(1), 315-344, 2005.

¹Dipartimento di Fisica Generale
Università di Torino
Via Pietro Giuria 1. 10125 Torino, Italy
`mignone@ph.unito.it`

MUSTA schemes in magnetohydrodynamics and neutrino transfer: application to core-collapse supernovae and gamma-ray bursts

M. Obergaulinger¹

Keywords: Hydrodynamic and hydromagnetic problems, Radiative transfer

Minisymposia: Numerical methods in astrophysics

Abstract The influence of magnetic fields on the evolution of core-collapse supernovae and gamma-ray bursts is not fully understood. While the presence of a magnetic field in these events is very likely, it is still uncertain whether or not it can be amplified from its weak initial field strength to values roughly in energetic equipartition with the fluid flow, i.e., to a strength sufficient to affect the flow on the dynamical time scale of the explosion. Observationally, the detection of magnetars, very highly magnetised neutron stars, suggests that very strong magnetic fields may be generated in supernovae under certain conditions.

Some of the mechanisms proposed to account for the amplification involve magnetohydrodynamic turbulence. Among them are convection driven by gradients of entropy and electron number or the magneto-rotational instability (MRI) of a differentially rotating supernova core, or the Kelvin-Helmholtz instability of the shear layer forming in the merger of two neutron stars triggering a short gamma-ray burst.

In order to follow the MHD turbulence developing due to these instabilities, the numerical modelling of these processes requires highly accurate methods. A further complexity arises from the large range of additional physical effects that may have an influence on the explosion, chief among them the transport of neutrinos and their interaction with the stellar matter. Apart from physical uncertainties in the cross sections of neutrino-matter interactions, neutrino transport is numerically a very challenging problem because of the high dimensionality of the distribution function of the neutrinos. It is possible to reduce the dimensionality and, hence, the computational requirements by solving the equations governing the evolution of the first few moments of the neutrino distribution function. The equations for the zeroth and first moments can be written as a hyperbolic system, and therefore can be treated numerically with standard methods for hyperbolic systems.

I will describe a code employing high-order reconstruction methods and multi-stage (MUSTA) approximate Riemann solvers to solve the MHD equations coupled to the system of neutrino moments as well as simulations addressing the issue of field amplification in core-collapse supernovae and gamma-ray bursts, viz. local simulations of MRI-unstable flows in supernova cores and Kelvin-Helmholtz-unstable shear layers in mergers, and global models of collapsing stellar cores. Though these simulations do not yet put us in a position to answer the question for the origin of strong fields

and their dynamic feedback, I will point out some of their most important results and conclusions for future investigations.

Keywords: Hydrodynamic and hydromagnetic problems, Radiative transfer

Mathematics Subject Classification 2000: 85A30, 85A25

¹Racah Institute of Physics
The Hebrew University
Jerusalem 91904, Israel
maob@racah.phys.huji.ac.il

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

MHD-kinetic Modeling of Partially Ionized Plasma Flows: Solar Wind Interaction with the Interstellar Medium

N. V. Pogorelov¹ J. Heerikhuisen¹ S. N. Borovikov¹ I. A. Kryukov¹
G. P. Zank¹

Keywords: hyperbolic equations, discontinuous solutions, AMR, finite-volume methods, Monte Carlo modeling, time and space scale coupling

Minisymposia: Numerical methods in astrophysics

Abstract A common element across multiple, often independent, disciplines is that almost all complex systems occurring in geophysical, space, astrophysical, engineering, and even biological environments comprise multiple gas, fluid, or plasma components that are mutually coupled across many spatial and temporal scales via collision-like integral terms. The “collisional terms” may incorporate processes as diverse as chemical kinetics, charge-exchange, coagulation, etc. The range of scales found in complex systems often requires that some constituents must be modeled using a Boltzmann-like transport equation rather than a continuum system of equations. Continuum, or fluid-dynamics-like, systems of equations have been explored computationally for a variety of applications in considerable depth over the past 15 years but the development of models that incorporate a multiplicity of temporal and spatial scales and collision-like integral terms have received very little attention despite their considerable importance.

Flows of partially ionized plasma are frequently characterized by the presence of both thermal and nonthermal populations of ions. Such flows cannot be modeled using traditional MHD equations. When non-thermal ions are formed due to charge exchange and/or collisions between the thermal (core) ions and neutrals, they experience the action of magnetic field, their distribution function is isotropized by wave-particle interaction, and they soon acquire the velocity of the ambient plasma flow without being thermodynamically equilibrated with the core plasma. This occurs, e. g., in the outer heliosphere – the part of interstellar space beyond the solar system whose properties are determined by the solar wind (SW) interaction with the local interstellar medium (LISM). Collisions between atoms and ions in heliospheric plasma are so rare that they should be modeled kinetically. On the other hand, PUIs born when LISM neutral atoms charge-exchange with SW ions represent a hot, non-equilibrium component and also require a proper kinetic treatment.

Using Chombo as an adaptive mesh refinement (AMR) framework, we have implemented a suite of numerical codes (Multi-Scale Fluid-Kinetic Simulation Suite, MS-FLUKSS) that allows to perform high resolution simulations of MHD, multi-fluid, and MHD-kinetic flows. The AMR implemented in both Cartesian and curvilinear

coordinate systems allows us to perform a detailed analysis of discontinuous flows in the presence of shocks, tangential discontinuities, and current sheets [1]. Our physical model assumes that charged and neutral particles are governed by different sets of equations (MHD and kinetic Boltzmann) self-consistently connected by a vector source term responsible for charge-exchange collisions between these particles. The MHD subsystem consists of the conservation laws for the ion-electron mixture and is solved with the Godunov-type schemes accompanied by an efficient implementation of the far-field boundary conditions [2]. The Boltzmann equation is solved by a Monte Carlo method using a variable-weight particle approach, which increases the efficiency of the method substantially [3, 4]. A level-set method is implemented to describe the evolution of flow separation surfaces and current sheets.

We describe the peculiarities of coupling the MHD and Boltzmann equations and present numerical results of the SW–LISM interaction modeling obtained with MS-FLUKSS. The emphasis is on high-resolution time-dependent simulations requiring AMR simulations on leadership supercomputers. The influence of the interstellar magnetic field and directions covering the range from sub- to super-fast MHD flows is analyzed by comparing numerical simulations with observations. It is shown that the presence of ion-neutral interactions can dramatically change the character of the stream interaction. The results of the shock-turbulence interaction simulations are presented obtained with numerical schemes of the second to sixth order of accuracy.

References

- [1] POGORELOV, N. V., BOROVNIKOV, S. N., FLORINSKI, V., HEERIKHUISEN, J., KRYUKOV, I. A., ZANK, G. P. (2009c), Multi-scale fluid-kinetic simulation suite: A tool for efficient modeling of space plasma flows, in *Astronomical Society of the Pacific Conf. Ser.* 406, *Numerical Modeling of Space Plasma Flows: ASTRONUM-2008*, ed. N. V. Pogorelov et al., 149–159. ASP, San Francisco, 2009.
- [2] A.G. KULIKOVSKII, N. V. POGORELOV, A. Y. SEMENOV. *Mathematical Aspects of Numerical Solution of Hyperbolic Systems*. Chapman and Hall/CRC Press, Boca Raton, Florida, London, U.K. Book (540 pages). ISBN 0-8493-0608-6, 2001.
- [3] POGORELOV, N.V., HEERIKHUISEN, J., MITCHELL, J.A., CAIRNS, I.H., ZANK, G.P. Heliospheric asymmetries and 2-3 kHz radio emission under strong interstellar magnetic field conditions, *Astrophys. J.*, 695, L31, 2009
- [4] HEERIKHUISEN, J., POGORELOV, N. V., ZANK, G. P., CREW, G. B., FRISCH, P. C., FUNSTEN, H. O., JANZEN, P. H., MCCOMAS, D. J., REISENFELD, D. B., SCHWADRON, N. A. Pick-up ions in the outer heliosheath: A possible mechanism for the Interstellar Boundary Explorer ribbon, *Astrophys. J.*, 708, L126, 2010

¹Physics Department and Center for Space Plasma and Aeronomic Research
University of Alabama in Huntsville
320 Sparkman Dr., Huntsville, AL 35805, U.S.A.
np0002@uah.edu

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

ON THE RIEMANN PROBLEM IN RELATIVISTIC HD AND MHD

L. Rezzolla¹

Keywords: Hydrodynamic and hydromagnetic problems, Riemann Problem

Minisymposia: Numerical methods in astrophysics

Abstract Although the Riemann problem is more than 100 years old, it still offers a fertile ground for research and surprises. I will review our present understanding of this problem both in relativistic hydrodynamics and in relativistic magnetohydrodynamics, highlighting some new developments such as the ideas of a hydrodynamical-boost in astrophysical jets or the "shock-detector" in numerical hydrodynamics.

¹Albert Einstein Institute
14476 Potsdam-Golm, Germany
rezzolla@aei.mpg.de

Numerical Methods for Hyperbolic Equations: Theory and Applications

An international conference to honour Professor E.F. Toro
in the month of his 65th birthday

Minisymposia

Recent advances in the numerical
computation of environmental
conservation laws with source
terms

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Asymptotic behaviour of godunov-type numerical models with mobile bed with adaptation term.

L. Begnudelli¹ G. Rosatti¹ D. Zugliani¹

Keywords: Numerical Methods, Sediment Transport, Source Terms and E. F. Toro

Minisymposia: Recent advances in the numerical computation of environmental conservation laws with source terms

Abstract In this paper we present a study on the behaviour of sediment transport models with an adaptation source term representing the time or space required by the concentration to adapt to the flow conditions. In particular, the paper investigates the asymptotic behaviour, for values of the adaptation length approaching zero or infinite, of two finite-volume Godunov-type numerical models with mobile bed. Considering the asymptotic behaviour of the mathematical model, for vanishing values of the adaptation length, the model tends to the case of immediate adaptation. Vice-versa, when the adaptation length tends to infinite, we end up with a fixed bed model. On the other hand, two Godunov-type numerical models have been considered, one based on the LHLL Riemann solver (Fraccarollo et al., 2003), and the other based on a generalized Roe solver (Rosatti et al., 2008). The obtained results show that when the adaptation length tends to infinite the models converge correctly to the fixed bed case. However, when the adaptation length tends to zero, both models fail to converge to the immediate adaptation scheme for Froude numbers greater than roughly 0.8. We show that the reason of this behaviour is intrinsic to the hyperbolic structure of the models, which is dictated by the homogeneous part of the governing equations system. Thus, adaptive models present the same eigenstructure of fixed bed models, while models with immediate adaptation have the typical eigenstructure of mobile bed models. Thus, models with adaptation terms cannot converge to mobile bed models (case of vanishing adaptation length) because, when Froude is larger than 1, they cannot develop the wave propagating in the negative direction that is always present using a system of equation corresponding to a mobile bed model. Clearly, the problem arises when the flow becomes supercritical, because negative eigenvalues are not possible, and can be also seen when the Froude approaches unity. This study shows that adaptive models present the same eigenstructure of the fixed bed ones. Therefore, in general, they cannot converge for values of the adaptation length approaching zero.

References

- [1] L. FRACCAROLLO, H. CAPART, Y. ZECH. A Godunov method for the computation of erosional shallow water transient. *International Journal for Numerical Methods in Fluids* **41**(9), 951–976, 2003.

- [2] G. ROSATTI, J. MURILLO , L. FRACCAROLLO. eneralized Roe schemes for 1D, two-phases, free-surface flows over mobile-bed. *IJournal of Computational Physics* **227**(24), 10058–10077, 2008.

¹CUDAM (University Center for the Hydrogeologic Defence of the Mountain Environment)
University of Trento
Via Mesiano 77, I-38123 Trento, Italy
`lorenzo.begnudelli@unitn.it`
`daniel.zugliani@gmail.com`
`giorgio.rosatti@unitn.it`

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Finite volume discretisation of depth averaged scalar transport equations coupled to shallow water models.

L. Cea,¹ M.E. Vazquez-Cendon²

Keywords: Finite volumes, scalar transport, shallow water equations

Minisymposia: Recent advances in the numerical computation of environmental conservation laws with source terms

Abstract Shallow water models provide a useful and efficient tool in order to study environmental free surface flows. In this field, two-dimensional depth averaged models have become very popular not only to compute the hydrodynamics, but also to compute the dispersion of contaminants, sediment transport, or any other physical process which depends on the water depth and velocity fields.

The numerical discretisation of the convective flux in the shallow water equations has been extensively studied in many previous works [1, 3]. These works have shown that it is important for the numerical discretisation of the bathymetry to be related to the discretisation of the convective flux in order to have a well-balanced scheme. The same applies for the discretisation of any scalar transport equation linked to the hydrodynamic equations [2]. The scalar transport is strongly determined by the velocity and water depth fields. Therefore, in order to define a well-balanced conservative scheme, the numerical discretisation of the hydrodynamic and the scalar transport equations cannot be done independently of each other. This means that it is not only important to consider the flow field, but it is also necessary to take into account the numerical scheme used to discretise the hydrodynamic equations, in particular the mass conservation equation, otherwise the resulting scheme for the scalar transport equation might not be conservative. This is specially true in the presence of unsteady wet-dry fronts, because in those cases the numerical implementation of the wetting and drying algorithm defines how the water front moves and therefore, how the mass of the scalar is transported through the domain.

In this paper we analyse the previous issues using a finite volume solver for the hydrodynamic equations. First, we analyse the implications that the discretisation used for the bathymetry and convective flux have in the numerical scheme used to solve a scalar transport equation linked to the shallow water equations. An alternative discretisation for computing the convective transport of a scalar is proposed, which guarantees the mass conservation of the scalar in the presence of unsteady wet-dry fronts over irregular bathymetries. Several test cases are used to show the accuracy and stability of the different discretisation alternatives.

References

- [1] R. J. LEVEQUE, *Finite Volume Methods for Hyperbolic Problems*. Cambridge Texts in Applied Mathematics, **31**, Cambridge, 2002.
- [2] J. MURILLO AND J. BURGUETE AND P. GARCIA-NAVARRO, Analysis of a second-order upwind method for the simulation of solute transport in 2D shallow water flow, *International Journal of Numerical Methods in Fluids*, **56**, 661–686, 2008.
- [3] E. F. TORO, *Shock-capturing Methods for Free-Surface Shallow Flows*, Wiley, Chichester, West Sussex PO19 1UD, England, 2001.

¹GEAMA, Water and Environmental Engineering Group.
Hydraulics Division.
University of A Coruña
E.T.S.I. Caminos, Canales y Puertos. 15071 Campus de Elviña, s/n.
A Coruña, Spain
`luis.cea@udc.es`

²Applied Mathematics Department
University of Santiago de Compostela
Faculty of Mathematics. 15782 Santiago de Compostela, Spain
`elena.vazquez.cendon@usc.es`

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

A finite volume/duality method for Bingham viscoplastic flow

J.M. Gallardo¹ E.D. Fernández-Nieto² P. Vigneaux³

Keywords: Shallow water systems, Bingham flows, viscoplastic flows, finite volumes, duality methods, well-balanced schemes

Minisymposia: Recent advances in the numerical computation of environmental conservation laws with source terms

Abstract In recent years there has been increasing interest in the understanding and numerical simulation of Bingham fluid flows: see [5] for a review. Bingham fluids behave rigidly below a certain stress yield and flow as an incompressible viscous fluid above that yield. Recently, the authors in [2] have deduced a viscous shallow water formulation for a Bingham fluid in the particular case of a plane slope bottom. The model has been applied to the simulation of avalanches in mountainous regions.

Our work focuses on the numerical analysis of the shallow water model in [2], which consists of a mass conservation law coupled with a variational inequality related to the conservation of momentum, which takes into account the viscoplastic behaviour of the fluid.

The conservation law is solved by means of a well-balanced finite volume method, built along the guidelines in [3]. In particular, the method is able to preserve stationary solutions representing material at rest, and also stationary solutions of constant height over an inclined plane.

The solution of the variational inequality is approximated by using a duality method. In particular, we have adapted to this case the Bermúdez-Moreno duality algorithm [1], which is based on the properties of the Yosida regularization of maximal monotone operators. An important issue in the numerical model is the coupling of the finite volume solver with the duality method, that must be performed maintaining the well-balancing properties.

The convergence of the Bermúdez-Moreno algorithm depends strongly on the choice of two constant parameters. To overcome this difficulty and improve the rate of convergence, we have also adapted the algorithms studied in [4] and the references therein, for which an automatic selection of the parameters is carried out.

Several experiments are shown to test the performances of the proposed coupled method, both in the one- and two-dimensional cases. Moreover, comparisons with the Augmented Lagrangian approach in [2] are also presented.

References

- [1] A. BERMÚDEZ AND C. MORENO. Duality methods for solving variational inequalities. *Computers & Mathematics with Applications*, **7**(1), 43–58, 1981.

- [2] D. BRESH, E.D. FERNÁNDEZ-NIETO, I.R. IONESCU AND P. VIGNEAUX. Augmented lagrangian method and compressible visco-plastic flows: Applications to shallow dense avalanches. *Advances in Mathematical Fluid Dynamics*, 57–89, 2010.
- [3] M. CASTRO-DÍAZ, T. CHACÓN-REBOLLO, E.D. FERNÁNDEZ-NIETO AND C. PARÉS. On well-balanced finite volume methods for nonconservative nonhomogeneous hyperbolic systems. *SIAM Journal on Scientific Computing*, **29**(3), 1093–1126, 2007.
- [4] J.M. GALLARDO, C. PARÉS AND M. CASTRO. A generalized duality method for solving variational inequalities: Applications to some nonlinear Dirichlet problems. *Numerische Mathematik*, **100**, 259–291, 2005.
- [5] E. DEAN, R. GLOWINSKI AND G. GUIDOBONI. On the numerical simulation of Bingham visco-plastic flow: Old and new results. *Journal of Non Newtonian Fluid Mechanics*, **142**, 36–62, 2007.

¹Departamento de Análisis Matemático
 Universidad de Málaga
 Facultad de Ciencias. 29080 Málaga, Spain
gallardo@anamat.cie.uma.es
<http://edanya.uma.es>

²Departamento de Matemática Aplicada
 Universidad de Sevilla
 E.T.S. Arquitectura. 41012 Sevilla, Spain
edofer@us.es

³Unité de Mathématiques Pures et Appliquées
 ENS Lyon. 69364 Lyon, France
Paul.Vigneaux@math.cnrs.fr

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

A numerical model for global climate: effect of the latent heat of fusion.

A. Hidalgo¹

Keywords: Climate model, Finite volume method, WENO reconstruction

Minisymposia: Recent advances in the numerical computation of environmental conservation laws with source terms

Abstract In this work we compute the numerical solution of a global climate model in which the effect of latent heat of fusion for ice is taken into account. The model is based on that proposed by Watts y Morantine in [4] but including a coalbedo temperature dependent and also the previously mentioned latent heat of fusion.

One of the main features of the model, which makes the problem particularly interesting, is the dynamic and diffusive boundary condition that represents the coupling between ocean and atmosphere. The numerical method used is a finite volume scheme with high order WENO reconstruction in space and third order Runge-Kutta TVD for time discretization. The nonlinearity due to the latent heat is solved by means of an iterative scheme based on a combination of Newton-Raphson and bisection methods.

Results of the continuous model can be found in [1] y [2]. Numerical results can be found in the reference [3].

This work has been carried out in collaboration with J.I. Díaz and L. Tello.

Mathematics Subject Classification 2000: 65N08, 35Q80

References

- [1] J.L. DÍAZ Y L. TELLO. A 2D climate energy balance model coupled with a 3D deep ocean model. *Electronic Journal of Differential Equations*, Conf. 16, pp. 129-135, 2007.
- [2] J.L. DÍAZ Y L. TELLO. On a climate model with a dynamic nonlinear diffusivity boundary condition. *Discrete and Continuous Dynamical systems*, Series S **1**(2), pp. 253-262, 2008.
- [3] A. HIDALGO Y L. TELLO. A finite volume scheme for simulating the coupling between deep ocean and an atmospheric energy balance model *Modern mathematical tools and techniques in capturing complexity. Springer Series in Synergetics*, pp. 210-224, 2011.

- [4] R.G. WATTS Y M. MORANTINE. Rapid climatic change and the deep ocean.
Climatic change **16**, pp. 83-97, 1990.

¹Departamento de Matemática Aplicada y Métodos Informáticos
Universidad Politécnica de Madrid
ETSI Minas. 28003 Madrid, Spain
`arturo.hidalgo@upm.es`

Residual Distribution for Shallow Water Flows

M.E. Hubbard¹

Keywords: residual distribution, shallow water model, source terms, well balanced schemes

Minisymposia: Recent advances in the numerical computation of environmental conservation laws with source terms

Abstract The residual distribution framework was developed as an alternative to the finite volume approach for approximating hyperbolic systems of conservation laws which would allow a natural representation of genuinely multidimensional flow features. The resulting algorithms are closely related to conforming finite elements, but their structure makes it far simpler to construct nonlinear approximation schemes, and therefore to avoid unphysical oscillations in the numerical solution. They have been successfully applied to a wide range of nonlinear systems of equations, producing accurate simulations of both steady and, more recently, time-dependent flows (see [1, 3] for fairly recent reviews of the development of these algorithms).

When designed carefully, residual distribution schemes have the following very useful properties.

- They can be simultaneously second order accurate (in space and time) and free of unphysical oscillations, even in the presence of turning points in the solution. Even higher order accuracy can also be achieved, but a form of limiting has to be applied, which reverts to the second order scheme, where unwanted oscillations would otherwise appear. This differs from finite volume schemes, which almost invariably drop to first order accuracy when limiters are applied.
- The CRD (Conservative Residual Distribution) formulation [2] provides a very natural way to approximate balance terms in a manner which automatically retains equilibria inherent in the underlying system.
- It is possible to construct residual distribution schemes which allow for a discontinuous representation of the dependent variables [4]. In particular, the inclusion of discontinuities in time allows for the development of schemes which are unconditionally positive [5], *i.e.* they are free of unphysical oscillations whatever size of time-step is taken.

This presentation will discuss the application of residual distribution to steady and time-dependent shallow water flows. It will include an overview of recent work, such as that of Ricchiuto and Bollermann [6], and progress on the development of a second order accurate, unconditionally positive, well-balanced scheme for the two-dimensional shallow water equations with variable bed topography.

References

- [1] R.ABGRALL, Residual distribution schemes: Current status and future trends, *Computers and Fluids*, **35**:641–669, 2006.
- [2] A.CSIK, M.RICCHIUTO, H.DECONINCK, A conservative formulation of the multidimensional upwind residual distribution schemes for general nonlinear conservation laws, *Journal of Computational Physics*, **179**(2):286–312, 2002.
- [3] H.DECONINCK, M.RICCHIUTO, Residual distribution schemes: Foundation and analysis, *Encyclopedia of Computational Mechanics*, Volume 3: Fluids, Wiley, 2007.
- [4] M.E.HUBBARD, Discontinuous fluctuation distribution, *Journal of Computational Physics*, **227**(24):10125–10147, 2008.
- [5] M.E.HUBBARD, M.RICCHIUTO, Discontinuous upwind residual distribution: A route to unconditional positivity and high order accuracy, *Computers and Fluids*, DOI 10.1016/j.compfluid.2010.12.023, 2010.
- [6] M.RICCHIUTO, A.BOLLERMANN, Stabilized residual distribution for shallow water simulations, *Journal of Computational Physics*, **228**:1071–1115, 2009.

¹School of Computing
University of Leeds
Leeds, LS2 9JT, UK
`m.e.hubbard@leeds.ac.uk`

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Modelling of Flood Problems based on a Two-Dimensional Well-Balanced Wave Propagation Algorithm with Bed Efflux /Influx Source Terms Including a Coupled Pipe Network Solver

H. Mahdizadeh¹ P.K. Stansby¹ B.D. Rogers¹

Keywords: Numerical Methods, Shallow Water Equations, Riemann solver, Underground pipe networks

Minisymposia: Recent advances in the numerical computation of environmental conservation laws with source terms

Parallel Session: Shallow water flows

Poster Session: Shallow water flows

Abstract Flood propagation over urban areas can cause an interaction between the free-surface flow and large underground pipe networks used for storm drainage and sewage, causing outflows and inflows at the bed. The associated waves may collide with each other and the surface waves. In this paper the shallow water equations are used to model this type of wave interaction over dry or wet beds with friction terms and sharp gradients in the bathymetry. The proposed shallow water scheme is solved based on finite volume high-resolution Godunov-type methods. The solver is well-balanced and uses a new type of wave speed based on the combination of exact and approximate Riemann speeds for propagation over nearly dry and dry states with depth-positive results. Additionally a new type of source term in the continuity equation defined by the authors [1] for 1-D problems is extended to 2-D to model pipe inflow and outflow conditions from underground sewage networks.

For standard 1-D problems the solver is used to model dam-break problems over wet and dry states with the interaction of efflux flows including flow over a step change in the bed. For problems with influx/efflux source terms comparisons are made with STAR-CD, a commercial Navier-Stokes solver for general fluid flow prediction. An efflux number, En , is defined based on the vertical efflux velocity and the gap length. The model gives good predictions at all times provided $En < 0.5$, and after the initial stages for $En > 0.5$ [1].

To solve two-dimensional problems the shallow water scheme uses the dimensional-splitting method which solves each one-dimensional Riemann problem in the x - and y -directions separately. The cross-derivative terms for second-order accuracy are incorporated by solving another Riemann problem in the orthogonal direction. Good agreement is obtained between the two-dimensional shallow water model and STAR-CD for the efflux numbers of $En < 0.5$.

For modelling the inundation problem over an underground pipe network the solver is coupled with a general underground pipe network solver to calculate the efflux discharges as the flood water pass through the pipe network. For analysing the pipe network with unknown effluxes an additional set of equations is incorporated into the solution of a general pipe network solver. The shallow water solver coupled to an underground pipe network is then used to simulate dam-break interaction with pipe networks and solid blocks with 9 and 25 nodes (Figures 1 and 2) which demonstrate the versatility of the method.

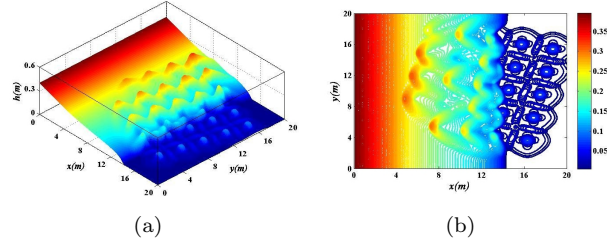


Figure 1: Dam-break propagation over sewage pipe network of 25 nodes for time $t = 4$ s (a) 3-D surface plot, (b) contour plot. Colours blue to red correspond to the water depth 0-0.6m.

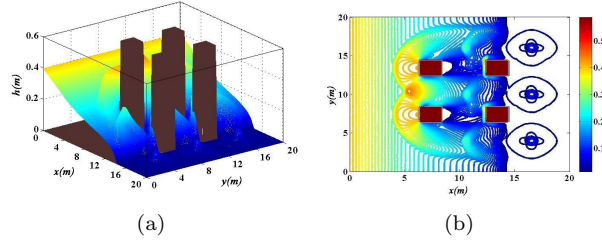


Figure 2: Dam-break propagation over sewage pipe network of 9 nodes with block interactions for time $t = 4$ s (a) 3-D surface plot, (b) contour plot. Colours blue to red correspond to the water depth 0-0.6m.

References

- [1] MAHDIZADEH H, STANSBY PK, ROGERS BD. On the Approximation of Local Efflux/Influx Bed Discharge in the Shallow Water Equations Based on a Wave Propagation Algorithm. *Int. J. Num. Meth. Fluids* DOI: 10.1002/fld.2314, 2010.

³School of Mech., Aerosp. and Civil Engineering
The University of Manchester, Manchester, M60 1QD, UK
ho.mahdizadeh@gmail.com, p.k.stansby@manchester.ac.uk,
benedict.rogers@manchester.ac.uk

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

A Large Time Step Upwind Scheme for the Shallow Water Equations with source terms

M. Morales-Hernández¹ J. Murillo² P. García-Navarro³ J. Burguete⁴

Keywords: Large Time Steps, Shallow Water Flows, Explicit Schemes, Source Terms

Minisymposia: Recent advances in the numerical computation of environmental conservation laws with source terms.

Abstract Numerical methods to predict the water profile and discharge variations in the case of steady and unsteady flows in hydraulic systems modelling have become a common tool. Upwind methods in particular have proved a suitable way to discretize the shallow water equations.

Finite difference schemes for time dependent equations are traditionally divided in two main groups, according to the way of discretization used for the time derivative, as explicit and implicit. Implicit schemes offer unconditional numerical stability at the extra cost of having to deal with the resolution of an algebraic, and often nonlinear, system with as many unknowns as grid points at every time step. On the other hand, conceptual simplicity is the most valuable characteristic of the former in which variables at a future time can be independently evaluated at every single point. The allowable time step size is nevertheless restricted by stability reasons to fulfil the Courant-Friedrichs-Lewy (CFL) condition.

It is possible to relax the condition over the time step size when using explicit schemes. A generalization of the first order explicit upwind and Roe method (see Roe 1981), modified to allow large time steps, was explored by Leveque (see Leveque 1981, Leveque 1985) first in the scalar non-linear case and then adapted to systems of equations. It becomes stable for CFL's larger than 1 and provides an accurate and correct solution of shocks. The technique is devised to cope well with flow transients and even discontinuities far from the boundaries, being able to give a resolution of the shocks even sharper for $CFL > 1$. When rarefactions containing a sonic point are present it needs some adjustments and the preferable procedure is not clear. A way to overcome that situation will be proposed and explored.

The extension of the explicit method considers situations with or without source terms. In the first case, they are discretized according to the upwind formulation (see Murillo et al., 2010). Also, the use of this technique at open and closed boundaries is considered.

The outline is as follows: the one dimensional discretization is described first, for a scalar equation. After, the performance of the scheme is evaluated to solve the shallow water equations and the way to deal with bed slope and friction source terms was

incorporated into the proposed procedure. Steady open channel flow problems with analytical solutions and where bed slope and friction source terms play an important role are used as validation test cases. Also this method has been tested for the ideal dam break problem and other Riemann problems with analytical solutions.

References

- [1] P.L. ROE. Approximate Riemann Solvers, Parameter Vectors and Difference Schemes. *J. Comp. Physics*, **43**, 357–372, 1981.
- [2] R.J. LEVEQUE. A Large Time Step Generalization of Godunov’s Method for System of Conservation Laws. *SIAM J. Numer. Anal.* **22**, 1051–1073, 1985.
- [3] R.J. LEVEQUE. Large Time Step Shock-Capturing Techniques for Scalar Conservation Laws. *Numerical Analysis Project, Manuscript NA-81-13, Stanford University, Stanford, 1981*
- [4] E.F. TORO. Riemann Solvers and Numerical Methods for Fluid Dynamics. *Springer, Berlin. 1997* p. 334, 1997.
- [5] P.L. ROE. Upwind Differencing Schemes for Hyperbolic Conservation Laws with Source Terms. *Proceeding of the 1st International Congress on Hyperbolic Problems. Springer: Heidelberg*, 41–51, 1986.
- [6] J. BURGUETE AND P. GARCÍA-NAVARRO. Efficient Construction of High-Resolution Conservative Schemes for Equations with Source Terms: Application to Shallow Water Flows. *International Journal for Numerical Methods in Fluids* **37**, 209–248, 2001.
- [7] E.F. TORO. Shock-Capturing Methods for Free Surface Shallow Flows. *Wiley, New York. 1997*
- [8] J. MURILLO AND P. GARCÍA-NAVARRO. Weak Solutions for Partial Differential Equations with Source Terms. *J. Comp. Physics*, **229**, 4327–4368, 2010.
- [9] J. MURILLO, P. GARCÍA-NAVARRO, P. BRUFAU AND J. BURGUETE. Extension of an Explicit Finite Volume Method to Large Time Steps ($CFL > 1$): Application to Shallow Water Flows *International Journal for Numerical Methods in Fluids* **50**, 63–102, 2006.

¹Fluids Mechanics
CPS, University of Zaragoza
50018 Zaragoza, Spain
`mmorales@unizar.es`

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Augmented Roe's approaches for Riemann problems including source terms: definition of stability region with application to the shallow water equations with rigid and deformable bed.

Javier Murillo¹ P. García-Navarro¹

Keywords: Hyperbolic Equations, Source terms, approximate solutions, stability region

Minisymposia: Recent advances in the numerical computation of environmental conservation laws with source terms.

Parallel Sessions: Shallow water flows.

Abstract Approximate solutions of problems with m equations with source terms can be defined using an augmented Riemann solver with $m + 1$ states instead of increasing the number of involved equations. These weak solutions use propagating jump discontinuities connecting the $m+1$ states to approximate the Riemann solution, and are of great interest when applied to the shallow water equations in complex scenarios. The average of the propagated waves in the computational cell leads to a reinterpretation of the Roe's approach and in the upwind treatment of the source term of Vázquez-Cendón. It is derived that the numerical scheme can not be formulated evaluating the physical flux function at the position of the initial discontinuities, as usually done in the homogeneous case. From the analysis of the approximate solutions it is possible to establish the water depth positivity requirement, providing correct rules to control the global stability of the method. The stability region is shown to differ from the one defined for the case without source terms, explaining the appearance of negative values of the thickness of the water layer in the proximity of the wet/dry front. In cases of deformable bed the complete system can be formed by a set of coupled partial differential equations and it is possible to define a genuinely Roe-type first order scheme, leading to an augmented Roe Riemann solver, by means of the reconstruction of a Jacobian matrix that integrates both fluxes and source terms. In this way it is also possible to provide correct rules that control the global stability of the method.

References

- [1] D.L. GEORGE. Augmented Riemann solvers for the shallow water equations over variable topography with steady states and inundation. *Journal of Computational Physics* **227**, 3089–3113, 2008.

- [2] S.K. GODUNOV. A difference method for numerical calculation of discontinuous solutions of the equations of hydrodynamics. *Mat. Sb.* **47**, 271–306, 1959.
- [3] J. MURILLO, P. GARCÍA-NAVARRO AND J. BURGUETE. Conservative Numerical Simulation of Multicomponent Transport in Two-Dimensional Unsteady Shallow Water Flow, *Journal of Computational Physics*, **228**, 5539–5573, 2009
- [4] J. MURILLO, P. GARCÍA-NAVARRO. Weak solutions for partial differential equations with source terms: Application to the shallow water equations, *Journal of Computational Physics* **229**, 4327–4368, 2010.
- [5] J. MURILLO, P. GARCÍA-NAVARRO. An Exner-based coupled model for two-dimensional transient flow over erodible bed, *Journal of Computational Physics* **229**, 8704–8732 2010.
- [6] P.L. ROE. *Some contributions to the modelling of discontinuous*. Lect. Notes Appl. Math., 22 163–193, 1985.
- [7] P.L. ROE. *Approximate Riemann solvers, parameter vectors, and difference schemes*, *J. Comput. Phys.*, **43**, 357–372, 1981
- [8] R. BERNETTI, V.A. TITAREV, E.F. TORO. Exact solution of the Riemann problem for the shallow water equations with discontinuous bottom geometry. *Journal of Computational Physics* **227**, 3212–3243, 2008.
- [9] M.E. VÁZQUEZ-CENDÓN. Improved treatment of source terms in upwind schemes for the shallow water equations in channels with irregular geometry. *Journal of Computational Physics* **148**, 497–498, 1999.

¹Fluid Mechanics
 Universidad de Zaragoza
 C.P.S. 50018 Zaragoza, Spain
 Javier.Murillo@unizar.es
<http://ghc.unizar.es>

Balancing source terms revisited in depth-averaged Smoothed Particle Hydrodynamics (SPH)

R. Vacondio¹ B.D. Rogers² P.K. Stansby² P. Mignosa¹

Keywords: Numerical Methods, Shallow Water Equations, Source Terms, Balanced formulation, Smoothed Particle Hydrodynamics

Minisymposia: Recent advances in the numerical computation of environmental conservation laws with source terms

Parallel Session: Shallow water flows

Poster Session: Shallow water flows

Abstract In this paper a Smoothed Particle Hydrodynamics (SPH) numerical model for SWEs with slope source term balancing is presented. The solution of the Shallow Water Equations (SWEs) using smoothed particle hydrodynamics (SPH) is attractive being a mesh-free, automatically adaptive method without special treatment for wet/dry interfaces. Recently the capability of the SWE-SPHs numerical scheme of being conservative and shock capturing has been shown obtaining results similar to Finite Volume [5]. The capability of the numerical model to reproduce steady state at rest is considered a fundamental feature to obtain reliable simulations of floodings over bathymetry, and it has been obtained by different authors in the framework of Eulerian schemes such as Finite Volume [1, 2, 3, 4]. To the authors' knowledge the formulation proposed here is the first well-balanced correction for Lagrangian schemes such as SPH.

In the classical SPH formulation for the SWEs the water depth is calculated directly by means of an SPH approximation through the fluid particles, the bottom source term is calculated using a different set of particles called bottom particles. The key idea adopted herein is to link these two formulations: the water depth of each fluid particle is calculated as the difference between the water surface and the bottom elevation; moreover a new, more accurate method for the bed slope source term based on the fluid particles is also adopted.

The technique is first applied to the simulation of still water in presence of a bottom discontinuity. Figure (1) shows the Froude number and the water elevation at time 2 s: the Froude numbers are of the order of magnitude of 10^{-5} , and they are due to the residual error in the SPH interpolation used for the water surface elevation. Figures (2 - 3) show the results of two dam breaks over a step with and without wet bed, compared against the analytical solutions, the Figures (2 - 3) show the same test cases using the classical SPH-SWEs formulation. With no balancing correction the SPH scheme is not able to reproduce the analytical solution, whereas the numerical results obtained with the correction are in good agreement with the analytical solution.

References

- [1] GARCÍA-NAVARRO P, VÁZQUEZ-CENDÓN ME. On numerical treatment of the source terms in the shallow water equations. *Computers & Fluids* **29**, 95179, 2000.

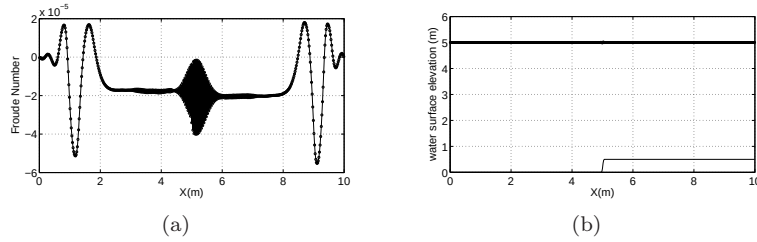


Figure 1: still water simulation over a step: non dimensional velocity (a) and water surface elevation (b) at time 2 s

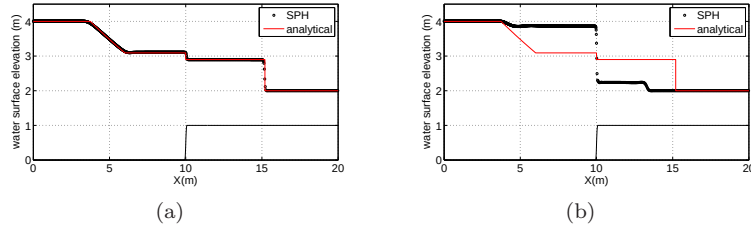


Figure 2: Riemann problem of SWE with a bottom step with wet bed: water surface elevation at time 1 s (a) simulation *with* step correction, (b) *without* step correction

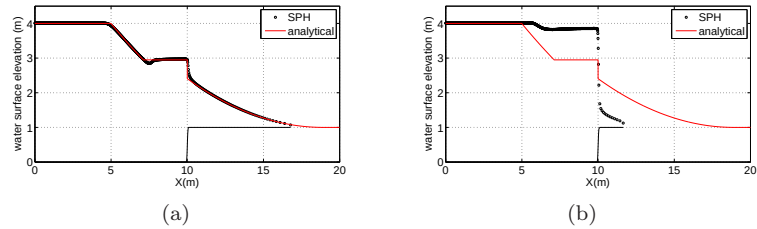


Figure 3: Riemann problem of SWE with a bottom step with drybed : water surface elevation at time 0.8 s (a) simulation *with* step correction, (b) *without* step correction

- [2] ZHOU JG, CAUSON DM, MINGHAM CG, INGRAM DM. The surface gradient method for the treatment of source terms in the shallow water equations. *Journal of Computational Physics* **168**(1), 1-25, 2001.
- [3] VALIANI A, BEGNUDELLI L. Divergence Form for Bed Slope Source Term in Shallow Water Equations. *J. of Hydraulic Engineering*. **132**(7), 652-665, 2006.
- [4] LIANG Q, MARCHE F. Numerical resolution of well-balanced shallow water equations with complex source terms. *Adv. Water Resources* **32**(6), 873-884, 2009.
- [5] VACONDIO R, ROGERS BD, STANSBY PK, MIGNOSA P. SPH numerical model for shallow flow with open boundaries and real flooding simulations. *Journal of Hydraulic Engineering* (submitted), 2011.

¹Department of Civil Engineering, Parma University
43100, Parma, Viale G.P. Usberti 181/A Italy
renato.vacondio@unipr.it, paulo.mignosa@unipr.it

²School of Mechanical, Aerospace and Civil Engineering
The University of Manchester, Manchester, M60 1QD, UK
benedict.rogers@manchester.ac.uk, p.k.stansby@manchester.ac.uk

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Flooding simulations with subgrids

V. Casulli¹ G.S. Stelling²

Minisymposia: Recent advances in the numerical computation of environmental conservation laws with source terms

Abstract To simulate floodings due to dam breaks or other causes such as rain storms and storm surges, two flow types play an important role: overland flow and rapidly varying flows. Overland flow is dominated by friction and rapidly varying flows, such as bores and hydraulic jumps, are dominated by conservation laws. GIS systems now a day contain very detailed DEM's (digital elevation models) with more data points than can be handled by a numerical simulation of flooding of a large area. The sub-grid method, as proposed by Casulli [1], however allows the use of very detailed bathymetric data without the need for flow variables at every pixel.

This contribution shows optimal flooding simulations that are based on this sub-grid approach. Special pixel integrated treatment of non-linear volumes, of momentum conservation and of bottom friction allows accurate solutions for realistic and detailed terrain models. The flows include rapid local variations and overland flow with frontal contact discontinuities. A number of examples will be given.

References

- [1] CASULLI V., A high-resolution wetting and drying algorithm for free-surface hydrodynamics, *International Journal for Numerical Methods in Fluids*, **60**:391–408, 2009.

²Delft University of Technology
Faculty of Civil Engineering and Geosciences
P.O.Box 5048
2600 GA Delft
The Netherlands
G.S.Stelling@tudelft.nl

Numerical Methods for Hyperbolic Equations: Theory and Applications

An international conference to honour Professor E.F. Toro
in the month of his 65th birthday

Minisymposia

Seismology and geophysics
modelling

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Dynamics of gravitational instabilities on Earth and on Mars

Anne Mangeney¹

Antoine Lucas, Pascal Favreau, Lev Tsimring, François Bouchut

Keywords: Gravitational instabilities, debris flows, avalanches

Minisymposia: Seismology and geophysics modelling

Abstract Gravitational instabilities such as debris flows, landslides or avalanches play a key role in erosion processes on the Earth's surface and represent one of the major natural hazards threatening life and property in mountainous, volcanic, seismic and coastal areas. A great amount of experimental, numerical and field studies have been devoted to analyze the physical processes involved in gravitational flows and to propose rheological models describing their behavior. Even though recent rheological models for granular materials are able to reproduce some basic features of granular flows in laboratory, they fail in simulating real landslides with realistic rheological parameters. As a result, the physical processes at work during debris flows or landslides are still an open question. In particular, there is no consensus to explain the high mobility of natural avalanches.

A new model able to describe natural debris flows over complex topography is presented here. At the same time, the models currently used for avalanche modeling and their predictive power will be discussed, in the light of laboratory experiments on granular flows and of application to real cases on Earth and on Mars. Finally, a new methodology for measuring the dynamics of natural landslides will be proposed based on the study of seismic waves generated by these gravitational flows. This approach consists of solving two systems of hyperbolic equations: the equations for granular flow dynamics and those for seismic wave propagation. We show that simulation of the seismic signal generated by landslides makes it possible to discriminate between possible alternative scenarios for flow dynamics and to provide first estimates of the rheological parameters. As landslide generated seismic waves are continuously recorded by seismic networks, our results provide a new way to collect data on the dynamics and rheology of natural flows.

¹Anne Mangeney,
Professor at University Paris-Diderot,
Institut de Physique du Globe de Paris
mangeney@ipgp.fr

Two-phase models for debris flows

F. Bouchut¹ E.D. Fernández-Nieto² A. Mangeney³ G. Narbona-Reina²

Keywords: Debris flows, avalanches, two-phases, buoyancy.

Minisymposia: Seismology and geophysics modelling

Abstract This work deals with the mathematical modelling of some geophysical mass flows regarding a mixture of solid material and interstitial fluid in order to simulate avalanches evolution.

Savage and Hutter presented in 1989 a pioneering work on the study of aerial dry granular avalanches. A model of shallow water type in local coordinates on an inclined plane is obtained. This model includes a Coulomb friction term that depends on the internal friction angle.

Interaction forces between solid and fluid phases may play an important role in landslides. Iverson and Denlinger argued in [5] that this fact should be included in the constitutive behavior of the flowing material, so in this paper the authors developed a system of mass and momentum balance laws for the mixture.

Anderson and Jackson presented in [2] the so called two-phase model based on an averaging of mass and momentum balance laws for fluid and solid separately. Interaction between the two phases are modelled including buoyancy effects.

Pitman and Le proposed in [3] an averaged model considering mass and momentum equations for both the liquid and the solid phase based on the previous equations of Anderson and Jackson.

In [4] a modified Pitman and Le model is analyzed and numerically solved. The presented two-phase system differs from the original one in the description of the fluid phase dynamics, and it has the new property of recovering a conservative equation for the momentum of the mixture.

In this work we propose a two-phase model for the study of partially fluidized aerial avalanches. This model is also based in the equations proposed in [2]. It considers mass and momentum equations for both liquid and solid phases and buoyancy effects. The main difference with Pitman-Le model is the existence of an associated energy.

References

- [1] S. B. SAVAGE, K. HUTTER. The motion of a finite mass of granular material down a rough incline. *J. Fluid Mech.* **199**, 177–215, 1989.
- [2] T.B. ANDERSON, R. JACKSON. A fluid mechanical description of fluidized beds. *Ind. Eng. Chem. Fundam.* **6**, 527–539, 1967.
- [3] E.B. PITMAN, L. LE. A two-fluid model for avalanche and debris flows. *Phil. Trans. R. Soc. A* **363**, 1573–1601, 2005.

- [4] M. PELANTI, F. BOUCHUT, A. MANGENEY. A Roe-type scheme for two-phase shallow granular flows over variable topography. *M2AN* **42**, 851–885, 2008.
- [5] R.M IVERSON, R.P. DENLINGER.: Flow of variably fluidized granular masses across three-dimensional terrain 1: Coulomb mixture theory. *J. Geophys. Res.* **106**, 537–552, 2001.

¹Laboratoire d'Analyse et de Mathématiques Appliquées
 LAMA - UMR 8050
 Université Paris-Est - Marne-la-Vallée
 77454 Marne-la-Vallée cedex 2, France
 francois.bouchut@univ-mlv.fr

²Departamento de Matemática Aplicada I
 Universidad de Sevilla
 E.T.S. Arquitectura, 41012 Sevilla, Spain
 edofer@us.es, gnarbona@us.es

³Institut de Physique du Globe de Paris
 Université Paris Diderot 7
 75005 Paris, France
 mangeney@ipgp.jussieu.fr

Meshless methods for lava flow modeling and simulation[†]

G. Bilotta G. Russo¹ A. H  rault C. Del Negro A. Vicari²

Keywords: meshless methods, SPH, MLS, lava flow, fluid/solid interaction

Minisymposia: Seismology and geophysics modelling

Abstract We present an innovative approach based on mesh-free (particle) methods for the simulation of the fluid-dynamics and thermal aspects of lava flows, a challenging task with critical importance in applications such as volcanic hazard assessment. The coupled fluid/thermal equations are solved with a Smoothed Particle Hydrodynamics (SPH) approach, with corrections based on the Moving Least Squares (MLS) method to improve the accuracy in the computation of second-order derivatives. The model is extended to include solid/fluid interactions within the same purely mesh-free method (figure 1).

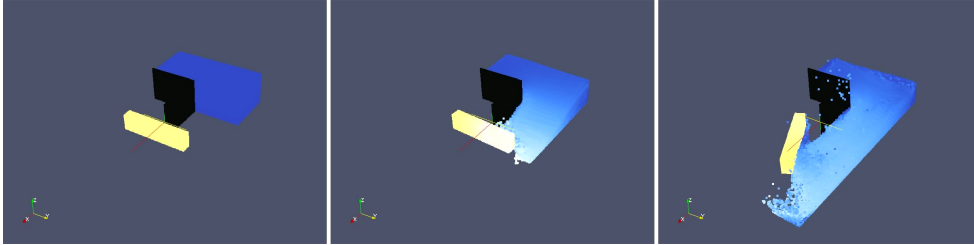


Figure 1: Pure SPH simulation of water pushing a block

Our approach allows us to easily model a variety of different rheologies, and in particular Newtonian, Bingham and Hershell-Bulkley fluids, with constant as well as temperature-dependent viscosity parameters. Complete information about all properties of the flow can be easily extracted (figure 2). Prospective applications include volcanic hazard assessment as well as verification and advancements in the physical models for lava, and for relationships, such as the one between effusion rate and thermal flux, which have important applications in monitoring and forecasting.

[†]Work undertaken with the financial support from the V3-LAVA project (INGV-DPC 2007-2009 contract)

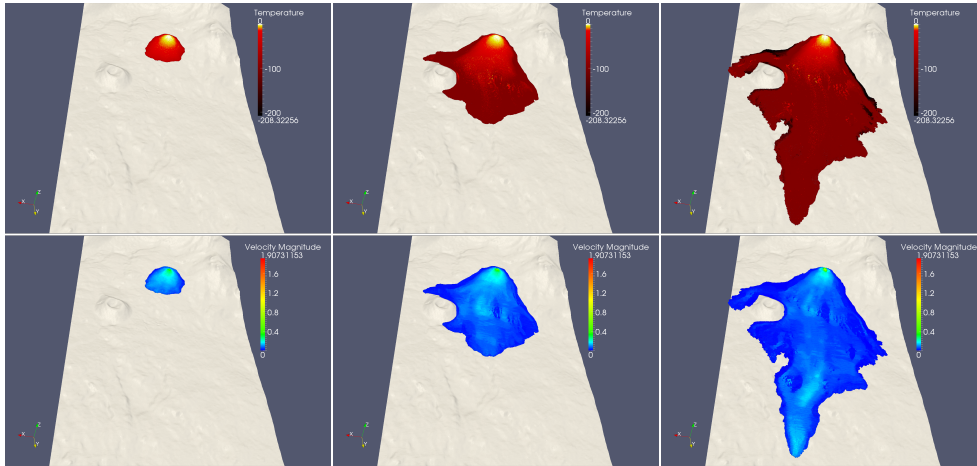


Figure 2: Temperature and velocity evolution during a simulated lava flow

¹Dipartimento di Matematica e Informatica
 Università di Catania
 viale A. Doria, 6
 95125 Catania, ITALY
 corresponding author: G. Bilotta
 bilotta@dmi.unict.it

²Istituto Nazionale di Geofisica e Vulcanologia
 Sezione di Catania
 piazza Roma, 2
 95125 Catania, ITALY

Finite volume schemes for balance laws on time-dependent surfaces

J. Giesselmann¹

Keywords: Conservation laws with source terms, Finite volume schemes, Riemannian manifolds

Minisymposia: Seismology and geophysics modelling

Abstract In many applications conservation laws are not posed in Euclidean space but on surfaces or more generally Riemannian or Lorentzian manifolds. Such problems and their solution by finite volume schemes have been investigated in several papers [1, 2, 5, 7]. The problem becomes more involved, when the geometry of the manifold changes in time. This might occur when we are interested in processes taking place on the surface of (biological) cells, e.g. receptor clustering, or on the interfacial manifold between different phases in three dimensional multiphase flow [4]. Given a scalar balance law on some moving, closed surface $M(t)$, we obtain - by mapping the surface to a reference configuration using isometrics at each time - a problem on a fixed, closed manifold M equipped with a family of Riemannian metrics $\{g(t)\}_{t \in \mathbb{R}_+}$ changing in time:

$$u_t + \nabla_{g(t)} \cdot f(x, t, u) = -\frac{1}{2} \text{tr}_{g(t)}(g_t(t)) + q(x, t, u) \quad \text{in } M \times \mathbb{R}_+, \quad (1)$$

$$u(\cdot, 0) = u_0 \quad \text{on } M. \quad (2)$$

Here $u(x, t) \in \mathbb{R}$ is the unknown, $f(x, \cdot, \cdot) : \mathbb{R}_+ \times \mathbb{R} \rightarrow T_x M$ for every $x \in M$, is the flux function, $q : M \times \mathbb{R}_+ \times \mathbb{R} \rightarrow \mathbb{R}$ is the source term and $u_0 \in L^\infty(M)$ is the initial data. In addition $\text{tr}_{g(t)}(g_t(t))$ denotes the trace of the time derivative $g_t(t)$ with respect to $g(t)$ and $\nabla_{g(t)} \cdot$ is the divergence operator induced by $g(t)$. Problem (1) fits (for $q = 0$) into the very general framework of finite volume schemes based on differential forms [6]. However to treat source terms and not divergence-free fluxes, we remain with the more tangible setting (1).

Given a triangulation of M with cells K and faces e , and a time sequence $0 = t^0 < t^1 < \dots$ we propose a finite volume scheme given by

$$u_K^0 = \frac{1}{m_0(K)} \int_K u_0(x) dv_{g(0)}(x), \quad (3)$$

$$u_K^{n+1} = u_K^n - \sum_{e \in \partial K} \frac{t^{n+1} - t^n}{m_{t_{n+1}}(K)} f_{K,e}^n(u_K^n, u_{K_e}^n) + q_K^n, \quad (4)$$

$$q_K^n = \frac{1}{m_{t_{n+1}}(K)} \int_{t^n}^{t^{n+1}} \int_K q(x, t, u_K^n) dv_{g(t)} dt - u_K^n \frac{m_{t_{n+1}}(K) - m_{t_n}(K)}{m_{t_{n+1}}(K)}, \quad (5)$$

$$u^h(x, t) = u_K^{n+1} \text{ when } t \in [t^n, t^{n+1}), x \in K, \quad (6)$$

where $m_t(K)$ denotes the volume of K at time t , K_e is the cell sharing the face e with K , the $f_{K,e}^n$ are numerical flux functions satisfying conservation, consistency and monotonicity conditions and $dv_{g(t)}$ is the volume form induced by $g(t)$.

We present numerical experiments and, generalizing ideas from the Euclidean setting [3], a convergence analysis of the scheme (3)-(6).

References

- [1] P. AMORIM, P.G. LEFLOCH AND B. OKUTMUSTUR. Finite volume schemes on Lorentzian manifolds, *Commun. Math. Sci.*, **6**, 1059–1086, 2008
- [2] M. BEN-ARTZI AND P. G. LEFLOCH. Well-posedness theory for geometry-compatible hyperbolic Error estimate for conservation laws on manifolds *Ann. Inst. H. Poincaré Anal. Non Linéaire*, **24**, 989–1008, 2007
- [3] C. CHAINAIS-HILLAIRET AND S. CHAMPIER. Finite volume schemes for nonhomogeneous scalar conservation laws: error estimate, *Numer. Math.*, **88**, 607–639, 2001
- [4] W. DREYER, J. GIESSELMANN, C. KRAUS AND C. ROHDE. Asymptotic analysis for Korteweg models, *SimTech Preprint 2010-67*, <http://www.simtech.uni-stuttgart.de/publikationen/preprints10.php>, submitted to *Interfaces and Free Boundaries*,
- [5] J. GIESSELMANN. A convergence result for finite volume schemes on Riemannian manifolds, *M2AN Math. Model. Numer. Anal.*, **43**, 929–955, 2009
- [6] P. G. LEFLOCH AND B. OKUTMUSTUR. Hyperbolic conservation laws on spacetimes. A finite volume scheme based on differential forms *Far East J. Math. Sci. (FJMS)* **31**, 49–83, 2008
- [7] P. G. LEFLOCH, B. OKUTMUSTUR AND W. NEVES. Hyperbolic conservation laws on manifolds. An error estimate for finite volume schemes, *Acta Math. Sin. (Engl. Ser.)*, **25**, 1041–1066, 2009

¹University of Stuttgart

Institute of applied analysis and numerical simulation

Pfaffenwaldring 57

D-70569 Stuttgart

Germany

jan.giesselmann@mathematik.uni-stuttgart.de

<http://www.ians.uni-stuttgart.de/am/Giesselmann/>

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Finite volume schemes for nonlinear dispersive wave equations

D. Dutykh¹ Th. Katsaounis² D. Mitsotakis³

Keywords: Water waves, dispersive waves, Boussinesq equations, finite volumes, runup

Minisymposia: Seismology and geophysics modelling

Abstract Finite volume schemes are commonly used to construct approximate solutions to conservation laws. In this study we extend the framework of the finite volume methods to dispersive water wave models, in particular to Boussinesq type systems [1]. We focus mainly on the application of the method to bidirectional nonlinear, dispersive wave propagation in one space dimension. Special emphasis is given to important nonlinear phenomena such as solitary waves interactions, dispersive shock wave formation and the runup of breaking and non-breaking long waves [2].

References

- [1] D. DUTYKH, TH. KATSAOUNIS, D. MITSOTAKIS. Finite volume methods for unidirectional dispersive wave models. *Submitted*, 2011.
<http://hal.archives-ouvertes.fr/hal-00538043/>
- [2] D. DUTYKH, TH. KATSAOUNIS, D. MITSOTAKIS. Finite volume schemes for dispersive wave propagation and runup. *J. Comput. Physics* **230**, 3035–3061, 2011.

¹LAMA, UMR 5127 CNRS
Université de Savoie, Campus Scientifique
73376 Le Bourget-du-Lac, France
Denys.Dutykh@univ-savoie.fr
<http://www.lama.univ-savoie.fr/~dutykh/>

²Department of Applied Mathematics
University of Crete, 71409 Heraklion, Greece
thodoros@tem.uoc.gr
<http://www.tem.uoc.gr/~thodoros/>

³IMA, University of Minnesota
Minneapolis MN 55455, USA
dmitsot@gmail.com
<http://sites.google.com/site/dmitsot/>

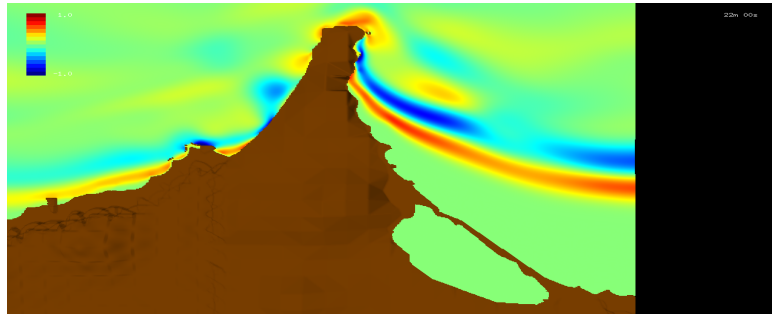
Tsunami modelling using high order finite volume schemes on GPUs

M.J. Castro¹ M. de la Asunción² E.D. Fernández-Nieto² J.M. González-Vida¹ J. Macías¹ J.M. Mantas³ T. Morales¹ S. Ortega¹ C. Parés¹

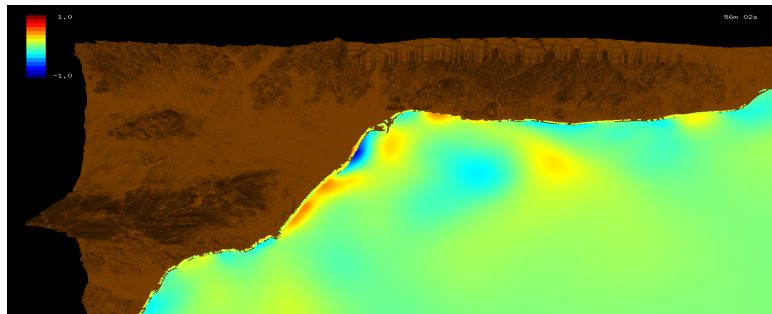
Keywords: Tsunami modelling, High order finite volume schemes, GPUs, CUDA

Minisymposia: Seismology and geophysics modelling

Abstract In this talk, we present a two-layer model of Savage-Hutter type to model tsunamis generated by landslides. A layer composed of fluidized granular material is assumed to flow within an upper layer composed of an inviscid fluid (e.g. water). The sediment layer is modelled by a Savage-Hutter type model where buoyancy effects have been considered (see [5] for more details).



(a) Cape Tres Forcas/Melilla. Time $t = 22$ min.



(b) Spanish coast (Málaga). Time $t = 56$ min.

Figure 1: Free surface at different locations and times in the Alboran Sea.

The system is discretized using a high order PVM finite volume scheme. PVM (Polynomial viscosity matrix) schemes are defined in terms of viscosity matrices computed by a suitable polynomial evaluation of a Roe matrix (see [2] and [3]). These methods have the advantage that they only need some information about the eigenvalues of the system to be defined, and no spectral decomposition of Roe Matrix is needed. As a consequence, they are faster than Roe method. In fact, these methods can be seen as a generalization of the schemes introduced by Degond et al. in [4]. We will see that some well known solvers such as Rusanov, Lax-Friedrichs, FORCE ([6], [1]), GFORCE (see [7], [1]) or HLL can be written like a PVM scheme.

Finally some hints about the efficient implementation of such schemes on GPUs will be given and realistic applications will be shown, as the simulation of a tsunami generated by a submarine landslide in the Alboran Sea (see Figure 1).

References

- [1] M.J. Castro, A. Pardo, C. Parés, and E.F. Toro. *On some fast well-balanced first order solvers for non-conservative systems*. Math. Comp. 79, 1427-1472, 2010.
- [2] E.D. Fernández-Nieto, M.J. Castro and C. Parés. *On an Intermediate Field Capturing Riemann Solver Based on a Parabolic Viscosity Matrix for the Two-Layer Shallow Water System*. J. Sci. Comput., doi: 10.1007/s10915-011-9465-7, 2011.
- [3] M.J. Castro, E.D. Fernández-Nieto. *A class of computationally fast first order finite volume solvers: PVM methods*. Submitted to SIAM J. Sci. Comput., 2010.
- [4] P. Degond, P.F. Peyrard, G. Russo, Ph. Villedieu. *Polynomial upwind schemes for hyperbolic systems*. C. R. Acad. Sci. Paris 1 328, 479-483, 1999.
- [5] E.D. Fernández, F. Bouchut, D. Bresh, M.J. Castro, A. Mangeney. *A new Savage-Hutter type model for submarine avalanches and generated tsunamis*. J. Comp. Phys., 227: 7720-7754, 2008.
- [6] E.F. Toro and S.J. Billett. *Centred TVD schemes for hyperbolic conservation laws*. IMA J. Numer. Anal. 20(1): 4779, 2000.
- [7] E.F. Toro and V.A. Titarev. *MUSTA fluxes for systems of conservation laws*. J. Comput. Phys. 216(2): 403429, 2006.

¹Dpto. Análisis Matemático
 Universidad de Málaga
 Campus Teatinos s/n, 29071 Málaga, Spain
 [castro,vida,macias,morales,pares]@anamat.cie.uma.es

²Dpto. Lenguajes y Sistemas Informáticos
 Universidad de Granada
 [marc,jmmantas]@ugr.es

³Departamento de Matemática Aplicada I
 Universidad de Sevilla.
 E.T.S. Arquitectura. Avda, Reina Mercedes, s/n. 41012 Sevilla, Spain
 edofer@us.es

On multilayer shallow water systems

T. Chacón Rebollo¹, E.D. Fernández-Nieto² and E.H. Koné³

Keywords: Shallow Water System, Multilayer Modeling, Hydrodynamical Modeling, Navier-Stokes

Minisymposia: Seismology and geophysics modelling

Abstract The work is devoted to the derivation of a multilayer shallow water model. Considering flows with large friction coefficients, with significant water depth or with important wind effects, the horizontal velocity can hardly be approximated by a vertically constant velocity as in the classical shallow water system. In [2] multilayer shallow water model is proposed, by considering a constant profile of the horizontal velocity at each layer and by including mass and momentum exchange terms between the layers.

In this work, starting from the 2D instationary incompressible Navier-Stokes equations and following the technique introduced in [1], we derive a model where the unknowns are the horizontal and vertical velocity as well as the pressure. The horizontal velocity averaging in each layer and the incompressible property lead to a mass equation involving the mass exchanges through the upper and lower interfaces of this layer. The system with that mass equation and the 2D Navier-Stokes problem is first time-discretised in each layer leading to a non linear problem, which is next written in a 2D mixed variational form. Afterwards, the proposed model is obtained by the projection of the 2D variational form on subspaces chosen by specifying the dependence on the vertical variable of the unknown functions.

As a particular case we obtain a multilayer shallow water model with hydrostatic pressure, where the unknowns are the height of the fluid and the horizontal velocities at each layer, similarly to the model proposed in [2]. Although the definition of the momentum transference terms are different for these two models.

Finally, we present some numerical tests.

References

- [1] MOHAMED AMARA , DANIELA CAPATINA AND DAVID TRUJILLO. Variational Approach for the Multiscale Modeling of an Estuarine River. Part1: Derivation and Numerical Approximation of a 2D Horizontal Model. *Report, INRIA, 2008, no RR-6742* <http://hal.inria.fr/inria-00342858/en/>
- [2] E. AUDUSSE, M. O. BRISTEAU, B. PERTHAME AND J. SAINT-MARIE. A Multilayer Saint-Venant System with Mass Exchanges for Shallow Water Flows. Derivation and Numerical Validation. *M2AN Math. Model. Numer. Anal.* **45**, 169-200, DOI: 10.1051/m2an/2010036, 2011.

¹Dpto. Ecuaciones Diferenciales y Análisis Numérico
Universidad de Sevilla
Faculty of Mathematics.
`chacon@us.es`

²Dpto. Matemática Aplicada I
Universidad de Sevilla
E.T.S. Arquitectura
`edofer@us.es`

³Dpto. Matemática Aplicada I
Universidad de Sevilla
E.T.S. Arquitectura
`kone_elhadji@yahoo.fr`

Numerical Methods for Hyperbolic Equations: Theory and Applications

An international conference to honour Professor E.F. Toro
in the month of his 65th birthday

Parallel Sessions

Numerical Methods for Hyperbolic Equations: Theory and Applications

An international conference to honour Professor E.F. Toro
in the month of his 65th birthday

Parallel Sessions

High Order Methods for
Hyperbolic Conservation Laws

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Comparison among different high order time integration methods useful for explicit schemes

J. Burguete¹ P. García-Navarro² B. Latorre³

Keywords: Numerical Methods, Hyperbolic Equations and E. F. Toro

Parallel Session: High order methods for hyperbolic conservation laws

Abstract There are three strategies considered useful to reach high order time integration:

1. The first idea is to search for high order approximations of the time derivative in order to formulate a single step method, as in [2] or in [4]. The main drawback of this technique in conservation laws is that it involves the calculation of differential tensors of increasing order and complexity depending on the desired accuracy.
2. A second family of methods is based on the Runge-Kutta approach. An interpolation is made among several simple substeps within every time step. The main advantage of these methods is that they ensure the TVD property provided that they are defined in a convenient form based on positive coefficients [1, 3].
3. As a third option the multistep methods involve the numerical solutions from previous time steps. These methods also ensure the TVD property when based on positive coefficients. The obvious difficulty is that there are no solutions available at the beginning of the simulation. Therefore, they need to be combined with one of the previous approaches.

In this work the three strategies are formulated and the results are compared. Also, the interest of building a combination of them is explored.

References

- [1] S. GOTTLIEB AND C. W. SHU. Total variation diminishing Runge-Kutta schemes. *Math. Comp.* **67**(221), 73–85, 1998.
- [2] P. D. LAX, B. WENDROFF. Systems of conservation laws. *Comm. Pure Appl. Math.* **13**, 217–237, 1960.
- [3] S. J. RUUTH, R. J. SPITERI. Two barriers on strong-stability-preserving time discretization methods.. *J. Sci. Comput.* **17**, 211–220, 2002.
- [4] V. A. TITAREV, E. F. TORO. ADER: Arbitrary high order Godunov approach. *J. Sci. Comput.* **17**, 609–618, 2002.

¹Suelo y Agua.
Estación Experimental de Aula Dei. CSIC.
Avda Montañana 1005. 50059 Zaragoza, Spain
`jburguete@eead.csic.es`

²Mecánica de Fluidos.
Universidad Zaragoza.
María de Luna 3. 50018 Zaragoza, Spain
`pigar@unizar.es`

³Suelo y Agua.
Estación Experimental de Aula Dei. CSIC.
Avda Montañana 1005. 50059 Zaragoza, Spain
`borja.latorre@eead.csic.es`

High resolution schemes: TVD region its dependence on smoothness parameter.

Ritesh Kumar Dubey¹

Keywords: High resolution schemes, flux limiters, TVD Region, Smoothness parameter, Hyperbolic Equations

Parallel Session: High order methods for hyperbolic conservation laws.

Abstract It has been around three decades since the CFD community is celebrating the class of high resolution (HRS) total variation diminishing (TVD) schemes for excellent results and theoretical support e.g. [3, 1, 2, 4, 5]. We can classify high resolution schemes into Central difference and upwind biased schemes. In this work we investigate the relation between the TVD region of high resolution schemes using flux limiters and its dependence on the smoothness parameter.

Among all, the approach proposed by Sweby can be considered a representative framework for designing central difference based high resolution total variation diminishing schemes using flux limiters [1]. More importantly a TV stability region is given for flux limiters to yield total variation diminishing schemes (Figure 1 left). Later tremendous development took place in designing a variety of high resolution schemes and flux limiters. The most unique feature for all these limiters is that they satisfy the TVD stability region proposed in [1]. It seems that no attempt is made to find alternate TVD region except in [4], where a general framework is proposed to construct upwind biased high resolution TVD schemes using flux limiters. A entirely new TV stability region and a class of new limiters for proposed schemes are designed which satisfies the TVD region (Figure 1 right). In this work, we give unifying relation for the well known TVD region for flux limiter proposed in [1] and [4]. This unification enables the chance to use flux limiters proposed in [4] with the classical flux limiter e.g. Superbee, min-mod, vanleer etc based high resolution schemes. Limiters which satisfy both class of TVD regions are investigated and applied to some high resolution schemes for comparative study.

Mathematics Subject Classification 2000: 35L65, 65M06, 65M12

References

- [1] P. K. SWEBY, High resolution schemes using flux limiters for Hyperbolic conservation Laws, *SIAM J. Numer. Anal.*, 21(1984), pp 995-1011.
- [2] H C YEE Construction of explicit and implicit symmetric tvd schemes and their applications, *J. Comp. Phys.* 68(1987), pp 151-179.

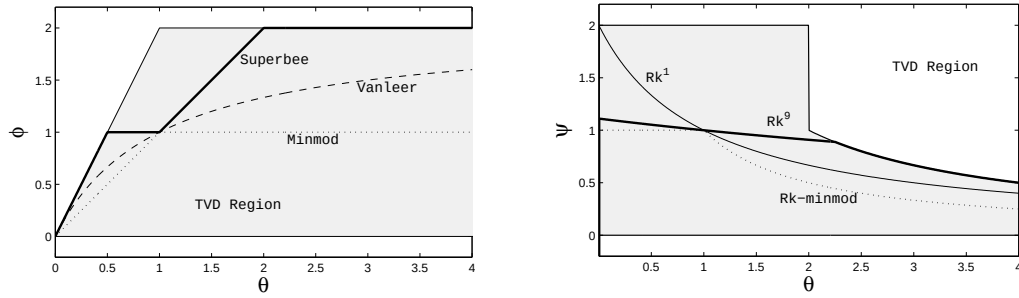


Figure 1: TVD regions and some limiter passing through them: Sweby(left), Kumar(right)

- [3] B. VAN LEER, Towards the ultimate conservative difference scheme II. Monotonicity and conservation combined in a second order scheme. *J. Comp. Phys.*, 14(1974), pp. 361-70.
- [4] M. K. KADALBAJOO, RITESH KUMAR, A high resolution total variation diminishing scheme for hyperbolic conservation law and related problems, *App. Math. and Comp.*, 175(2006), pp. 1556-1573.
- [5] P. L. ROE, Characteristic-based schemes for the Euler equations, *Ann. Rev. Fluid Mech.* 18(1986), pp. 337-365.

¹Department of Mathematics
Indian Institute of Technology Madras,
Chennai -600036, India
riteshkd@gmail.com

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

A numerical treatment of wet/dry zones in Well-Balanced Hybrid Schemes for Shallow Water Flow

R. Donat¹, A. Martinez-Gavara²

Keywords: Numerical Methods, Hyperbolic Equations

Parallel Session: High order methods for hyperbolic conservation laws.

Abstract In [1] we have proposed a novel numerical technique to obtain second order, high resolution shock capturing schemes for hyperbolic balance laws. It is based on the ‘homogeneous form’ of the balance law, as defined in [2], and makes use of flux-limiting concepts in hybrid schemes developed for homogeneous conservation law. The analysis in [1] is rather general, and identifies the key elements necessary to obtain a well balanced scheme.

In [3] we generalize this technique to hyperbolic systems of balance laws. The application to the shallow water system carried out in [3] shows that we indeed obtain a hybrid scheme with all the expected properties. Here we analyze the modifications needed in order for the scheme to solve situations where dry areas occur in shallow water flows.

Mathematics Subject Classification 2000: 65M06, 76M20, 35L65

References

- [1] R. DONAT AND A. MARTINEZ-GAVARA. Hybrid Second Order Schemes for Scalar Balance Laws. *To appear in J. Sci. Comput.* (2010).
- [2] LL. GASCÓN AND J. M. CORBERÁN. Construction of second-order TVD schemes for nonhomogeneous hyperbolic conservation laws. *J. Comput. Phys.* **172**, 261–297, 2001.
- [3] A. MARTINEZ-GAVARA AND R. DONAT. A Hybrid second order scheme for shallow water flows. *To appear in J. Sci. Comput.* (2010).

¹Applied Mathematics Department
University of Valencia
Faculty of Mathematics. 46100 Burjassot (Valencia), Spain
donat@uv.es

²Statistics and Operations Research Department
University of Valencia
Faculty of Mathematics. 46100 Burjassot (Valencia), Spain
gavara@uv.es

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

On the Analysis of a Solver for Generalized Riemann Problems by Asymptotic Expansion.

C.R. Goetz¹

Keywords: Generalized Riemann Problem, ADER Methods, Generalized Godunov Schemes.

Parallel Session: High order methods for hyperbolic conservation laws.

Abstract We consider high order generalized Godunov finite volume methods for the numerical solution of hyperbolic systems of conservation laws in one spatial dimension,

$$\partial_t u + \partial_x f(u) = 0, \quad x \in \mathbb{R}, t > 0, \quad (1)$$

with Cauchy data

$$u(x, 0) = \hat{u}(x), \quad x \in \mathbb{R}, t = 0. \quad (2)$$

Following the ADER state-expansion approach, we approximate the solution at the cell interfaces by a Taylor-expansion in time,

$$u(x_{i+1/2}, \tau) \approx u(x_{i+1/2}, 0_+) + \sum_{k=1}^{r-1} \partial_t^k u(x_{i+1/2}, 0_+) \frac{\tau^k}{k!}.$$

Toro and Titarev [2] suggested to compute the time derivatives at the origin that appear as coefficients in this expansion, $u(x_{i+1/2}, 0_+)$, $\partial_t^k u(x_{i+1/2}, 0_+)$, with the help of a generalized Riemann problem (GRP). The GRP is the initial value problem (1)-(2) where the initial data consists of piecewise smooth functions and is discontinuous at the origin. In a generalized Godunov scheme, one uses piecewise smooth functions that are reconstructed from cell averages.

The approach of Toro and Titarev reduces the GRP to a series of classical Riemann problems. First, time derivatives are expressed as functions of (lower order) spatial derivatives by a Cauchy-Kowalewskaya method and then a classical Riemann problem is solved for each spatial derivative. More precisely, evolution equations of the form

$$\partial_t(\partial_x^k u) + Df(u)\partial_x(\partial_x^k u) = H^k(u, \partial_x u, \dots, \partial_x^k u) \quad (3)$$

are used. These equations are then linearised around $u(x_{i+1/2}, 0_+)$ and the source terms are neglected, thus leaving only linear and homogeneous Riemann problems.

Since the solution of the GRP is discontinuous at the origin, it is not straightforward to show that these equations approximate the exact solution well. Equation (3) can be derived wherever the solution is smooth, but it is not clear, whether such an equation holds for discontinuous solutions. It seems to be an open question whether it is justified to work with the linearised equations.

We reinterpret the above solution methodology in the context of an asymptotic expansion for the GRP constructed by LeFloch and Raviart [1]. We show that the leading coefficients of polynomials appearing in their expansion can be interpreted as the higher order spatial derivatives of the solution at the origin. For *scalar* problems, we show that computing these coefficients is equivalent to solving the linear Riemann problems occurring in the GRP-solver of Toro and Titarev. We give some preliminary results on the question whether a similar argument could also work for the case of systems.

References

- [1] PH. LEFLOCH AND P.A. RAVIART. An asymptotic expansion for the solution of the generalized Riemann problem. Part I: General theory. *Ann. Inst. H. Poincaré Anal. Non Linéaire* **5**(2), 179–2007, 1988.
- [2] E.F. TORO AND V.A. TITAREV. Derivative Riemann solvers for systems of conservation laws an ADER methods. *J. Comput. Phys.* **212**,150–165, 2006.

¹University of Hamburg
Department of Mathematics
Bundesstr. 55 , 20146 Hamburg, Germany
`claus.goetz@math.uni-hamburg.de`

CFL-Number-dependent TVD-Limiters

Friedemann Kemm¹

Keywords: TVD, Limiter

Parallel Session: High order methods for hyperbolic conservation laws

Abstract Although with ENO and WENO schemes there are higher order alternatives, TVD schemes are still in widespread use. One reason for that is the simple implementation. Other reasons are the—compared to ENO and WENO—compact stencil and the robustness of the resulting schemes. Unfortunately, in the literature only a few limiters are treated in more detail. Many limiters are only found or cited in one or two papers. Some of them are just given as an incidental remark. As a result, only a small class of limiters is found in textbooks and therefore in practical use. This class is restricted in two senses: it only contains limiters which do not take into account the local CFL number, and it only contains symmetric limiter functions, i.e. the information on the upwind direction is not used. For implicit schemes or for the computation of steady state solutions, this is no severe restriction since in this case the choice of the limiter function is restricted to the above mentioned class and without too much impact on the quality of the results. For explicit codes there is much more freedom in the choice of limiters. It is possible to use CFL-dependent limiters, which allow for a better reproduction of amplitudes and even for third order schemes.

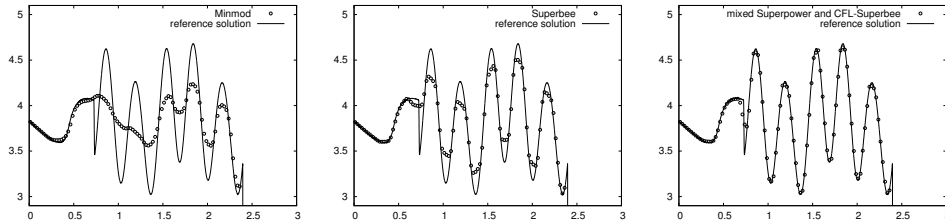


Figure 1: Shu-Osher problem (shock entropy wave interaction) at $t = 1.8$; results for Minmod, Superbee, and an advanced CFL-number dependent limiting strategy

In the last years some comparative studies of numerical schemes were published, which compare different types of schemes, e.g. TVD schemes to WENO schemes. Unfortunately all of them treat the schemes as a fixed one, with a fixed setting of parameters and a fixed limiter. It turns out that in some cases the results for the TVD scheme could be considerably improved by the choice of an alternative limiter. An example is shown in Figure 1.

In our talk, we present and classify limiters which take into account the local CFL-number. Besides the old limiters—most of them by Roe—we introduce some new variants. One of them is a modification of Roe’s original Ultrabee, but slightly less compressive, in order to reduce the squaring effect. Another limiter is based on a generalization of the Power-limiter by Serna and Marquina which, in turn, is based on a generalization of the harmonic mean. It turns out to be a fine choice for compressive nonlinear waves. Furthermore, we discuss some variants of Roe’s third order TVD-limiter.

We compare the results obtained with these limiters to standard limiters. The test cases include linear and nonlinear 1d-problems as well as some 2d-problems. Since the limiters are originally constructed for 1d linear advection, there are restrictions for the use on nonlinear waves, on systems, and in 2d- or 3d-settings. We show how the limiters may be modified for the use on nonlinear waves by using the nonlinear TVD-condition by Jeng and Payne.

Mathematics Subject Classification 2000: 65M08, 35L02, 35L40

References

- [1] F. KEMM. A comparative study of TVD limiters—well known limiters and an introduction of new ones. *International Journal for Numerical Methods in Fluids* (Accepted for publication), 2010.

¹Institute for Applied Mathematics and Scientific Computing
 Brandenburg University of Technology
 Konrad-Wachsmann-Allee 1, 03046 Cottbus, Germany
kemm@math.tu-cottbus.de
<http://www.math.tu-cottbus.de/~kemm/>

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Arbitrary High Order Schemes for Transport Problems.

B. Latorre¹ P. García-Navarro²

Keywords: high order methods, transport problems, Legendre polynomials, passive solute transport, shallow water model

Parallel Session: High order methods for hyperbolic conservation laws.

Abstract In this work, a method based on Legendre polynomials is proposed for the simulation of the passive transport of a concentration field. The formulation is conservative, explicit and made in a single step. The spatial accuracy is achieved by means of polynomial approximations by cell using Legendre series. This kind of spatial representation is also found in finite element discretization and allows for information on the variation of the fields at subgrid scale. The time resolution of the transport is based on both a numerical estimation of the displacement at the advection speed and a grid deformation, according to the Semi-lagrangian rules, followed by a projection of the solution on the fixed initial grid.

First, the simulation of the transport in 1D and 2D velocity fields will be presented. The main interest is focused on the analysis of accuracy and efficiency of the method when moving from order 1 to order 20 as compared to standard methods of virtual reconstruction. Even though high order methods are commonly used when a high accuracy is required, the interest in this work is in the study of the computational saving that can be achieved if the required accuracy is medium or low. This is possible thanks to the subgrid information that offers the possibility to solve problems with enough accuracy using only a few grid cells and high order polynomials. Furthermore, this enables the use of large time steps hence leading to low computational times.

In a second part, the method will be applied to the simulation of the transport of a passive solute in 1D and 2D shallow water flows. A technique has been developed to couple the Legendre schemes to any conservative method used for the resolution of the shallow water equations. The coupling offers the possibility to combine Legendre schemes and shallow water solvers of different order of accuracy, always enforcing conservative and monotone behaviour in the numerical solution of the solute concentration. This strategy is interesting since it is possible to require high order of accuracy only in the solute transport simulation, hence concentrating the computational effort in the component with more numerical error. The coupled methods will be applied to solve dam break problems with solute transport in 1D and 2D configurations including bed level variations (source terms in the flow equations) in order to analyze the performance of the proposed coupling technique. Finally, the coupled method will also be validated using laboratory experimental data of solute transport in a channel flow.

¹Suelo y Agua. Estación Experimental de Aula Dei. CSIC.
Avenida de Montañana 1005, 50059 Zaragoza, Spain
borja.latorre@csic.es - <http://ghc.unizar.es/>

²Fluid Mechanics Department. University of Zaragoza
Edificio Torres Quevedo, C/ María de Luna 3, 50018 Zaragoza, Spain
pigar@unizar.es - <http://ghc.unizar.es/>

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

An Five-Equation Model Based ALE Method for Compressible Multifluid Fluids.

Baolin Tian¹ Jin Qi¹ and Shuanghu Wang¹

Keywords: Multifluid flow, Five Equation model, ALE method, Mixed model

Minisymposia: Numerical methods in astrophysics, nuclear fusion and plasma physics, Multiphase flows and porous media

Parallel Session: High order methods for hyperbolic conservation laws

Abstract Multifluid flows can be found in a variety of science areas and practical applications, such as astrophysics and ICF. In this work, an ALE(Arbitrary Lagrangian-Eulerian) method coupled with five-equation model was proposed based on a novel closure model for multifluid mixed cells. The five-equation based ALE method is a two phase ALE method, including a Lagrangian phase and a rezone-remap phase. In the first phase, a set of multifluid Lagrangian dynamics equations was discretized with a cell-centered Lagrangian scheme, and a multifluid Riemann solver was proposed by extending traditional Riemann solver for mixed cells with different materials. The rezone-remap technique was taken from our previous work listed in references.

Closure model for mixed cells is the key part of the five-equation based ALE method, and it was developed based on the analysis of variation relation of volume fraction in mixed cells with the assumption of isentropic compression. The proposed model is different from traditional five-equation model, and it is consistent with the compressibility of different fluids in mixed cells. The five-equation based ALE method can be used to simulate multi-material flows with large deformation, which is an obstacle for many traditional ALE methods. Moreover, it can be used to simulate multiphase flows. A series of multi-material and multiphase test problems were simulated with the five-equation based ALE method, and numerical results agree well exact solutions and reference results.

References

- [1] R. ABGRALL AND S. KARNI. Computations of Compressible Multifluids. *J. Comput. Phys.* **169**, 594 (2001).
- [2] B.L. TIAN, W.D. SHEN, S. JIANG, S.H. WANG AND Y. LIU. A Global Arbitrary Lagrangian-Eulerian Method for Stratified Richtmyer-Meshkov Instability. *Computer and Fluids* , doi:10.1016/j.compfluid.2011.01.022 (2011).

- [3] BAOLIN TIAN, WEIDONG SHEN, SONG JIANG, SHUANGHU WANG AND YAN LIU. An arbitrary Lagrangian-Eulerian method based on the adaptive Riemann solvers for general equations of state. *Int. J. Numer. Meth. Fluids* **59**, 1217:1240 (2009).
- [4] G. ALLAIRE, S. CLERC AND S. KOKH. A five-equation model for the simulation of interfaces between compressible fluids. *J. Comput. Phys.* **181**, 577:616 (2002).
- [5] A. MURRONE AND H. GUILLARD. A five equation reduced model for compressible two phase flow problems. *J. Comput. Phys.* **202**, 664:698 (2005).

¹Science and Technology on Computation Physics Laboratory
 Institute of Applied Physics and Computational Mathematics
 Beijing, China, 100094
 tian_baolin@iapcm.ac.cn

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Efficient deterministic modelling of three-dimensional rarefied gas flows

V.A. Titarev¹

Keywords: kinetic, S-model, unstructured, microscale, rarefied, implicit

Parallel Session: High order methods for hyperbolic conservation laws

Abstract The accurate numerical solution of the Boltzmann kinetic equation with the exact or model collision integrals is important in mathematical modelling of gaseous rarefied flows inside micro-scale systems, for which statistical methods are inefficient. Although past few years have seen rapid development of numerical methods and associated computer codes for solving this equation in complex geometries, there is still much room for improvement in terms of efficiency of time marching, spatial accuracy and parallel scalability before these methods can be used for industrial applications.

The present talk describes a recent numerical framework and associated family of implicit unstructured-mesh methods [2, 4, 3] applicable for computing rarefied gas flows in arbitrary two- and three-dimensional geometries on the basis of the Boltzmann kinetic equation with the S-model collision integral [1]. The framework circumvents the deficiencies of the existing methods and consists of the three main blocks: high-order accurate implicit advection scheme on hybrid unstructured meshes, conservative procedure for the calculation of the model collision integral and a simple and efficient implementation on modern high-performance clusters. As a result, it can be used to model rarefied problems in practical geometries with reasonable computing time.

The performance of the proposed methods is demonstrated on a number of problems of both academic and practical interests, such as gas through long planar and three-dimensional microchannels of finite length as well as flows inside micro devices. The results include estimates of accuracy, demonstration of the gains of the efficiency due to the implicit time marching as well as the analysis of parallel scalability for up to 512 cores.

References

- [1] SHAKHOV, E. Generalization of the Krook kinetic relaxation equation. *Fluid Dynamics* 3, 5 (1968), 142–145.
- [2] TITAREV, V. Numerical method for computing two-dimensional unsteady rarefied gas flows in arbitrarily shaped domains. *Computational Mathematics and Mathematical Physics* 49, 7 (2009), 1197–1211.

- [3] TITAREV, V. Implicit numerical method for computing three-dimensional rarefied gas flows using unstructured meshes. *Computational Mathematics and Mathematical Physics* 50, 10 (2010), 1719–1733.
- [4] TITAREV, V. Implicit unstructured-mesh method for calculating Poiseuille flows of rarefied gas. *Communications in Computational Physics* 8, 2 (2010), 427–444.

Mathematics Subject Classification 2000: 82D05, 76P05, 82B40, 82C40, 76M12, 65N08, 65B99

¹Dorodnicyn Computing Centre of Russian Academy of Sciences,
Vavilov st. 40, Moscow, Russia, postcode 119333
`titarev@mail.ru`

High order stochastic finite volume methods for the numerical solution of hyperbolic equations with random initial data

S.A. Tokareva, S. Mishra, C. Schwab¹,

Keywords: Hyperbolic equations, uncertainty quantification, high order numerical methods

Parallel Session: High order methods for hyperbolic conservation laws

Abstract. We construct the high order Stochastic Finite Volume (SFV) method and Global Collocation (GC) method applicable to quantify the uncertainty in hyperbolic conservation laws with random initial data. We assess the efficiency of the constructed methods for the numerical solution of the stochastic Burgers equation with two special cases of uncertainty in the solution: uncertain shock position and uncertain solution amplitude. We then perform the convergence analysis showing that both GC and SFV methods are superior to Monte Carlo type methods in the sense of error-work estimate.

Problem setting. Consider the following multidimensional hyperbolic system with uncertain initial data:

$$\frac{\partial \mathbf{U}}{\partial t} + \sum_{k=1}^N \frac{\partial \mathbf{f}_k(\mathbf{U})}{\partial x_k} = \mathbf{0}, \quad \mathbf{x} = (x_1, x_2, \dots, x_N) \in \mathbb{R}^N, t > 0; \quad (1)$$

$$\mathbf{U}(\mathbf{x}, 0, \omega) = \mathbf{U}_0(\mathbf{x}, \omega), \quad \omega \in \Omega. \quad (2)$$

Let $y_j = Y_j(\omega) \in \mathbb{R}$, $j = 1, \dots, K$, be stochastic variables; then the solution is parametrized as $\mathbf{U} = \mathbf{U}(\mathbf{x}, t, \mathbf{y})$, where $\mathbf{y} = (y_1, y_2, \dots, y_K) \in \mathbb{R}^K$ and $\rho(\mathbf{y})$ is the probability density function of a given distribution.

We compute the statistical quantities of the solution $\mathbf{U}(\mathbf{x}, t, \mathbf{y})$, such as its expectation $\mathbb{E}[\mathbf{U}]$ and variance $\mathbb{V}[\mathbf{U}]$.

Stochastic finite volume method. The SFV method to solve (1)–(2) is based on the treatment of the stochastic space as an additional deterministic space and thus constructing a two-dimensional deterministic finite-volume type method.

Consider the rectangular cell C_{ij} in the $(N + K)$ -dimensional space, where $\mathbf{i} = (i_1, i_2, \dots, i_N)$, $\mathbf{j} = (j_1, j_2, \dots, j_K)$. Define the time-dependent cell average as

$$\mathbf{U}_{ij}(t) = \frac{1}{|C_{ij}|} \int_{C_{ij}} \mathbf{U}(\mathbf{x}, t, \boldsymbol{\xi}) d\mathbf{x} d\boldsymbol{\xi}.$$

Integrating the equations (1)–(2) over the cell C_{ij} we get

$$\frac{d\mathbf{U}_{ij}(t)}{dt}|C_{ij}| + \sum_{k=1}^N \int_{C_{ij}^k} \left[\mathbf{f}_k(\mathbf{U}_h(x_{i_k+1/2})) - \mathbf{f}_k(\mathbf{U}_h(x_{i_k-1/2})) \right] dC_{ij}^k = \mathbf{0}, \quad (3)$$

where $\mathbf{U}_h$ is the approximate solution and C_{ij}^k denotes the k -th cell interface.

The resulting SFV scheme will be written as ODE system

$$\frac{d\mathbf{U}_{ij}(t)}{dt}|C_{ij}| + \sum_{k=1}^N \left[\mathbf{f}_{i_k+1/2}(t) - \mathbf{f}_{i_k-1/2}(t) \right] = \mathbf{0}. \quad (4)$$

To construct the first order SFV method it is sufficient to use the piecewise-constant solution representation, while the construction of the higher order SFV methods requires the use of the piecewise-polynomial solution reconstruction according, for example, to the ENO/WENO procedures. The quadrature rule to approximate the flux integral in (3) is chosen to guarantee the desired overall scheme order. The resulting ODE system (4) can be solved using the forward Euler or TVD Runge-Kutta methods.

Global collocation method. The idea of the GC method to solve (1)–(2) is to compute the integral statistical values, such as mean and variance, using the Gaussian quadrature rule for the integration with weight $\rho(\mathbf{y})$, namely,

$$\mathbb{E}[\mathbf{U}] = \int_{\mathbb{R}^K} \mathbf{U}(x, t, \mathbf{y}) \rho(\mathbf{y}) d\mathbf{y} \approx \sum_{j=0}^{N-1} \mathbf{U}(x, t, \mathbf{y}_j) w_j.$$

Here $\mathbf{y}_j$ denote the Gaussian integration nodes, or collocation points, and w_j are the Gaussian weights, derived for the Gaussian quadrature rule of integration with weight $\rho(\mathbf{y})$. High order ENO/WENO methods are used to solve the deterministic problem at each collocation point $\mathbf{y}_j$.

In 1d case, if y is distributed uniformly on $[-c, c] \subset \mathbb{R}$, i.e. $Y(\omega) \sim \mathcal{U}[-c, c]$, then its probability density function is: $\rho(y) = 1$, $x \in [-c, c]$ and $\rho(y) = 0$, elsewhere. The expectation of u can then be computed using the Gauss-Legendre quadrature rule. If y has a normal distribution on $\mathbb{R}$, i.e. $Y(\omega) \sim \mathcal{N}[M, \sigma]$, then its probability density function is given by $\rho(y) = \frac{1}{\sqrt{2\pi\sigma^2}} e^{-\frac{(y-M)^2}{2\sigma^2}}$. Hence, it is convenient to use the Gauss-Hermite quadrature rule to obtain the mean and the variance.

¹ETH Zurich, Seminar for Applied Mathematics
Zurich, Switzerland
svetlana.tokareva@sam.math.ethz.ch

Discontinuous-in-space explicit Runge-Kutta residual distribution schemes for time-dependent problems

A. Warzyński¹ M. E. Hubbard¹ M. Ricchiuto²

Parallel Session: High order methods for hyperbolic conservation laws.

Abstract The Residual Distribution (RD) framework for multidimensional hyperbolic conservation laws can be illustrated by considering the scalar conservation law given by

$$\nabla \cdot \mathbf{f} = 0 \tag{1}$$

on a domain Ω , with appropriate boundary conditions. The residual associated with a mesh cell E is defined to be

$$\phi_E = \int_E \nabla \cdot \mathbf{f} d\Omega, \tag{2}$$

and this is then distributed among the vertices of E . Assuming a piecewise linear representation of the approximate solution leads to the discrete system

$$\sum_{E \in \mathcal{D}_i} \beta_i^E \phi_E = 0 \quad \forall i \tag{3}$$

where the β_i^E signify the proportion of the residual in cell E assigned to node i and $\mathcal{D}_i$ denotes the subset of triangles containing i . System (3) is solved to find the approximate solution values at the mesh nodes, typically using a pseudo-time-stepping approach.

In the case of steady state problems, where f in (1) only has a spatial dependence, the residual distribution concept has already proven to be very successful. The RD approach, in a relatively natural manner, enables construction of positive, linearity preserving and conservative schemes able to carry out a truly multidimensional upwinding for both scalar and systems of hyperbolic conservation laws.

Extension to time-dependent problems is currently a subject of intensive ongoing research. It is possible to develop schemes of the form (3), as derived when the divergence in (1) includes the time variation, but solving the system (3) at each time-step is typically very cpu-intensive. To overcome this Abgrall and Ricchiuto in [1] proposed a framework for explicit, second order residual distribution schemes for transient problems. In this talk we will present their approach in conjunction with discontinuous-in-space data representation. This extends previous work on discontinuous residual distribution schemes for steady problems initiated in [2, 3]. It also extends work of Abgrall and Shu [4] in the sense that it reformulates the Runge-Kutta

Discontinuous Galerkin (DG) method in the framework of Runge-Kutta Residual Distribution schemes. This is, briefly speaking, done by considering flux differences (*edge* residuals in the RD framework) instead of the fluxes themselves.

Different types of cell- and edge-based distribution strategies can be applied and we will discuss the most interesting choices characteristic for either RD [2, 3] or DG type approaches [4, 5]. Numerical results for two-dimensional hyperbolic conservation laws on structured and unstructured triangular meshes will also be presented. A brief comparison with other recent developments in the discontinuous residual distribution framework (i.e. [6]) will also be made.

Keywords: Hyperbolic conservation laws, Time dependent problems, Explicit schemes, Residual distribution, Runge-Kutta time-stepping, Discontinuous Galerkin

Mathematics Subject Classification 2000: 65M99, 76M25

References

- [1] R. ABGRALL, M. RICCHIUTO Explicit Runge-Kutta residual distribution schemes for time dependent problems: second order case. *J. Comput. Phys.* **229**(16), 5653–5691, 2010.
- [2] M. E. HUBBARD A framework for discontinuous fluctuation distribution. *Internat. J. Numer. Methods Fluids* **56**(8), 1305–1311, 2008.
- [3] M. E. HUBBARD Discontinuous fluctuation distribution. *J. Comput. Phys.* **227**(24), 10125–10147, 2008.
- [4] R. ABGRALL, C.-W. SHU Development of Residual Distribution Schemes for the Discontinuous Galerkin Method: The Scalar Case with Linear Elements. *Commun. in Comp. Phys.* **5**(2), 376-390, 2009
- [5] B. COCKBURN, S.-C. HOU, C.-W. SHU The Runge-Kutta local projection discontinuous Galerkin finite-element method for conservation laws. 4 The Multidimensional Case. *Mathematics of Computation* **54**(190), 545-581, 1990
- [6] M. E. HUBBARD, M. RICCHIUTO Discontinuous upwind residual distribution: A route to unconditional positivity and high order accuracy. *Comput. Fluids* doi:10.1016/j.compfluid.2010.12.023, 2011

¹School of Computing
University of Leeds
Leeds, LS2 9JT, United Kingdom
scaw@leeds.ac.uk (A. Warzyński)
meh@comp.leeds.ac.uk (M. E. Hubbard)

²Inria Bordeaux - Sud-Ouest, équipe BACCHUS
351 cours de la Liberation, Bat. A29 bis
33405 Talence Cedex - France
Mario.Ricchiuto@inria.fr

Numerical Methods for Hyperbolic Equations: Theory and Applications

An international conference to honour Professor E.F. Toro
in the month of his 65th birthday

Parallel Sessions

Numerical Methods for Reactive Flows

Numerical simulation of a pulverized coal jet

A. Bermúdez¹ J.L. Ferrín² A. Liñán³ L. Saavedra⁴

Keywords: pulverized coal, group combustion, pulverized coal jet simulation, conserved scalars, char gasification

Parallel Session: Reactive Flows

Abstract: A mathematical model for the group combustion of pulverized coal particles was developed by Bermúdez *et al* in [1]. It includes the Lagrangian description of the dehumidification, devolatilization and char gasification reactions of the coal particles in the homogenized gaseous environment resulting from the three fuels, CO , H_2 and volatiles, supplied by the gasification of the particles and their simultaneous group combustion by the gas phase oxidation reactions, which are considered to be very fast. This model is complemented with an analysis of the particle dynamics, determined principally by the effects of aerodynamic drag and gravity, and its dispersion based on a stochastic model. It was also extended to include two other simpler models for the gasification of the particles: the first one for particles small enough to extinguish the surrounding diffusion flames, and a second one for particles with small ash content when the porous shell of ashes remaining after gasification of the char, non structurally stable, is disrupted.

The goal of this work is to show how these three models can be applied in the numerical simulation of group combustion. Here they are used for the analysis of a simple example of a non-swirling pulverized coal jet with a nearly stagnant air at ambient temperature, with an initial region of interaction with a small annular methane flame. The experiment that we have simulated was done in Hwang *et al* [3, 4], where a study of the structure of a turbulent pulverized coal jet flame has been carried out. Firstly, the results obtained with the three versions of the model are compared among them and subsequently we show how the first of the simpler models fits better the experimental results. Although it is shown good agreement between the experimental measurements and our simulations, it can be seen that the particle dispersion is overpredicted. This fact could be explained because the BFLs1 model does not consider changes in particle sizes, due to partial disruption or swelling effects, and also because turbulent dispersion of the particles is overestimated due to the use of the $k - \epsilon$ model and hence larger particles are more dispersed.

Concerning to the numerical methods, to solve the partial differential equations modeling the gas phase, second order modified Lagrange-Galerkin methods are used. These modified methods, analyzed in Bermejo and Saavedra [5], have the same accuracy as the standard ones but they are much more efficient.

Referencias

- [1] A. Bermúdez, J. L. Ferrín, and A. Liñán. The modelling of the generation of volatiles, H_2 and CO, and their simultaneous diffusion controlled oxidation in pulverised coal furnaces. *Combustion Theory and Modelling*, 11(6):949–976, 2006.
- [2] A. Bermúdez, J. L. Ferrín, A. Liñán, and L. Saavedra. Numerical simulation of group combustion of pulverized coal. *Accepted for publication in Combustion and Flame*, 2011.
- [3] S. Hwang, R. Kurose, F. Akamatsu, H. Tsuji, H. Makino, and M. Katsuki. Observation of detailed structure of turbulent pulverised-coal flame by optical measurement, (part 1, time-averaged measurement of behavior of pulverised-coal particles and flame structure). *JSME International Journal*, 49(4):1316–1327, 2006.
- [4] S. Hwang, R. Kurose, F. Akamatsu, H. Tsuji, H. Makino, and M. Katsuki. Observation of detailed structure of turbulent pulverised-coal flame by optical measurement, (part 2, instantaneous two-dimensional measurement of combustion reaction zone and pulverised coal particles). *JSME International Journal*, 49(4):1328–1335, 2006.
- [5] R. Bermejo and L. Saavedra. Modified Lagrange-Galerkin methods of first and second order in time for convection-diffusion problems. *Submitted to Numerische Mathematik*, 2010.

¹Departamento de Matemática Aplicada
Universidad de Santiago de Compostela
Facultad de Matemáticas. 15782 Santiago de Compostela, Spain
`alfredo.bermudez@usc.es`

²Departamento de Matemática Aplicada
Universidad de Santiago de Compostela
Facultad de Matemáticas. 15782 Santiago de Compostela, Spain
`joseluis.ferrin@usc.es`

³Departamento de Motopropulsión y Termofluidodinámica
Universidad Politécnica de Madrid
E.T.S.I. Aeronáuticos. 28040 Madrid, Spain
`amable.linan@upm.es`

⁴Departamento Fundamentos Matemáticos de la Tecnología Aeronáutica
Universidad Politécnica de Madrid
E.T.S.I. Aeronáuticos. 28040 Madrid, Spain
`laura.saavedra@upm.es`

Chapman-Jouguet conditions with temperature and velocity disequilibrium

Richard Saurel¹ Stefan Schoch² Nikolaos Nikiforakis²

Keywords: multiphase flow model, hyperbolic equations, detonation waves

Parallel Session: Reactive Flows

Abstract Generalised Chapman-Jouguet (CJ) conditions for detonation waves in heterogeneous high explosives have been recently proposed in Petitpas et al. [2]. These CJ conditions are based on the multiphase flow model of Kapila et al. [3] with heat and mass transfer (Saurel et al., [1]). Interesting results have been obtained by Baudin et al. [5] showing detonation velocities greater than the conventional CJ model. These predictions have shown perfect agreement with experimental measurements. This agreement is due to thermal disequilibrium effects between the phases, oppositely to conventional CJ relations based on the reactive Euler equations where a single temperature is present. In the present paper we consider velocity disequilibrium effects extension. This is rendered possible by the multiphase flow model of Saurel et al. [4] where velocity drift effects are restored in the Kapila et al. [2] model. A formulation involving both heat and mass transfers in a velocity disequilibrium approach is built. The model is thermodynamically consistent and its structure renders possible the determination of definitely generalised CJ conditions as well as detonation wave structure.

References

- [1] R. SAUREL, F. PETITPAS AND R. ABGRALL. Modelling phase transition in metastable liquids. Application to flashing and cavitating flows. *Journal of Fluid Mechanics* **607**, 313–350, 2008.
- [2] F. PETITPAS, R. SAUREL, E. FRANQUET AND A. CHINNAYYA. Modelling detonation waves in condensed materials: Multiphase CJ conditions and multidimensional computations. *Shock Waves* **19**(5), 377–401, 2009.
- [3] A.K. KAPILA, R. MENIKOFF, J.B. BDZIL, S.F. SON AND D.S. STEWART. Two-phase modelling of deflagration to detonation transition in granular materials: Reduced equations. *Physics of Fluids* **13**(10), 3002–3024, 2001.
- [4] R. SAUREL, N. FAVRIE, F. PETITPAS, M.H. LALLEMAND AND S. AND GAVRI-LYUK. Modelling irreversible dynamic compaction of powders. *Journal of Fluid Mechanics* **664**, 348–396, 2010.

- [5] G. BAUDIN, A. LEFRANOIS, R. SAUREL, F. PETITPAS, O. LE METAYER, J. MASSONI, V. BELSKI AND E. ZOTOV, E. Toward a thermal non-equilibrium multi-phase model for high explosive metallic particles. *Journal of Energetic Materials* **28**, 154–179, 2010.

¹SMASH Project, UMR CNRS 6595 IUSTI, Polytech Marseille, Aix-Marseille University, 5 rue E. Fermi, 13453 Marseille Cedex 13, France
`Richard.Saurel@polytech.univ-mrs.fr`

²University of Cambridge, Cavendish Laboratory, Laboratory for Scientific Computing, JJ Thomson Avenue, Cambridge CB3 0HE, UK

Multi-phase simulation of Ammonium Nitrate Emulsion Detonations

Stefan Schoch¹ Nikolaos Nikiforakis¹ and Richard Saurel²

Keywords: multiphase flow model, hyperbolic equations, detonation waves

Parallel Session: Reactive Flows

Abstract This work is concerned with the numerical simulation of detonation wave propagation in Ammonium Nitrate emulsion explosives. Due to the heterogeneous nature of the material, the behaviour of the detonation wave is highly non-ideal, exhibiting curved shock fronts and significant dependency of the detonation velocity on confiner type and charge diameter. Formulations based on the augmented Euler equations make assumptions on pressure and temperature equilibrium of the constituents in order to obtain the mixture equation of state which is required by this type of model. However it can be shown that typical heterogeneous explosives are not in temperature equilibrium even at the end of the reaction zone [1, 2]. We therefore adopt a continuum hydrodynamic representation of the material, in which each material is represented by an independent set of state variables in order to allow temperature disequilibrium. Due to the timescales of pressure and velocity relaxation in heterogeneous explosives, instantaneous relaxation can be assumed [1, 2]. Thus we adopt a single pressure and single velocity two-phase flow model, referred to as the five-equation model [1, 4]. The numerical treatment of this system follows the approach of Saurel et al [2, 3]. This model [2] is altered to allow simulations for Ammonium-Nitrate-based emulsion explosives as used in industrial applications.

The study focuses on the comparison of the detonation kinetics in a representative set of rate sticks below and above the critical diameter. The multi-dimensional unsteady reactive multiphase flow model is solved in the presence of material interfaces. The same set of equations is solved throughout the domain for pure materials and material mixtures, which occur at interfaces and within the detonation reaction zone.

The multi-dimensional results are compared to those obtained with the non-equilibrium ZND model of Petitpas et al. [2] with embedded front curvature effects. The complete non-equilibrium ZND system can be regarded as an extension of the classical quasi-one dimensional Wood-Kirkwood front curvature model.

References

- [1] A.K. KAPILA, R. MENIKOFF, J.B. BDZIL, S.F. SON AND D.S. STEWART. Two-phase modelling of deflagration to detonation transition in granular materials: Reduced equations. *Physics of Fluids* **13**(10), 3002–3024, 2001.

- [2] F. PETITPAS, R. SAUREL, E. FRANQUET AND A. CHINNAYYA. Modelling detonation waves in condensed materials: Multiphase CJ conditions and multidimensional computations. *Shock Waves* **19**(5), 377–401, 2009.
- [3] R. SAUREL, F. PETITPAS AND R.A. BERRY. Simple and efficient relaxation methods for interfaces separating compressible fluids, cavitating flows and shocks in multiphase mixtures. *J. Comput. Phys.* **228**(5), 1678–1712, 2009.
- [4] MURRONE, ANGELO AND GUILLARD, HERVÉ. A five equation reduced model for compressible two phase flow problems. *J. Comput. Phys.* **202**(2), 664–698, 2005.

¹University of Cambridge, Cavendish Laboratory, Laboratory for Scientific Computing, JJ Thomson Avenue, Cambridge CB3 0HE, UK

²SMASH Project, UMR CNRS 6595 IUSTI, Polytech Marseille, Aix-Marseille University, 5 rue E. Fermi, 13453 Marseille Cedex 13, France
Richard.Saurel@polytech.univ-mrs.fr

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

A Flux Vector Splitting Scheme for the Euler Equations

M. E. Vázquez-Cendón¹ E.F. Toro²

Keywords: Flux Vector Splitting, Euler Equations, Liou-Steffen scheme

Abstract This paper is concerned with a flux vector splitting scheme for the compressible Euler equations of gas dynamics. The scheme follows the framework set out by Liou and Steffen [1] in which the physical flux function is split into an advective part and a pressure part. The Liou-Steffen scheme is distinct from other flux vector splitting schemes in that the contact discontinuity is accounted for. This is a very important property of a numerical scheme, which results in a substantial reduction of numerical viscosity for intermediate characteristic fields. For a review of flux vector splitting schemes see chapter 8 of [2].

In this paper we propose a new way of splitting the flux. First, in the pressure flux component we include all pressure terms from the flux function, from the momentum and total energy equations. Then we consider a resulting hyperbolic subsystem whose Riemann problem solution provides the numerical flux component from the pressure flux. The resulting scheme retains the contact-capturing properties of the Liou-Steffen scheme but turns out to be more accurate and significantly more robust. We have tested the scheme for a whole suite of very severe test problems proposed in [2], with very satisfactory results. Sample applications are shown and an extension of the approach to gas dynamics with general equation of state is discussed.

Keywords: flux vector splitting, compressible Euler equations

References

- [1] M. S. LIOU AND C. J. STEFFEN. A New Flux Splitting Scheme. *Journal of Computational Physics* Vol. 107, pp 23-39, 1993.
- [2] ELEUTERIO TORO. *Riemann Solvers and Numerical Methods for Fluid Dynamics. A Practical Introduction. Third Edition.* Springer Verlag, 2009.

¹Applied Mathematics Department
University of Santiago de Compostela
Faculty of Mathematics. 15782 Santiago de Compostela, Spain
elena.vazquez.cendon@usc.es
<http://webspersoais.usc.es/persoais/elena.vazquez.cendon/>

²Laboratory of Applied Mathematics
Department of Civil and Environmental Engineering
University of Trento
38050 Mesiano di Povo, Trento (TN), ITALY toro@ing.unitn.it
<http://www.ing.unitn.it/~toroe/>

On the use of Moving Least Squares for pressure discretization in low Mach number flows

X. Nogueira¹ S. Khelladi² I. Colominas¹ F. Bakir²

Keywords: low Mach flows, Moving Least Squares

Parallel Session: High-order methods for hyperbolic conservation laws

Abstract Traditionally, two families of finite volume schemes have been developed to compute both compressible and incompressible flows. Thus, density-based solvers are used for the computation of flows when compressibility effects are important (mainly transonic, supersonic and hypersonic flows), whereas pressure-based solvers are designed to compute incompressible flows. However, it is preferable the development of solvers useful for all the regimes of a flow, this is not only for user's convenience, but also because the importance of flows where low and high Mach regions are present (for example flow past an aerodynamic profile at high angle of attack), or when compressibility effects are important, even in low Mach number flows. Thus, the modification of density or pressure-based solvers to compute all-speed flows is a current active area of research.

Godunov-like schemes are among the most widely used methods for CFD. When Godunov schemes are used in a density-based solver for low-Mach number computations, it is needed a central discretization for the pressure in order to obtain the adequate scaling of pressure with the square of the Mach number. However, the most widely used numerical schemes lead many times to non-physical solutions. Moreover many of the pressure discretization techniques used by the schemes developed for low Mach number, allow the existence of four-field solutions for the pressure (checkerboard) . In this work we present a modification in the pressure discretization of low-Mach numerical schemes. We propose using Moving-Least Squares (MLS) approximations to the discretization of the pressure flux in the numerical schemes developed for low-Mach number flows. This simple modification avoids all the problems related with checkerboard and it obtains a very accurate representation of the pressure field. The centered character of MLS approximations assures the correct scaling of the pressure with the square of the Mach number [1, 2, 3]. The high-order is achieved using MLS for the computation of the derivatives in the reconstruction step of a finite volume scheme [4]. We present here the results of the application of the proposed pressure flux discretization to the AUSM-family schemes (in particular we test the cases of the AUSM+-up [5] and SLAU [6] schemes).

References

- [1] S. SCHOCHET, Fast singular limits of hyperbolic PDEs, *Journal of Differential Equations*, **114**, 476–512, 1994.

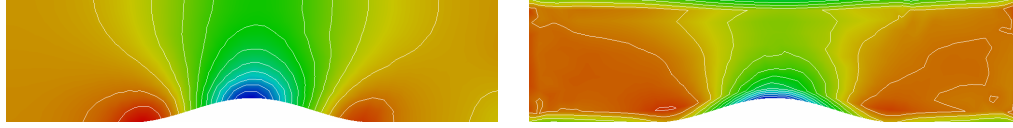


Figure 1: Pressure isolines for the $M = 10^{-2}$ flow in an unbouded 2D channel with a bump using the SLAU scheme (right) and using the SLAU scheme with MLS pressure flux computation (left).

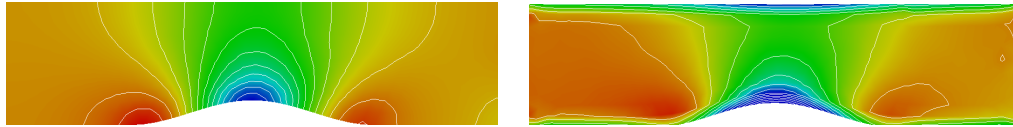


Figure 2: Pressure isolines for the $M = 10^{-2}$ flow in an unbouded 2D channel with a bump using the AUSM+-up scheme (right) and using the AUSM+-up scheme with MLS pressure flux computation (left).

- [2] H. GUILLARD, C. VIOZAT, *On the behaviour of upwind schemes in the low mach number limit*, *Computers and Fluids*, 28:63-86, 1999.
- [3] S. DELLACHERIE, Analysis of Godunov type schemes applied to the compressible Euler system at low Mach number, *Journal of Computational Physics*, **229**(4), 978–1016, 2010.
- [4] X. NOGUEIRA, L. CUETO-FELGUEROSO, I. COLOMINAS, S. KHELLADI. On the simulation of wave propagation with a higher-order finite volume scheme based on Reproducing Kernel Methods, *Computer Methods in Applied Mechanics and Engineering* **199**(23-24), 1471–1490, 2010.
- [5] M.-S. LIOU, A sequel to AUSM, Part II: AUSM+-up for all speeds, *Journal of Computational Physics*, **214**, 137–170, 2006.
- [6] E. SHIMA, K. KITAMURA, On New Simple Low-Dissipation Scheem of AUSM-Family for All Speeds, *AIAA paper*, 2010-1272, 2010.

¹Group of Numerical Methods in Engineering
 Dept. of Applied Mathematics
 Universidade da Coruña
 Campus de Elviña, 15071, A Coruña, Spain
xnogueira@udc.es
<http://caminos.udc.es/gmni/index.html>

²Arts et Métiers ParisTech
 Laboratoire de Dynamique des Fluides (DynFluid Lab)
 151 boulevard de l'Hôpital, 75013 Paris, France
sofiane.khelladi@ensam.eu

Numerical Methods for Hyperbolic Equations: Theory and Applications

An international conference to honour Professor E.F. Toro
in the month of his 65th birthday

Parallel Sessions

Shallow Water Flows

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Approximation of the Navier-Stokes system by multilayer models

E. Audusse^{1,2} M.-O. Bristeau² M. Pelanti³ J. Sainte-Marie^{2,4}

Keywords: Free surface flows, Navier-Stokes equations, variable density, multilayer models, kinetic interpretation

Mathematics Subject Classification 2000: 35Q30, 35Q35, 76D05

Parallel Session: Shallow water Flows

Abstract Due to computational issues associated with the free surface Navier-Stokes or Euler equations, the simulations of geophysical flows are often carried out with shallow water type models of reduced complexity. Thus the hydrostatic assumption for shallow water flows leads to non-linear hyperbolic systems of Saint-Venant type [1]. For these vertically averaged models, efficient and robust numerical techniques [2, 3] are available and avoid to deal with moving meshes.

Here, we are interested in flows where the horizontal velocity can hardly be approximated – as in the Saint-Venant system – by a vertically constant velocity (large bottom friction, significant water depth, wind effects, ...). We are also interested in lakes and estuarine waters, which typically exhibit a significant density stratification related to vertical variations of temperature and chemical composition. In these water bodies effects related to small density gradients may strongly affect the hydrodynamics. Density stratification processes are therefore often important in environmental flows.

To model and simulate such complex flows, multilayer Saint-Venant models are often used but the proposed models do not allow mass exchanges between neighboring layers and make a close relation to models for non-miscible fluids (see [4, 5] and the references therein). Here, we derive another and simpler multilayer model where we prescribe the vertical discretization of the layers taking into account the (unknown) total height of water. In our approach [6, 7] the layer partition is merely a discretization artefact, and it is not physical. Therefore, the internal layer boundaries do not necessarily correspond to isopycnic surfaces. A critical distinguishing feature of our model is that it allows fluid circulation between layers. This changes dramatically the properties of the model and its ability to describe flow configurations that are crucial for the foreseen applications, such as recirculation zones.

The numerical solution of the multilayer system we propose is based on a kinetic interpretation of the model and uses a finite volume kinetic scheme with an extended hydrostatic reconstruction technique [8]. In contrast with Navier-Stokes solvers, our discretization technique allows easily satisfying properties such as conservation, positivity, well-balancing of source terms (cf. [2]), and handling robustly wet/dry interfaces over variable bottom topography.

The presentation is organized as follows. We start giving the cornerstones of the derivation of the multilayer model starting from the Navier-Stokes system. Then we propose a numerical scheme for the solution of the obtained model. A special emphasis is placed on the discretization of the source terms. Finally, some numerical experiments in 3d including comparisons with analytical solutions and confrontations with experimental data are presented.

References

- [1] J.-F. Gerbeau and B. Perthame. Derivation of Viscous Saint-Venant System for Laminar Shallow Water; Numerical Validation. *Discrete Contin. Dyn. Syst. Ser. B*, 1(1):89–102, 2001.
- [2] F. Bouchut. *Nonlinear stability of finite volume methods for hyperbolic conservation laws and well-balanced schemes for sources*. Birkhäuser, 2004.
- [3] R.J. LeVeque. *Numerical methods for conservation laws*. Birkhauser, 1999.
- [4] F. Bouchut and T. Morales de Luna. An entropy satisfying scheme for two-layer shallow water equations with uncoupled treatment. *M2AN Math. Model. Numer. Anal.*, 42:683–698, 2008.
- [5] M.J. Castro, J.A. García-Rodríguez, J.M. González-Vida, J. Macías, C. Parés, and M.E. Vázquez-Cendón. Numerical simulation of two-layer shallow water flows through channels with irregular geometry. *J. Comput. Phys.*, 195(1):202–235, 2004.
- [6] E. Audusse, M.-O. Bristeau, B. Perthame, and J. Sainte-Marie. A multilayer Saint-Venant system with mass exchanges for Shallow Water flows. Derivation and numerical validation. *ESAIM: M2AN*, 45:169–200, 2011.
- [7] E. Audusse, M.-O. Bristeau, M. Pelanti, and J. Sainte-Marie. Approximation of the hydrostatic Navier-Stokes system for density stratified flows by a multilayer model. Kinetic interpretation and numerical validation. *Accepted for publication in J. Comp. Phys.*, doi: 10.1016/j.jcp.2011.01.042, 2011.
- [8] 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 J. Sci. Comput.*, 25(6):2050–2065, 2004.

¹Université Paris 13, Institut Galilée, 99 avenue Jean-Baptiste Clément,
93430 Villetaneuse, France
`audusse@math.univ-paris13.fr`

²INRIA Paris-Rocquencourt, 78153, Le Chesnay, France.
`Marie-Odile.Bristeau@inria.fr`, `Jacques.Sainte-Marie@inria.fr`

³ENSTA, Chemin de la Hunire, 91761 Palaiseau Cedex
`Marica.Pelanti@ensta.fr`

⁴Saint-Venant Laboratory, 6 quai Watier, 78401 Chatou, France.

Modeling of 3D phenomena in curved channels. Suitability of a 3D shallow waters approximation.

B. Fraga¹ L. Cea¹ E. Peña¹

Keywords: Finite Volume Method, 3D Modelling, Non-hydrostatic models, Shallow Waters

Parallel Session: Shallow Water Flows

Abstract Complex flows around in-stream structures or curved channels exhibit three dimensional features that 2-D or 1-D models cannot realistically represent. Flow in meandering or curved channels is characterized by circulation cells generated by centrifugal forces around the meander bend that move in opposite directions from one meander apex to the next. This results in a highly complex configuration, even for shallow flows.

In this work, a 3-D layer-structured finite volume model for free surface flows was applied to both a 270 bend curved channel and a meandering channel. There is in both cases a strong separation between horizontal and vertical scales, exhibiting a shallow flow behavior. 3-D unsteady momentum and mass conservation equations were solved using a Finite Volume code developed by the authors [1]. 3-D grid is collocated in the vertical coordinate adding several horizontal layers to a 2D horizontal unstructured mesh. More than 20 horizontal layers were calculated in simulations to get a good resolution of the shear stress at the bottom. 3-D Reynolds Averaged Navier-Stokes (non-hydrostatic) equations were solved, although the convenience of a shallow waters assumption will be discussed. K-eps and mixing length models of turbulence are included in the equations.

The model was tested in some study cases which exhibit important 3-D flow behavior. Results in the curved channel with a 270 bend and the meandering channel mentioned above were used to validate model's accuracy in complex flows. Comparisons with numerical and detailed experimental results were made. In both cases vertical profiles and cross sections of velocities were validated with other numerical and experimental results taken from the literature. Good agreement was found between numerical velocities and measured ones. Longitudinal velocities' distributions were accurately predicted, and have proved to be very influenced by the bed boundary condition imposed. Also transversal velocities and secondary flows patterns were simulated. Influence of turbulence modelling and bed friction is determinant for a good description of them. In steady state, simulations have revealed little dynamic pressure distributions, so that a shallow waters assumption would be suitable.

References

- [1] L. CEA, J. PUERTAS, M. ELENA VÁZQUEZ-CENDÓN. Depth Averaged Modelling of Turbulent Shallow Water Flow with Wet-Dry Fronts. *Archives of Computational Methods in Engineering* **14**(3), 303–341, 2007.
- [2] CEA L, STELLING G, ZIJLEMA M.. Non-hydrostatic 3D free surface layer-structured finite volume model for short wave propagation. *International Journal for Numerical Methods in Fluids* **81**, 382–410, 2009.
- [3] GAHMRY H. K., STEFFLER P. M.. Two-dimensional depth-averaged modeling of flow in curved open channels. *Journal of Hydraulic Research* **43**(1), 44–55, 2005.
- [4] SHIONO, K., SPOONER, J., CHAN, T., RAMESHWARAN, J., AND CHANDLER, J.. Flow Characteristics in Meandering Channels with Non-Mobile and Mobile Beds for Overbank Flows. *Journal of Hydraulic Research* **46**(1), 595–609, 2008.
- [5] THORSTEN STOESSER, NILS RUETHER AND NILS REIDAR BOE OLSEN. Calculation of primary and secondary flow and boundary shear stresses in a meandering channel. *Advances in Water Resources* **33**(2), 158–170, 2010.
- [6] TRITTHART M, GUTKNECHT D.. Three-dimensional simulation of free-surface flows using polyhedral finite volumes. *Engineering Applications of Computational Fluid Mechanics* **1**, 1–14, 2007.
- [7] PETER R. WORMLEATON, MANAYE EWUNETU. Three-dimensional k- ϵ numerical modelling of overbank flow in a mobile bed meandering channel with floodplains of different depth, roughness and platform. *Journal of Hydraulic Research* **44**(1), 18–32, 2006.

¹GEAMA, Water and Environmental Engineering Group.
Hydraulics Division.
University of A Coruña
E.T.S.I. Caminos, Canales y Puertos. 15071 Campus de Elviña, s/n.
A Coruña, Spain
`b.fraga@udc.es`

Porosity versus resolved approach in 2D shallow water models. Experimental validation.

M. Garrido¹, L. Cea¹, J. Puertas¹ and M.E. Vazquez-Cendon²

Keywords: Shallow water equations, porosity, finite volumes, experimental validation

Parallel Session: Shallow water flows

Abstract In recent years two-dimensional shallow water models have started to be used for flood inundation in urbanized areas. There are two possible ways in two-dimensional shallow water urban flood models to consider the effects of urban areas: including the exact geometry of the buildings (resolved approach) or considering only the macro-structure of the urban area (porosity approach) [2]. The resolved approach needs a more detailed geometry resolution and therefore, it provides a more detailed definition of the flow field. Hence the resolved approach requires a rather fine mesh in the proximities of the urban area. On the other hand, the porosity approach accounts in a macroscopic manner for the effects of urbanized areas, including an effective porosity parameter in the equations, as well as an additional source term which accounts for the additional drag and head losses generated in the urban region [1]. In the simulations presented in this paper rainfall-runoff in an urban area is computed with a 2D shallow water model using both, the resolved and the porosity approaches. In order to validate the numerical results and calibrate the head loss coefficient for different geometries, several experiments have been modelled with different urban configurations.

With a proper calibration of the drag coefficient, the porosity model is able to reproduce the average characteristics of the flow field outside and around the urban area, at a much lower computational cost than the classical shallow-water equations solved on a refined mesh. Obviously, the small-scale flow details inside the urban region are not reproduced by the porosity model, but the global velocity and water depth fields are very well represented. The numerical results were also compared with experimental data, obtaining encouraging results. It has been also compared the computational time and numerical stability of each approach. After applying the technique to scale models, its applicability to real scales should be investigated.

References

- [1] CEA, L., AND VÁZQUEZ-CENDÓN, M. E. Unstructured finite volume discretisation of two-dimensional depth averaged shallow water equations with porosity. *Int. J. Numer. Meth. Fluids* 63, 8 (2010), 903–930.

- [2] SOARES-FRAZAO, S., LHOMME, J., GUINOT, V., AND Y.ZECH. Two-dimensional shallow-water model with porosity for urban flood modeling. *Journal of Hydraulic Research* 46, 1 (2008), 45–64.

¹GEAMA, Water and Environmental Engineering Group.
Hydraulics Division.
University of A Corua
E.T.S.I. Caminos, Canales y Puertos. 15071 Campus de Elvia, s/n.
A Corua, Spain
`mgarrido@udc.es`, `luis.cea@udc.es`, `jpuertas@udc.es`

²Applied Mathematics Department
University of Santiago de Compostela
Faculty of Mathematics. 15782 Santiago de Compostela, Spain
`elena.vazquez.cendon@usc.es`

A WENO scheme for the integral form of contravariant shallow water equations

F. Gallerano, G. Cannata, M. Tamburrino¹

Keywords: 2D shallow water equations, Upwind WENO scheme, integral form, contravariant formulation, generalized curvilinear coordinates, Exact Riemann Solver

Parallel Session: Shallow water flows

Abstract Many authors solve shallow water equations by using Weighted Essentially Non-Oscillatory (WENO) schemes. Flow simulations over computational domains characterized by a complex boundary can be performed by numerical integrations of the contravariant form of motion equations on a generalized curvilinear boundary conforming grid.

In numerical solutions of motion equations in contravariant formulation, two contradictions appear. The first contradiction is related to the presence of Christoffel symbols in motion equations. It is well known that numerical methods for the solution of the conservation laws in which the convective terms are expressed in non-conservative form do not guarantee the convergence to weak solutions. Consequently in the integrations of the conservation laws in whose solutions shocks are present, convective terms must be expressed in conservative form. In the contravariant formulation of motion equations, covariant derivatives give rise to Christoffel symbols. These terms are extra source terms. They come in with the variability of base vectors and do not permit the definition of convective terms in a conservation form. The second contradiction is related to the difficulty of exactly numerically satisfying the metric identities and preserve the freestream conditions. A well known geometric identity is given by the condition that a cell is closed. In a curvilinear system of reference the summation condition becomes the metric identity. If numerical approximations of the metric coefficients do not exactly satisfy the above mentioned identities, the numerical approximations of derivatives of uniform physical quantities do not vanish and freestream conditions are not preserved.

The original contribution of this work is the definition of a new Upwind WENO scheme for the solution of the 2D shallow water equations expressed directly in contravariant formulation. An element of novelty presented in this paper regards the definition of a formal integral expression of the shallow water equations in contravariant formulation. The depth-integrated motion equations (in contravariant form) are integrated on an arbitrary surface and are resolved in the direction identified by a constant parallel field of unit vectors. In this way we present an integral form of the contravariant shallow-water equations in which Christoffel symbols are avoided. In order to correct the effects produced by the spurious source terms related to the difficulties of satisfying numerically the metric identities and in order to guarantee the satisfaction of the freestream preservation condition we propose an original procedure.

We define the water depth as h and the depth averaged velocity vector as $\vec{u}$, whose components are defined in the Cartesian system of reference. Let be $\vec{v} = \vec{u}h$. In order to introduce the notation to be used, we consider a transformation $x^l = x^l(\xi^1, \xi^2)$ from the Cartesian coordinates $\vec{x}$ to the curvilinear coordinates $\vec{\xi}$ (note that superscripts indicate components and not powers in the present notation). Let $\vec{g}_{(l)} = \partial\vec{x}/\partial\xi^l$ be the covariant base vectors and $\vec{g}^{(l)} = \text{grad}(\xi^l)$ the contravariant base vectors. Let r^l be the contravariant components in the curvilinear coordinate system of the vector $\vec{v}$. The integral form of the depth-integrated continuity equation is:

$$\int_{\Delta A} \frac{\partial h}{\partial t} dA + \int_L r^m n_m dL = 0, \quad (1)$$

where ΔA is an arbitrary surface element whose contour line is L , and n_m is the covariant outward normal. The integral form of the depth-integrated momentum equation is:

$$\begin{aligned} \int_{\Delta A} \vec{g}^{(l)} \cdot \vec{g}_{(k)} \frac{\partial r^k}{\partial t} dA + \int_L \vec{g}^{(l)} \cdot \vec{g}_{(k)} \left(\frac{r^k r^m}{h} + g^{mk} \frac{h^2}{2} \right) n_m dL = \\ \int_{\Delta A} \vec{g}^{(l)} \cdot \vec{g}_{(k)} \left(Ghg^{mk} \frac{\partial H}{\partial \xi^m} + R^k \right) dA, \end{aligned} \quad (2)$$

where $R^k = G(r^k |\vec{r}|)/(\chi^2 h^2)$ is the bed resistance term, $\chi = (1/M)h^{1/6}$ is the friction coefficient, M is the Manning friction factor, G is the constant of gravity, H is the bed height and $\vec{g}^{(l)}$ indicates the l^{th} contravariant base vector defined at the center of the surface element. Equations (1) and (2) represent the integral expressions of the shallow water equations in contravariant formulation, in which Christoffel symbols are not present.

The numerical integration of Equations (1) and (2) is performed by an Upwind WENO scheme. The Upwind WENO scheme needs a flux calculation at the cell interfaces. These fluxes are calculated by means of the solution of a Riemann problem. An Exact Riemann Solver is used in this framework. All necessary Riemann problems are solved in a locally valid orthonormal basis. This orthonormalization allows one to solve Cartesian Riemann problems that are devoid of geometric terms. The model is validated against several benchmark tests, and the results are compared with theoretical and alternative numerical solutions. The results of these tests show that: the proposed scheme is fifth-order accurate in space and fourth-order accurate in time; the exact C-property for quiescent flow over non-flat bottom profiles is achieved; the original procedure proposed in order to correct the effects produced by the spurious source terms related to the difficulties of satisfying numerically the metric identities, allows to satisfy the freestream preservation property even in highly distorted meshes.

Mathematics Subject Classification 2000: 76D99, 76M12

¹Department of Civil, Architectural and Environmental Engineering
University of Rome "La Sapienza"
Faculty of Engineering, Via Eudossiana 18, Rome, Italy
francesco.gallerano@uniroma1.it

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

A two-layer tidal-exchange flow.

W-K. Lee, A.G.L. Borthwick, P.H. Taylor¹

Keywords: non-conservative hyperbolic system, quadtree grid, two-layer shallow water model

Mathematics Subject Classification 2000: 65N50, 76B55, 76M12

Parallel Session: Shallow water flows

Assuming two superposed immiscible layers of shallow water fluids, the fluid mechanical behaviour of near-horizontal flows with a pronounced two-layer vertical structure can be resolved at reasonable computational cost for practical engineering purposes. Examples of two-layer flow phenomena include fresh water intruding upon denser sea water, mud flow or debris flow beneath water. Two-layer flow model can also be used to study circulation in a stratified ocean or lake, including the behaviour of internal waves or solitons. The two-layer shallow water system takes the form of a general non-homogeneous non-conservative hyperbolic system (see e.g. [1]). We extend the entropy-correction-free solver of [2, 3] for two-dimensional application using a dynamically adaptive quadtree grid. The scheme is asymptotically well-balanced and satisfies the C-property such that smooth steady solutions are up to second order accurate. Full details of the solver and the adaptive grid system can be found in [4].

We examine the dynamics of tidal effects on the behaviour of a two-layer flow in a 2D channel. Consider a channel ($x \in [-10, 10]$) with a hump at $x = 0$ and a constriction at $x = 1$. A lock exchange experiment is first conducted for two fluids with densities ratio $r = 0.98$ from water at rest. The presumed vertical barrier located at the hump is instantaneously removed at $t = 0$. The model is integrated until steady state is reached using exchange flow boundary conditions. Figure 1 shows the evolution of the interface and the grid during the initial transient.

Using the steady state solutions above as the initial conditions, we proceed to apply tidal forcing at the downstream open boundary while maintaining exchange flow boundary condition at the upstream end. Using horizontal and vertical scale of $L_h = 1 : 1000$ and $L_v = 1 : 10$ respectively, the problem thus represents a 20 km long channel with a minimum and maximum width of 2.5 km and 3.7 km respectively, and a hump of height 12 m. The quiescent water depth is 20 m. We consider an S_2 tide ($T = 60$ min) with a tidal range of 2 m where the equivalent tidal period in the model is 136.6. At the boundaries, the depth and velocities of both layers are determined using Riemann invariants specified according to the internal Froude number.

Simulation results show that when the downstream elevations are rising ($t/T = 0.5$), strong currents flow into the basin in both the layers and the flow approaches critical condition. On the other hand, when the downstream elevations are reducing ($t/T = 1.0$), the current is reversed with net flow out of the basin. The flow reaches supercritical condition resulting in a weak internal jump.

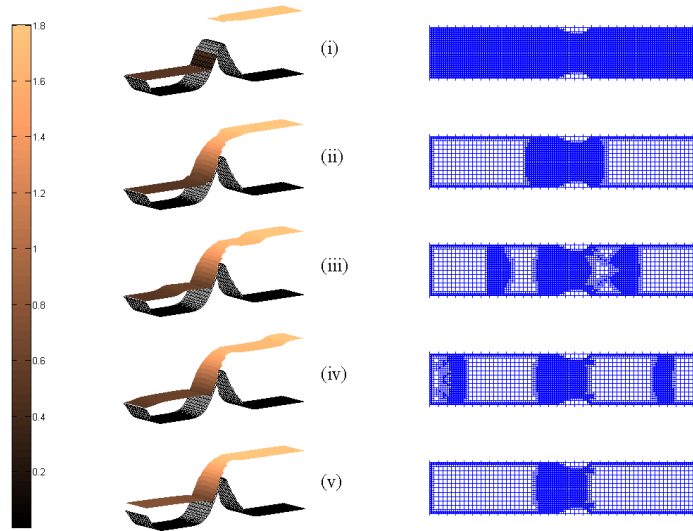


Figure 1: Lock exchange in a narrow channel with a hump and a constriction: Plots showing evolution of interface with reference to the bed (left column) and grid adaptation (right column) at times (i-v) $t = 0, 5, 10, 15, 30$.

References

- [1] DUMBSER, M., CASTRO, M., PARÉS, C. AND TORO, E. F. ADER schemes on unstructured meshes for nonconservative hyperbolic systems: Applications to geophysical flows. *Comput. Fluids*, **38**(9), 1731–1748, 2009.
- [2] CHACON-REBOLLO, T., DELGADO, A. D., AND FERNÁNDEZ-NIETO, E. D. An entropy-correction free solver for non-homogeneous shallow water equations. *Math. Modell. Numer. Anal.*, **37**(5), 755–772, 2003.
- [3] CASTRO, M. J., CHACON-REBOLLO, T., FERNÁNDEZ-NIETO, E. D., AND PARÉS, C. On well-balanced finite volume methods for nonconservative nonhomogeneous hyperbolic systems. *SIAM J. Sci. Comput.*, **29**(3), 1093–1126, 2007.
- [4] LEE, W.-K., BORTHWICK, A. G. L., AND TAYLOR, P. H. A fast adaptive quadtree scheme for a two-layer shallow water model. *J. Comput. Phys.*, submitted.

¹Department of Engineering Science
University of Oxford
Parks Road, Oxford OX1 3PJ, U.K.
wei.lee@eng.ox.ac.uk

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Numerical modelling of two-dimensional morphodynamics in gravel bed rivers.

A. Siviglia¹ G. Stecca¹ D. Vanzo¹ G. Zolezzi¹ M. Tubino¹

Keywords: Non-conservative hyperbolic systems, centred schemes, unstructured mesh, TVD Finite Volume methods, shallow water equations, sediment transport, FORCE scheme.

Parallel Session: Shallow water flows.

Abstract We study the numerical approximation of the two-dimensional morphodynamic model governed by the shallow water and Exner equations for simulating the bed evolution in gravel bed rivers. We adopt a coupled strategy solution that relies on a full coupling of the governing equations within each time step. The resulting system of governing equations contains nonconservative products. The numerical solution is obtained using a new first order accurate *centered* scheme of the finite volume type on unstructured meshes, which is an extension of the one-dimensional UFORCE scheme recently proposed in [1]. The resulting first-order accurate centered method is then extended to second order using a TVD approach.

The model is applied to well established steady solutions in order to verify the well-balanced properties. Then, it is applied to test cases with exact solutions.

A key aspect of this analysis relies on demonstrating the capability of the proposed model in capturing 2D morphodynamics features that arise in a real river context. In particular, we investigate the ability of the model in predicting free bars formations and upstream overdeepening influence comparing numerical solutions with findings from analytical theories. Finally, we investigate the ability of the model in describing the river bifurcations stability. In this case we compare numerical results with those obtained analytically and experimentally.

An attractive feature of the proposed method is that it is particularly suitable for engineering applications since it allows practitioners to adopt the most suitable sediment transport formula which better fits the field data.

References

- [1] G. STECCA, A. SIVIGLIA, E.F. TORO . Upwind-biased FORCE schemes with applications to free-surface shallow flows,. *Journal of Computational Physics* **229**(18), 6362–6380. , 2010.

¹Department of Civil and Environmental Engineering
University of Trento
Faculty of Engineering. 38122 Trento, Italy
nunzio.siviglia@ing.unitn.it
<http://sites.google.com/site/nunziosiviglia/Home>

Numerical Methods for Hyperbolic Equations: Theory and Applications

An international conference to honour Professor E.F. Toro
in the month of his 65th birthday

Poster Sessions

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Noninvasive Monitoring of Hepatic Damage from Hepatitis C Virus Infection

J. Alavez-Ramírez¹

Keywords: Hepatitis C, Hepatocytes, Viral Load, Liver Biopsy, Chronic HCV and Threshold Parameter

Poster Session: Methods and models for biomedical problems

Abstract The mathematical model for the dynamics of the hepatitis C proposed in Avendaño *et al.* (2002), with four populations (healthy and unhealthy hepatocytes, the viral load of the hepatitis C virus, and T killer cells), is revised. Showing that the reduced model obtained by considering only the first three of these populations, known as basic model, has two possible equilibrium states: the uninfected one where viruses are not present in the individual, and the endemic one where viruses and infected cells are present. A threshold parameter (the basic reproductive virus number) is introduced, and in terms of it, the global stability of both two possible equilibrium states is established. Other central result consists in showing, by model numerical simulations, the feasibility of monitoring liver damage caused by HCV, avoiding unnecessary biopsies and the undesirable related inconveniences/imponderables to the patient; another result gives a mathematical modelling basis to recently developed techniques for the disease assessment based essentially on viral load measurements.

References

- [1] J. ALAVEZ-RAMÍREZ, J.L. FUENTES-ALLEN AND J. LÓPEZ-ESTRADA. Noninvasive Monitoring of Hepatic Damage from Hepatitis C Virus Infection. *Computational and Mathematical Methods in Medicine* **2011**, doi:10.1155/2011/325470, 2011.
- [2] J. ALAVEZ-RAMÍREZ AND J. LÓPEZ-LÓPEZ. Dinámica del virus de la hepatitis C con carga viral y células T citotóxicas y monitoreo del daño hepático sin biopsias. *Revista Ingeniería y Ciencia* **6**(11), 9–33, 2010.
- [3] J. ALAVEZ-RAMÍREZ. *Estimación de Parámetros en Ecuaciones Diferenciales Ordinarias: Identificabilidad y Aplicaciones a Medicina*, Doctor in Sciences Thesis. National University of Mexico (UNAM), 2007.
- [4] J. ALAVEZ-RAMÍREZ, J. LÓPEZ-ESTRADA AND G. REYES-TERÁN. Dinámica del Virus de la Hepatitis C con Carga Viral y ALT y Monitoreo del Daño Hepático Libre de Biopsias. *Revista Ingeniería y Ciencia* **2**(4), 125–144, 2006.

- [5] R. AVENDAÑO, L. ESTEVA, J.A. FLORES, J.L. FUENTES ALLEN, G. GÓMEZ AND J. LÓPEZ-ESTRADA. A Mathematical Model for the Dynamics of Hepatitis C. *J. of Theoretical Medicine* **4**, 109–118, 2002.
- [6] L. EDSBERG AND P.-Å. WEDDIN. Numerical Tools for Parameter Estimation in ODE-Systems. *Optimization Methods and Software* **6**, 193–217, 1995.
- [7] M.G. GHANY, D.B. STRADER, D.L. THOMAS AND L.B. SEEFF. Diagnosis, Management, and Treatment of Hepatitis C: An Update. *Hepatology* **49**, 1335–1374, 2009.
- [8] A. KOROBEINIKOV. Global Properties of Basic Virus Dynamics Models. *Bull. Math. Biology* **66**, 879–883, 2004.
- [9] J.P. LA SALLE. Some Extensions of Liapunov’s Second Method. *IRE Transactions Circuit Theory* CT-7, 520–527, 1960.
- [10] A.U. NEUMANN, N.P. LAM, H. DAHARI, D.R. GRETCH, TH.E. WILEY, TH.J. LAYDEN AND A.S. PERELSON. Hepatitis C Viral Dynamics in Vivo and the Antiviral Efficacy of Interferon- α Therapy. *Science* **282**, 103–107, 1998.
- [11] M.A. NOWAK AND R.M. MAY. *Dynamics of Immune Responses*. In: *Virus Dynamics: Mathematical Principles of Immunology and Virology*. Oxford University Press, New York, 52–68, 2000.
- [12] A.S. PERELSON. Modelling Viral and Immune System Dynamics. *Nature Reviews | Immunology* **2**, 28–36, 2002.
- [13] R.M. RIBEIRO, J. LAYDEN-ALMER, K.A. POWERS, T.J. LAYDEN AND A.S. PERELSON. Dynamics of Alanine Aminotransferase During Hepatitis C Virus Treatment. *Hepatology* **38**, 296–305, 2003.
- [14] D.C. ROCKEY AND D.M. BISSELL. Noninvasive Measures of Liver Fibrosis. *Hepatology* **43**, S113–S120, 2006.

¹División Académica de Ciencias Básicas
 Universidad Juárez Autónoma de Tabasco
 Km. 1.5, Carretera Cunduacán–Jalpa de Méndez, Cunduacán 86690,
 Tabasco, México
 justino.alavez@basicas.ujat.mx

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Extension of fourth-order non-oscillatory central schemes to sediment transport equations[†]

A. Balaguer-Beser, M.T. Capilla, Ll. Gascón¹

Keywords: Central schemes, non-oscillatory, sediment transport equations

Mathematics Subject Classification 2000: 65M08, 65M12, 35Q35

Poster Session: Shallow water flows

Abstract Shallow water equations are widely used to model flows in rivers and coastal areas. For practical applications the inclusion of a non-flat bottom topography is required. For example, in order to study the river morphodynamic evolution it becomes necessary to develop numerical models that simulate the fluid flow over a movable bed. Much effort is devoted to design high-accuracy schemes to reduce the number of computational cells and minimize the computational time [2]. Upwind schemes have a characteristic-wise structure that allows to obtain a better resolution of shocks. However, they require the calculation of flows in the boundary of the cells in which the solution is integrated. Central schemes avoid this problem so the discontinuities in the pointwise solution, produced by the reconstruction algorithm, are located at the center of the staggered control volumes, allowing a simpler reconstruction of the numerical fluxes.

Balaguer-Beser [1] presents a new point value reconstruction algorithm based on average values or flux values for central RungeKutta schemes in the resolution of hyperbolic conservation laws. This reconstruction employs a fourth-order accurate approximation of point values of the solution at the two extrema and at the mid-point of each cell which are modified in order to enforce monotonicity and shape preserving properties. In this paper, we are concerned with the construction of high-order well-balanced non-oscillatory central finite volume schemes for solving the sediment transport equations. We present an extension of the central non-oscillatory Runge-Kutta scheme given in [1] to solve the shallow water equations over a movable non-flat bed described in [2].

Thus, time integration is computed following a central Runge-Kutta scheme. Spatial accuracy is obtained in each cell with the three-degree reconstruction polynomials of [1], keeping the local monotonicity of the interpolation data. We have defined a new treatment for bed slope source term which maintains the established order of accuracy and satisfies the exact conservation property (C-property). In order to do this, we have used the procedure described in Gascón and Corberán [3]. These authors propose to write the source term in divergence form so that it can be incorporated into

[†]Financial support: Universidad Politécnica de Valencia (PAID-06-10) and Ministerio de ciencia e innovación de España (Projects CGL2009-14220-C02-01, CGL2010-19591/BTE and DPI2010-20333).

the flux vector of the central finite volume scheme. Several standard one-dimensional test cases are used to verify the behaviour of our scheme and its non-oscillatory properties. We have compared our results with the ones obtained by using the source term treatment described in Caleffi et al. [2].

References

- [1] A. BALAGUER-BESER. A new reconstruction procedure in central schemes for hyperbolic conservation laws. *International journal for numerical methods in engineering* (Accepted for publication), 2011. DOI: 10.1002/nme.3105.
- [2] V. CALEFFI, A. VALIANI, A. BERNINI. High-order balanced CWENO scheme for movable bed shallow water equations *Advances in water resources* **30**, 730-741, 2007.
- [3] LL. GASCÓN AND J.M. CORBERÁN. Construction of second-order TVD schemes for nonhomogeneous hyperbolic conservation laws. *Journal of computational physics*. **172**, 261-297, 2001.

¹Departamento de matemática aplicada
 Universidad Politécnica de Valencia
 Camino de Vera s/n, 46022 Valencia, Spain
 abalague@mat.upv.es, tcapilla@mat.upv.es, llgascon@mat.upv.es
<http://cgat.webs.upv.es/presentacion/index.php>

First approach to the application of operational 4DVAR Data Assimilation to the Regional Ocean Model System.

M. Cobas-García¹ E. Vázquez-Cendón² A. Gómez-Tato¹ B. Gómez-Hombre³ J. Triñanes-Fernández⁴ C. Cotelo-Queijo¹ P. Carracedo-García³ P. Melo-Costa³ M. Ruiz-Villarreal⁵

Keywords: Data assimilation, ocean circulation models, shallow water

Poster Session: Shallow water flows

Abstract Many uncertainties are present in the application of numerical models to real-life problems: numerical uncertainties (for instance, the numerical methods involved), or physical uncertainties (e.g. bottom drag, or the parametrization of some sub-scale phenomena, like turbulence, etc.), among others. As a consequence of these uncertainties, models initialized with accurate initial conditions will eventually drift away from reality at some point.

This is actually the case for models that simulate the motion of earth fluids, like the Regional Ocean Model System (ROMS). The ROMS model is a state-of-the-art, three dimensional, free-surface ocean model, that solves the *Naver-Stokes equations*, combined with the *conservation of mass*, a *state equation* and boundary and initial conditions. A wide community of users (www.myroms.org) is involved in its development, and it is being used for a variety of studies, from ecosystem [1] to circulation dynamics studies [2].

To prevent (or minimise) this drift, several data assimilation algorithms are used to merge the model results with observations. The ROMS model incorporates the 4DVAR data assimilation algorithm, that consists in the optimization of a quadratic function along multiple dimensions, to find the initial state that minimises the distance to the observations, taking into account the spatial correlation scales (scs) of the variables being corrected. This algorithm will be used within the framework of the OCEANO project, as part of an operational run of the model.

We present in this contribution promising preliminary results of the application of the method. Several experiments were carried out as follows: Firstly, a "reference" simulation was run, from which we extracted initial conditions (i.c.) to feed the model, and some data that will play the role of observations to be assimilated. This way, the evolution of these i.c. would match exactly the observations extracted. Then, the i.c. were perturbed in different ways, so that there would be a misfit between the observations and the evolution of the perturbed i.c. Finally, the observations were assimilated into the perturbed i.c., and new corrected i.c. were generated by the algorithm. This way we can assess the resemblance of the corrected and the

unperturbed initial conditions, or, in other words, we can compare the initial and final perturbation fields, and therefore estimate the efficiency of the algorithm.

As the data usually available for operational assimilation consists mainly in satellite data of both temperature (SST) and sea surface height (SSH) and, to a minor extent, buoy and/or mooring stations data, the assimilation procedure tried to resemble this scenario, and therefore, data of SST, SSH and some salinity stations were assimilated. The results so far show a diversity of degrees of correction of the perturbed fields, with good average results. Owing to the high correlation between temperature and salinity, it was found that although scarce salinity data were provided, the salinity correction was to a wide extent of the same order of magnitude as the temperature correction. The 4DVAR algorithm needs to be tuned to the expected spatial correlation scale (scs) of the perturbation, in order to optimize the corrections, as can be seen in the following table. Initially, the scs was set to 0,58 degrees (lat/lon) in the 4DVAR algorithm, and the initial conditions were perturbed with noises with different scs. The percentage of the corrections decreases as the mismatch between the noise and algorithm scs. increases. Finally, the 4DVAR algorithm was re-set for the worse of the different scs (0,15 degrees, row 0.15_{rs} in the table), and the average correction ratios were recovered.

Perturb. scs	Avg Temp. Perturb. Before	Avg Temp. Perturb. After	%Red.
1.00	0.243	0.114	0.53
0.58	0.168	0.085	0.50
0.30	0.169	0.085	0.50
0.15	0.126	0.083	0.44
0.15 _{rs}	0.126	0.063	0.50

ACKNOWLEDGEMENTS: Océano project, founded by Xunta de Galicia, code number 09MDS009CT.

References

- [1] A. PELIZ, P. MARCHESIELLO, J. DUBERT, M. MARTA-ALMEIDA, C. ROY AND H. QUEIROGA. A study of crab larvae dispersal on the Western Iberian Shelf: Physical processes. *Journal of Marine Systems* **68**(1-2), volume 68, 215236, 2007.
- [2] P. OTERO, M. RUIZ-VILLARREAL, AND A. PELIZ. Variability of river plumes off Northwest Iberia in response to wind events. *Journal of Marine Systems* **72**, 238255, 2008.

¹Centro de Supercomputación de Galicia
Avda. Vigo, s/n. 15705 Santiago de Compostela, Galicia, Spain
mcobas@cesga.es

²Applied Mathematics Department - University of Santiago de Compostela.

³MeteoGalicia - Xunta de Galicia

⁴Systems Laboratory - University of Santiago de Compostela

⁵Centro Oceanográfico de A Coruña - Instituto Español de Oceanografía.

Vascular network hemodynamics: Application to the study of Arteriovenous Malformations

J.R. Duque¹ D. Gómez-Ullate¹ M. A. Herrero² L. Núñez³ P. Suárez¹

Keywords: Arteriovenous malformation, Anatomical variations, One-dimensional modelling, Pulse wave propagation

Poster Session: Methods and models for biomedical problems

Abstract Blood flow in arbitrary vascular networks is modelled in order to explain the behaviour of an arteriovenous malformation before and after treatment. Arteriovenous malformations (AVMs) are anomalies of the vascular tissue, and consist of a conglomerate of tortuous and abnormal vessels that shunt blood from the arterial to the venous system with a lack of intervening capillary bed. The deficient muscular layer of the arteries and the high arterial and venous pressures make them susceptible to hemorrhage. The treatment of an AVM includes surgical excision of the nidus

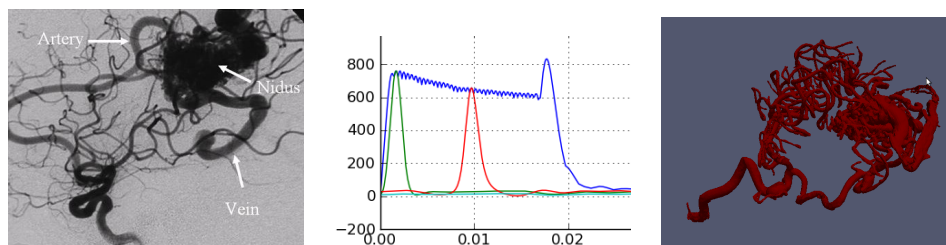


Figure 1: **a.** AVM image obtained through angiography showing the nidus, feeding arteries and draining veins. **b.** Numerical simulation of the pressure pulse propagation in a vessel. **c.** Computer reconstruction of an AVM medical image.

(whenever possible), embolization or radiotherapy. This last treatment is prescribed in cases where a surgical intervention is not possible and its aim is to achieve a gradual obliteration of the nidus while reducing the risk of rupture.

The aim of our work is to simulate the blood flow in an AVM under normal and pathological conditions, including the effect of the different treatments (embolization, partial occlusion of vessels and effect of radiotherapy) and predicting the risk of hemorrhage.

We model the blood flow using a 1D model for pressure and flow wave propagation in compliant vessels, that is able to capture the main features of pulse wave propagation throughout the cerebral arteries [1]. The equations are a 2x2 hyperbolic system of partial differential equations of conservations laws whose unknowns are the

cross section area and flow. Area is related to pressure by a constitutive vessel law which describes the fluid-wall interaction. The equations are solved numerically using a variant of the Lax-Wendroff finite difference scheme.

In a subsequent extension of the project we plan to model the effects of radiotherapy on the mechanical properties of the vascular tissue. Radiation is known to induce a fibrosis but the modification of the properties as a function of time and dosis have not been described quantitatively. Some experiments are under progress to complement the model with experimental data.

References

- [1] ALFIO QUARTERONI AND LUCA FORMAGGIA. Mathematical modelling and numerical simulation of the cardiovascular system. *Modelling of Living Systems*, Handbook of Numerical Analysis Series, Elsevier, 2002.
- [2] DANIEL LAMPONI. One dimensional and multiscale models for blood flow circulation. *These n. 3006* 2004.
- [3] Stereotactic Radiosurgery for Patients with Intracranial Arteriovenous Malformations (AVM). Radiosurgery Practice Guideline Report 2-03, March 2009. *IRSA*
- [4] TARIK F. MASSOUD ET AL. Experimental Radiosurgery Simulations Using a Theoretical Model of Cerebral Arteriovenous Malformations. *Stroke* **31**, 2466-2477, 2000.

¹Department of Theoretical Physics II
 Universidad Complutense de Madrid
 Faculty of Physics, Madrid, 28040, Spain
david.gomez-ullate@fis.ucm.es

²IMI and Department of Applied Mathematics
 Universidad Complutense de Madrid
 Faculty of Physics, Madrid, 28040, Spain
herrero@mat.ucm.es

³Servicio de Radiofísica y Protección Radiológica
 Hospital Universitario Puerta de Hierro Majadahonda
 C/ Manuel de Falla, 1, Majadahonda, 28222, Madrid
lnunez.hp@salud.madrid.org

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Changes in buoyancy-driven instabilities by using a reaction-diffusion system

Darío Martín Escala, Jacobo Guiu-Souto, Jorge Carballido-Landera,
Alberto Pérez Muñuzuri¹, Elena Vázquez-Cendón²

Keywords: Fluids Instabilities, Reactive Flows, Chemical Oscillators

Minisymposia: Multiphase flows and porous media

Abstract At the interface generated in the mixing of miscible fluids, instabilities can be displayed by the difference between the fluids densities and diffusion coefficients. These instabilities generate characteristic patterns that affect the mass transport between the two species.

BZ reaction (Belousov-Zhabotinsky) is a chemical reaction where, due to the autocatalysis of its intermediaries and the difference between diffusion coefficients of the same, are generated chemical oscillations and waves that result in pattern formation when the reaction is carried out in two-dimensional media.

The aim of this study is to analyze the influence of reaction diffusion on the instabilities caused by the contact of two fluids of different density and diffusion coefficient.

The mathematical models involved in these phenomena are solved using numerical methods such as finite differences or finite volumes.

References

- [1] T. Bansagi, Jr., D. Horvath, A. Toth, J. Yang, S. Kalliadasis, and A. De Wit. On The Classification Of Buoyancy-Driven Chemo-Hydrodynamic Instabilities Of Chemical fronts. *Physical Review. E* **68**, 055301(R) (2003)
- [2] S. Kalliadasis, J. Yang and A. De Wit, Fingering Instabilities In Exothermic Reaction-Diffusion Fronts In Porous Media, *Physics of Fluids*. **16**, 1395-1409 (2004)
- [3] P.M.J. Trevelyan, C. Almarcha and A. De Wit, Buoyancy-Driven Instabilities Of Miscible Two-Layer Stratifications In Porous Media And Hele-Shaw Cells, *Journal of Fluid Mechanics*. **670**, 38-65, (2011)

¹Non Linear Physics Group
University of Santiago de Compostela
Faculty of Physics. 15782 Santiago de Compostela, Spain
dmescala@gmail.com
chaos.usc.es

²Applied Mathematics Department
University of Santiago de Compostela
Faculty of Mathematics. 15782 Santiago de Compostela, Spain
elena.vazquez.cendon@usc.es
<http://webspersoais.usc.es/persoais/elena.vazquez.cendon/>

A Mean Gradient Method to Solve Shallow Flows.

J. Fe¹, F. Navarrina¹ & L. Cueto-Felgueroso²

Keywords: Shallow water equations, Turbulent term, Numerical viscosity, Gradient mean values

Poster Session: Shallow water flows

Abstract The two-dimensional shallow water equations (2D-SWE) form a hyperbolic system of non linear conservation laws. They are derived from the Navier-Stokes equations through some simplifying hypothesis, the most important of which is the hydrostatic pressure distribution [1]. This system describes the behaviour of incompressible fluids when the ratio of the depth to the horizontal dimensions is small, situation that can be found, for instance, in the flow in channels and rivers.

2D-SWE take into account the turbulence effects both through the frictional terms and the second derivatives term. This last term may not be significant when we only need an estimate of energy losses and 2D-SWE are frequently used without considering it. However, in the simulation of flows in which recirculation zones play an important role, the inclusion of this term may become critical.

To discretize the hydrodynamic equations, our first choice was to implement the first order finite volume method proposed by Bermudez et al. [2], which assumes that the variables values are constant at both sides of every cell edge and at every time step. An important point in this method is to properly calculate the numerical flux at the cell edges. The upwinding of the flux term has proved to be a useful stabilizing technique, but it also produces a considerable amount of numerical diffusion. For this reason we have developed a method second order accurate in space. It makes use of the mean gradient of the variables in a cell, which is calculated from their values at the cell edges, that in turn depend on the values at the adjacent cells. In this way, to calculate the values at the edges of a cell, the considered cell and the ones that surround it are involved.

To test this technique, the Cavity Flow problem has been solved in both homogeneous and non homogeneous meshes, the latter with a finer grid near the corners. Calculations have been made for viscosity values corresponding to Reynolds numbers up to 20000. The computational meshes have 81 x 81 nodes. In Figure 1 we represent the resulting streamlines obtained with Reynolds numbers of 1000, 5000 and 10000 by using the second order method in a non homogeneous mesh.

In a subsequent stage, to deal with practical problems, we needed to calculate the turbulent viscosity at every point, therefore a turbulence model was needed. To this aim, the depth-averaged $k - \varepsilon$ model, described in [3] was chosen for our work. Calculations were made for a domain consisting in a channel that supplies water to two shallow ponds. The obtained results by using both first and second order models

were compared with experimental measurements, showing the dramatic improvement achieved with the second order model.

In this work: 1) we describe the proposed second-order-in-space method; 2) we show the results of using the model in the Cavity Flow problem, for uniform values of the viscosity; 3) we apply the model to a real case, using the turbulent viscosity values obtained from the $k-\varepsilon$ model, and we compare the computational results with experimental measurements.

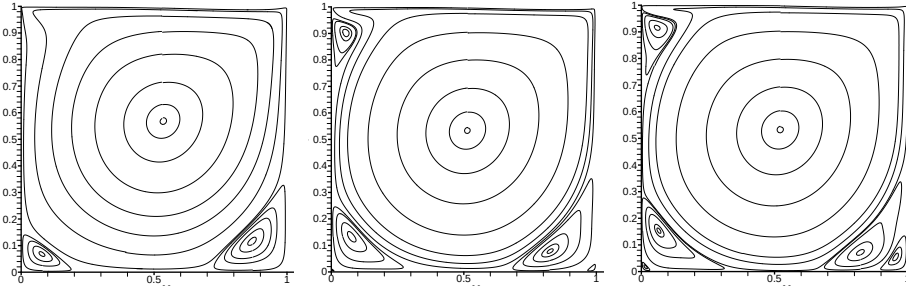


Figure 1: Streamlines (left to right): $Re = 1000, 5000, 10000$.

References

- [1] C. B. VREUGDENHIL. Numerical methods for shallow-water flow. *Kluwer Academic Publishers*, Dordrecht, The Netherlands, 1994.
- [2] A. BERMÚDEZ, A. DERVIEUX, J. DESIDERI & M. ELENA VÁZQUEZ-CENDÓN. Depth Averaged Modelling of Turbulent Shallow Water Flow with Wet-Dry Fronts. *Computer Methods in Applied Mechanics and Engineering* **155**, 49–72, 1998.
- [3] W. RODI. Turbulence models and their application in hydraulics. A state-of-the-art review. *IAHR monograph*, 3rd. Edition, Balkema, Rotterdam, 1993.

¹Grupo de Métodos Numéricos en Ingeniería
University of A Coruña
Campus de Elviña. 15071 A Coruña, Spain
jfe@udc.es
fermin.navarrina@udc.es
<http://caminos.udc.es/gmni>

²Aerospace Computational Design Laboratory
Massachusetts Institute of Technology
77 Massachusetts Ave, Cambridge, MA 02139 USA
lcueto@mit.edu

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Upwind and central schemes for nonhomogeneous hyperbolic conservation laws recognizing stationary solutions[†]

Ll. Gascón, A. Balaguer-Beser, M.T. Capilla¹

Keywords: Upwind schemes, central schemes, well balanced, nonhomogeneous conservation laws

Mathematics Subject Classification 2000: 65M08, 65M12, 35Q35

Poster Session: Shallow water flows

Abstract The present research is concerned with the construction of upwind and central high order numerical schemes for solving nonhomogeneous hyperbolic conservation laws balancing source and flux terms.

In a previous work, Balaguer and Conde [1] describe a fourth-order non-oscillatory scheme for solving hyperbolic conservation laws, which is based on a new reconstruction procedure that may be applied to upwind and central schemes. In this method, point values of the solution are calculated from the cell averages by avoiding the increase in the number of solution extrema at the interior of each cell. In order to do this, once the cell-averages of the solution are known in cells I_j , a new three-degree reconstruction polynomial is defined so that, in each cell I_j , it presents the same shape as the cell averages at cells $I_k, k \in \{j-1, j, j+1\}$. A fourth-order scheme that satisfies the total variation bounded (TVB) property is obtained by combining this reconstruction with a nonoscillatory property and a maximum principle requirement. Several standard one-dimensional test cases are used to verify the behaviour of this scheme and its non-oscillatory properties. Extension to systems is carried out in the central schemes by componentwise application of the scalar framework.

In [3] Gascón and Corberán describe a strategy which provides a suitable technique to extend well-known schemes for the homogeneous case to nonhomogeneous conservation laws, guaranteeing the balance of the flux and source terms at steady states of a natural way. The technique is based on the transformation of the nonhomogeneous problem to homogeneous form through the definition of a new flux formed by the physical flux and the primitive of the source term. Additionally, this transformation allows us to include correctly the source terms as a divergence term providing the balance between flux and source terms of a natural way. For this, the formulation must be such that all the differences that appear in the schemes are expressed as flux-differences including source terms and not as flow variable differences. This strategy

[†]Financial support: Universidad Politécnica de Valencia (PAID-06-10) and Spanish Ministry of Science and Innovation (Projects CGL2009-14220-C02-01, CGL2010-19591/BTE and DPI2010-20333).

has been applied to extend the second-order non-oscillatory schemes of Harten [3] and the flux limiters of Sweby to the nonhomogeneous case in [4] and [5]. In [2] Caselles et al. extend the Marquina's flux formula in upwind schemes to the shallow water system with source terms coming from a non-flat topography, using the mentioned transformation. Numerical results in the context of the quasi-one-dimensional flow and the shallow water equations validate the capacity of the technique suggested to construct schemes that capture non-homogeneous stationary solutions. In all the cases, the source term has been discretized in an upwind manner similar to the numerical flux.

In this paper, we present a revision of the procedure described in [3] in order to adapt it to obtain an extension of the numerical schemes developed in [1] for nonhomogeneous hyperbolic conservation laws. We apply the schemes to solve the shallow water system in some problems. We show that the proposed treatment for bed slope source term maintains the established order of accuracy and satisfies the exact conservation property (C-property). A comparison of numerical results obtained with the adapted upwind schemes described in [3] and [5] and the extension to the central fourth-order non-oscillatory scheme proposed in [1] will be presented.

References

- [1] A. BALAGUER AND C. CONDE. Fourth-order non-oscillatory upwind and central schemes for hyperbolic conservation laws. *SIAM Journal on Numerical Analysis*. **43**, 455-473, 2005.
- [2] V. CASELLES, R. DONAT, G. HARO. Flux-gradient and source term balancing of certain high resolution shock-capturing schemes. *Computers and fluids* **38**, 16-36, 2009.
- [3] LL. GASCÓN AND J.M. CORBERÁN. Construction of second-order TVD schemes for nonhomogeneous hyperbolic conservation laws. *Journal of computational physics*. **172**, 261-297, 2001.
- [4] LL. GASCÓN, J.M. CORBERÁN, J.A. GARCÍA. Esquemas de Alta Resolución Aplicados a Sistemas Hiperbólicos no Homogéneos con Soluciones Estacionarias. *Actas del XVII CEDYA*, 1-8 (CD-ROM), Salamanca, 2001.
- [5] LL. GASCÓN, J.M. CORBERÁN, J.A. GARCÍA. Construction of High-order Schemes for Hyperbolic Equations involving Source Terms. *New and Advanced Numerical Strategies in Forming Processes Simulation (6th International ESAFORM Conf.)* **I**, 683-686, Salerno (Italy), 2003.

¹Departamento de Matemática Aplicada
 Universidad Politécnica de Valencia
 Camino de Vera s/n, 46022 Valencia, Spain
 llgascon@mat.upv.es, abalague@mat.upv.es, tcapilla@mat.upv.es
<http://cgat.webs.upv.es/presentacion/index.php>

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Miscible Viscous Fingering on Reactive Interface

Jacobo Guiu-Souto, Darío Martín Escala, Jorge Carballido-Landeira,
Alberto Pérez Muñuzuri¹

Keywords: Fluids Instabilities, Viscous Fingering, Reaction-Diffusion System

Minisymposia: Multiphase flows and porous media

Abstract In living organisms, fingering structures are often observed in open systems such as growing bacterial colonies. Viscous fingering phenomenon classically occurs when a less viscous fluid displaces a more viscous one in a porous media. We investigate numerically how the instability can be triggered by a simple reaction-diffusion flow. In order to perform our simulations we employed two solutions with different viscosities reacting in the interface and generating a more viscous product. The properties of the fingering pattern observed in the region where the less viscous reactant pushes the more viscous product are studied as a function of the relevant parameters of the problem.

References

- [1] I. R. Tokita, T. Katoh, Y. Maeda, J. Wakita, M. Sano, T. Matsuyama, and M. Matsushita Pattern Formation of Bacterial Colonies by Escherichia coli Journal of Physics Society of Japan. 78 074005 (2009)
- [2] C. T. Tan and G. M. Homsy, Simulation of nonlinear viscous fingering in miscible displacement Physics Fluids 31, 1330 1988.
- [3] T. Grard and A. De Wit Miscible viscous fingering induced by a simple $A+B \rightarrow C$ chemical reaction physical review E 79, 016308 (2009)

¹Non Linear Physics Group
University of Santiago de Compostela
Faculty of Physics. 15782 Santiago de Compostela, Spain
`jacobo.guiu@usc.es`
`chaos.usc.es`

Numerical Methods for Hyperbolic Equations: Theory and Applications. An international conference to honour Professor E.F. Toro
University of Santiago de Compostela, 4-8 July 2011, Spain

Discontinuous Galerkin Methods for Vlasov-Poisson system

S. Hajian¹

Poster Session: Numerical methods in astrophysics, nuclear fusion and plasma physics

Abstract One of the simplest model problems in the kinetic theory of plasma-physics is the Vlasov-Poisson (VP) system with periodic boundary conditions. Such system describes the evolution of a plasma of charged particles (electrons and ions) under the effects of the transport and self-consistent electric field Φ_x . Denoting by f the distribution of particles, the Vlasov-Poisson system reads:

$$\begin{cases} \frac{\partial f}{\partial t} + v \frac{\partial f}{\partial x} - \Phi_x \frac{\partial f}{\partial v} = 0 & (x, v, t) \in \Omega_x \times \mathbb{R} \times [0, \infty) \\ -\Phi_{xx} = \rho(x, t) - 1 & (x, t) \in \Omega_x \times [0, \infty) \end{cases}$$

We consider a family of semi-discrete numerical schemes for the approximation of the Vlasov-Poisson system. The methods are based on the coupling of discontinuous Galerkin (DG) approximation to the Vlasov equation and several finite element (conforming, non-conforming and mixed) approximations for the Poisson problem. We investigate the numerical performance of all the schemes in challenging questions such as the Landau damping and two stream instability. We study and validate the conservation of physical properties such as L_p -norms, mass and total energy.

¹MathMods program,
Universitat Autònoma de Barcelona &
CRM (centre de Recerca Matemàtica)
`soheil.hajian@mathmods.eu`



This volume contains the abstracts of the papers presented at the International Conference on *Numerical Methods for Hyperbolic Equations: Theory and Applications* held in the Faculty of Mathematics of the University of Santiago de Compostela, Spain, from 4th to 8th July 2011. The conference was organized to honour Professor Toro in the month of his 65th birthday. We think that all contributions are a valuable state of the art of the most recent research in the topic of numerical methods for hyperbolic equations providing the reader with the latest developments concerning the mathematical aspects and the applications of this active field of mathematics.

