Activity Report 2011

Team CAGIRE

Computational Approximation with discontinous Galerkin methods and compaRison with Experiments

RESEARCH CENTER
Bordeaux - Sud-Ouest

THEME
Computational models and simulation

Table of contents

Team CAGIRE

Keywords: Fluid Dynamics, Direct Numerical Simulation, Finite Elements, Experiments, Internal Aerodynamic, Numerical Methods, Simulation Tool

This project is a common project with CNRS and University of Pau and Pays de l'Adour (UPPA). The team has been created on June 1st, 2011 and is located at UPPA premises in Pau.

1. Members

Research Scientists

Pascal Bruel [Team leader, Junior Researcher CNRS, HdR]
Vincent Perrier [Junior Researcher INRIA]

Faculty Members

Erwin Franquet [Associate Professor, Université de Pau et des Pays de l'Adour]
Tarik Kousksou [Associate Professor, Université de Pau et des Pays de l'Adour, HdR]

Technical Staff

Maxime Mogé [Since November 1, 2011]

Administrative Assistant

Josy Baron [shared with another team]

2. Overall Objectives

2.1. Overall Objectives

This project aims at studying a particular sort of jet that is often encountered in internal aerodynamic: the jets in crossflow (see Figure 1-top). The originality of this project stems from the simultaneous and strongly coupled experimental and numerical studies of such jets.

From an experimental point of view, the test facility MAVERIC [1] built three years ago at LMA and its metrology is used. An overview of this test rig is presented in Figure 1-bottom. This test facility is able to produce the kind of flow depicted in Figure 1-top. The configuration of an isolated jet in a turbulent crossflow will be carefully investigated to produce high quality data (mainly related to the velocity field properties). One-component laser Doppler velocimetry (LDV) as well as particle image velocimetry (PIV) are the two workhorses to be used in order to experimentally characterize the flowfield.

A close interaction during the course of the project between experiments and simulation will be established. From the simulation point of view, the aim is to be able to perform within two years a direct numerical simulation of an isothermal configuration of an inclined jet in crossflow in turbulence conditions, with a compressible solver that must be still accurate at low Mach number. Considering the challenge that such a task represents, a collaboration has been established with the Bacchus team in order to avoid too many useless redundancies. The Cagire team shares with Bacchus a common framework of development in which both common and team specific tools are being elaborated. From a numerical point of view, the challenge stems from the recourse to hybrid unstructured meshes, which is mandatory for our flow configuration, and implicit time integration, which is induced by the low Mach number of the flow. From the point of view of the interaction between experiments and CFD, the challenge will be mostly related to the capability of ensuring that the flow simulated and the flow experimentally investigated are as identical as possible.

[1] MAquette pour la Validation et l'Expérimentation sur le Refroidissement par Injection Contrôlée

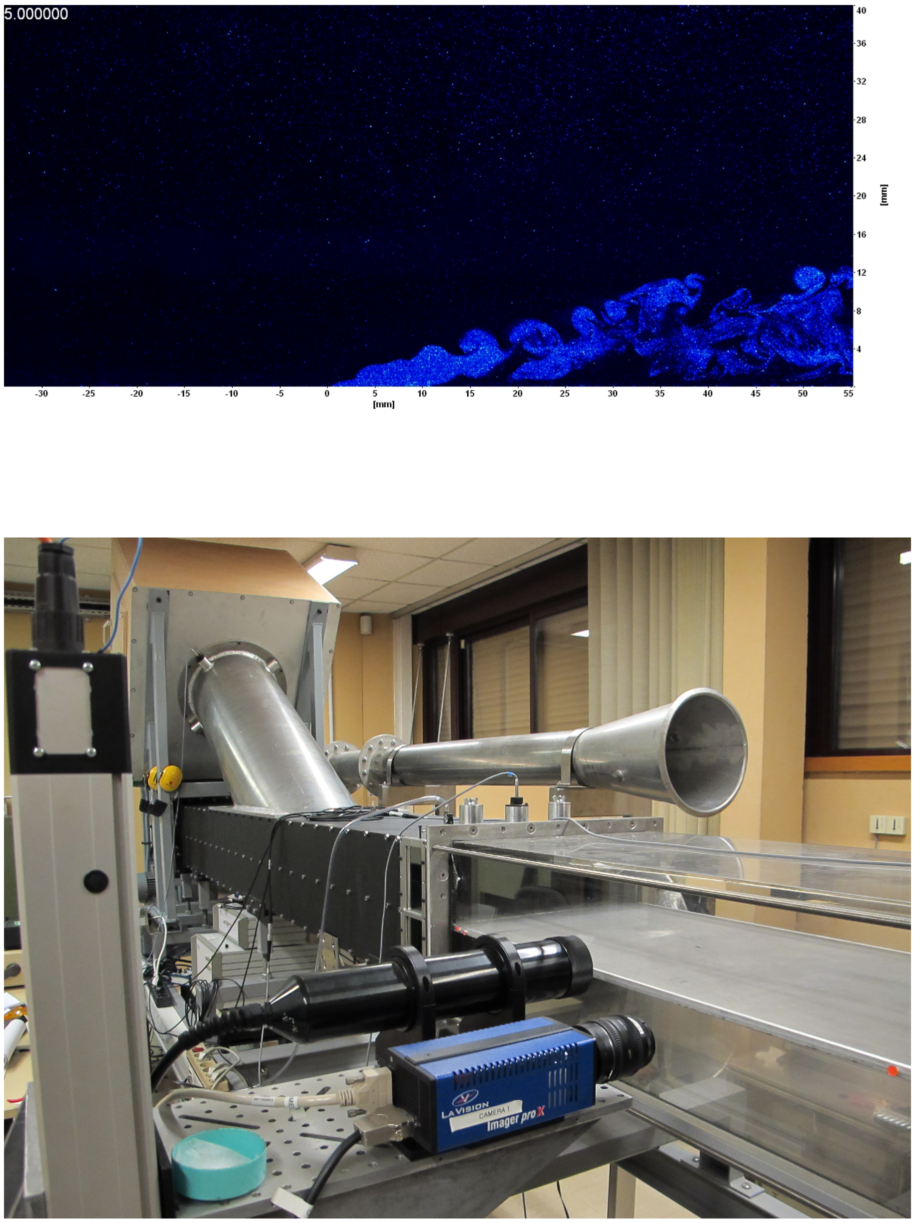

Figure 1. MAVERIC test facility: visualization of a single jet in crossflow (top) and overview of the test rig (bottom).

3. Scientific Foundations

3.1. Computational fluid mechanics: resolving versus modelling small scales of turbulence

A typical continuous solution of the Navier Stokes equations is governed by a spectrum of time and space scales. The broadness of that spectrum is directly controlled by the Reynolds number defined as the ratio between the inertial forces and the viscous forces. This number is quite helpful to determine if the flow is turbulent or not. In the former case, it indicates the range of scales of fluctuations that are present in the flow under study. Typically, for instance for the velocity field, the ratio between the largest scale (the integral length scale) to the smallest one (Kolmogorov scale) scales as $Re^{3/4}$. The smallest scales may have a certain effect on the largest ones which implies that an accurate framework for the computation of flows must take into account all these scales. This can be achieved either by solving directly the Navier-Stokes equations (Direct numerical simulations or DNS) or by first applying a time filtering (Reynolds Average Navier-Stokes or RANS) or a spatial filtering operator to the Navier-Stokes equations (large-eddy simulations or LES). The new terms brought about by the filtering operator have to be modelled. From a computational point of view, the RANS approach is the less demanding, which explains why historically it has been the workhorse in both the academic and the industrial sectors. Although it has permitted quite a substantive progress in the understanding of various phenomena such as turbulent combustion or heat transfer, its inability to provide a time-dependent information has led to promote in the last decade the recourse to either LES or DNS. By simulating the large scale structures while modelling the smallest ones supposed to be more isotropic, LES proved to be quite a step through that permits to fully take advantage of the increasing power of computers to study complex flow configurations. In the same time, DNS was progressively applied to geometries of increasing complexity (channel flows, jets, turbulent premixed flames), and proved to be a formidable tool that permits **(i)** to improve our knowledge of turbulent flows and **(ii)** to test (i.e. validate or invalidate) and improve the numerous modelling hypotheses inherently associated to the RANS and LES approaches. From a numerical point of view, if the steady nature of the RANS equations allows to perform iterative convergence on finer and finer meshes, this is no longer possible for LES or DNS which are time-dependent. It is therefore necessary to develop high accuracy schemes in such frameworks. Considering that the Reynolds number in an engine combustion chamber is significantly larger than 10000, a direct numerical simulation of the whole flow domain is not conceivable on a routine basis but the simulation of generic flows which feature some of the phenomena present in a combustion chamber is accessible considering the recent progresses in High Performance Computing (HPC). Along these lines, our objective is to develop a DNS tool to simulate a jet in crossflow configuration which is the generic flow of an aeronautical combustion chamber as far as its effusion cooling is concerned.

3.2. Computational fluid mechanics: numerical methods

All the methods we describe are mesh-based methods: the computational domain is divided into *cells*, that have an elementary shape: triangle and quadrangle in two dimensions, and tetrahedra, hexahedra, pyramids, and prism in three dimensions. If the cells are only regular hexahedra, the mesh is said to be *structured*. Otherwise, it is said to be unstructured. If the mesh is composed of more than one sort of elementary shape, the mesh is said to be *hybrid*.

The basic numerical model for the computation of internal flows is based on the Navier-Stokes equations. For fifty years, many sorts of numerical approximation have been tried for this sort of system: finite differences, finite volumes, and finite elements.

The finite differences have met a great success for some equations, but for the approximation of fluid mechanics, they suffer from two drawbacks. First, structured meshes must be used. This drawback can be very limiting in the context of internal aerodynamics, in which the geometries can be very complex. The other problem is that finite difference schemes do not include any upwinding process, which is essential for convection dominated flows.

The finite volumes methods have imposed themselves in the last thirty years in the context of aerodynamic. They intrinsically contain an upwinding mechanism, so that they are naturally stable for linear as much as for nonlinear convective flows. The extension to diffusive flows has been done in [15]. Whereas the extension to second order with the MUSCL method is widely spread, the extension to higher order has always been a strong drawback of finite volumes methods. For such an extension, reconstruction methods have been developed (ENO, WENO). Nevertheless, these methods need to use a stencil that increases quickly with the order, which induces problems for the parallelisation and the efficiency of the implementation. Another natural extension of finite volume methods are the so-called discontinuous Galerkin methods. These methods are based on the Galerkin' idea of projecting the weak formulation of the equations on a finite dimensional space. But on the contrary to the conforming finite elements method, the approximation space is composed of functions that are continuous (typically: polynomials) inside each cell, but that are discontinuous on the sides. The discontinuous Galerkin methods are currently very popular, because they can be used with many sort of partial differential equations. Moreover, the fact that the approximation is discontinuous allows to use modern mesh adaptation (hanging nodes, which appear in non conforming mesh adaptation), and adaptive order, in which the high order is used only where the solution is smooth.

Discontinuous Galerkin methods where introduced by Reed and Hill [35] and first studied by Lesaint and Raviart [28]. The extension to the Euler system with explicit time integration was mainly led by Shu, Cockburn and their collaborators. The steps of time integration and slope limiting were similar to high order ENO schemes, whereas specific constraints given by the finite elements nature of the scheme were progressively solved, for scalar conservation laws [19], [18], one dimensional systems [17], multidimensional scalar conservation laws [16], and multidimensional systems [20]. For the same system, we can also cite the work of [22], [26], which is slightly different: the stabilisation is made by adding a nonlinear stabilisation term, and the time integration is implicit. Then, the extension to the compressible Navier-Stokes system was made by Bassi and Rebay [14], first by a mixed type finite element method, and then simplified by means of lifting operators. The extension to the $k-\omega$ RANS system was made in [13]. Another type of discontinuous Galerkin method for Navier Stokes is the so-called Symmetric Interior Penalty (SIP) method. It is used for example by Hartmann and Houston [24]. The symmetric nature of the discretization is particularly well suited with mesh adaptation by means of the adjoint equation resolution [25]. Last, we note that the discontinuous Galerkin method was already successfully tested in [21] at Direct Numerical Simulation scale for very moderate Reynolds, and also by Munz'team in Stuttgart [29], with local time stepping.

For concluding this section, there already exist numerical schemes based on the discontinuous Galerkin method which proved to be efficient for computing compressible viscous flows. Nevertheless, there remain things to be improved, which include for example: efficient shock capturing term methods for supersonic flows, high order discretization of curved boundaries, or low Mach behaviour of these schemes (this last point will be detailed in the next subsection). Another drawback of the discontinuous Galerkin methods is that they are very computationally costly, due to the accurate representation of the solution. A particular care must be taken on the implementation for being efficient.

3.3. Experimental aspects

A great deal of experiments has been devoted to the study of jet in crossflow configurations. They essentially differ one from each other by the hole shape (cylindrical or shaped), the hole axis inclination, the way by which the hole is fed, the characteristics of the crossflow and the jet (turbulent or not, isothermal or not), the number of holes considered and last but not least the techniques used to investigate the flow. A good starting point to assess the diversity of the studies carried out is given by [30]. For inclined cylindrical holes, the experimental database produced by Gustafsson and Johansson [2] represents a sound reference base and for normal injection, the work by [37] served as reference for LES simulations [34]. For shaped holes, the studies are less numerous and are aimed at assessing the influence of the hole shape on various flow properties such as the heat transfer at the wall [27]. In 2007, A. Most developed at UPPA a test facility for studying jet in crossflow issued from shaped holes [31]. The hole shape was chosen as a 12.5 scale of the holes (i.e. at scale 1) drilled by laser in a

[2] http://www.tfd.chalmers.se/~gujo/WS11_2005/Slanted_jet/INDEX.HTM

combustion chamber. His preliminary 2-component PIV results have been used to test RANS simulations [32] and LES [33]. This test facility will be used in the framework of the present project to investigate a 1-hole plane i.e. an isolated jet in crossflow. PIV and LDV metrology will be used.

4. Application Domains

4.1. Effusion cooling of aeronautical combustion chambers walls

The industrial applications of our project is the cooling of the walls of the combustion chambers encountered in the helicopter engines, and more precisely, we wish to contribute to the improvement of effusion cooling.

Effusion cooling is nowadays very widespread, especially in the aeronautical context. It consists in piercing holes on the wall of the combustion chamber. These holes induce cold jets that enter inside the combustion chamber. The goal of this jet is to form a film of air that will cool the walls of the chamber, see Figure 2.

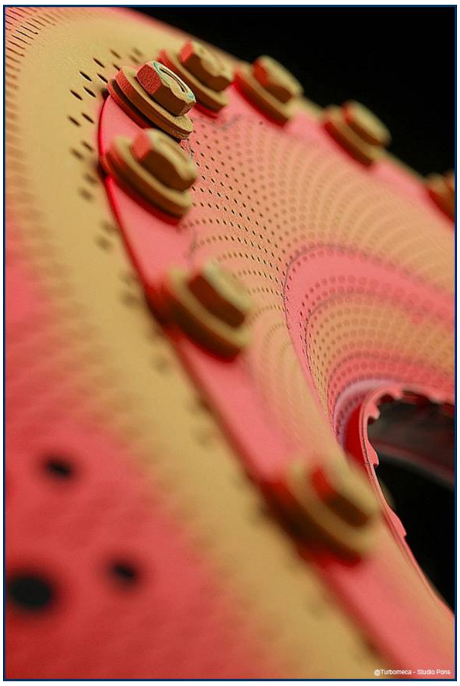

Figure 2. Effusion cooling of aeronautical combustion chambers: close view of a typical perforated chamber wall

Effusion cooling in a combustion chamber takes at the wall where thousands of small holes allow cool air to enter inside the combustion chamber. This induces jets in crossflow in charge of cooling the walls, whatever the heat and the acoustic waves present inside the chamber. Nevertheless, this technique is not straightforward to put in practice: the size, design and position of the holes can have an important effect on the cooling efficiency. For a safe and efficient functioning of the combustion chamber, it is required that the cooling jets and the combustion effects be as much independent as possible. For example, this means that

- The jets of cool air should not mix too much with the internal flow. Otherwise it will decrease the efficiency of the combustion.

- The jets should be as much stable as possible when submitted to waves emitted in the combustion chamber, e.g. acoustic waves induced by combustion instabilities. Otherwise the jets may not cool enough the walls of the combustion chamber which can then undergoes severe damages.

The first point is what we aim at simulate in this project. As the model chosen is the fully compressible Navier Stokes system, there should not be any problem in the future for being able to simulate the effect of an acoustic forcing on the jet in crossflow.

Having a database of Direct Numerical Simulations is also fundamental for testing closure laws that are used in turbulence models encountered in RANS and LES models. With such models, it is possible for example to perform optimisation.

A last aspect, that will not be dealt with in this project, but that could be dealt with in the future, is the interaction between the flow and the wall. The aim is to understand the effect of coupling between the heat propagation in the wall and the flow near the wall. A careful study of this interaction can allow to determine the exchange coefficients, and so the efficiency of the cooling by the jet. Such determination may particularly useful to develop one or multidimensional models of wall-fluid interaction [23].

From the application point of view, compressibility effects must be taken into account since the Mach number of the flow can reach values equal to 0.3, hence/or acoustic waves may be present inside the combustion chamber. This can raise a problem, because upwind numerical schemes are known to be less accurate in the low Mach limit.

5. Software

5.1. AeroSol

Participants: Damien Genet [Bacchus], Maxime Mogé, Francois Pellegrini [Bacchus], Vincent Perrier [correspondant].

The software AeroSol is jointly developed in the team Bacchus and the team Cagire. It is a high order finite element library written in C++. The code design has been carried for being able to perform efficient computations, with continuous and discontinuous finite elements methods on hybrid and possibly curvilinear meshes. The distribution of the unknowns is made with the software PaMPA, developed within the team Bacchus and the team Pumas. Maxime Mogé has been hired on a young engineer position (IJD) obtained in the ADT OuBa HOP for participating to the parallelization of the library, and arrived on November, 1st 2011.

Current features include

- **development environement** use of CMake for compilation, CTest for automatic tests and memory checking, lcov and gcov for code coverage reports.
- **In/Out** link with the XML library for handling with parameter files. Reader for GMSH, and writer on the VTK-ASCII legacy format.
- **Quadrature formula** up to 11th order for Lines, Quadrangles, Hexaedra, Pyramids, Prisms, up to 14th order for tetrahedron, up to 21st order for triangles.
- **Finite elements** up to fourth degree for Lagrange finite elements on lines, triangles and quadrangles.
- **Geometry** elementary geometrical functions for first order lines, triangles, quadrangles.
- **Time iteration** explicit Runge-Kutta up to fourth order, explicit Strong Stability Preserving schemes up to third order.
- **Linear Solvers** link with the external linear solver UMFPack.
- **Memory handling** discontinuous and continuous discretizations based on PaMPA for triangular and quadrangular meshes.
- **Numerical schemes** continuous Galerkin method for the Laplace problem (up to fifth order) with non consistent time iteration or with direct matrix inversion. Scalar stabilized residual distribution schemes with explicit Euler time iteration have been implemented for steady problems.

6. New Results

6.1. Low Mach number flows simulations issue

Participants: Pascal Bruel, Tarik Kousksou.

Since the targeted flows simulations (DNS) are by essence unsteady and at low Mach number, the use of a compressible solve has to be considered with great care. As a preliminary step towards the use of a fully implicit DG approach, we have joined the group of E. Dick at the Ghent University (Belgium) to help studying different aspects linked with low Mach number flow simulations [8], [7], [3]. The question of time consitency of the hyperbolic fluxes schemes for unsteady calculations has been dealt with and some pathological behavior of schemes such as the ones belonging to the AUSM family have been evidenced. A modification of AUSM+-up that satisfies the time-step dependency as well as the suitable scaling property of the pressure-velocity coupling evidenced has been proposed and tested [6].

6.2. Simulations of jets in crossflow

Participants: Pascal Bruel, Tarik Kousksou.

In order to prepare our benchmarking activity, we have been using the LES computer code AVBP (from Cerfacs and IFP) in order to simulate a multijet in crossflow configuration corresponding to the MAVERIC flow configuration. These simulations have been done in partnership with Turbomeca. The comparisons bteween LES and experiments is quite encouraging. An acoustic forcing technique has been used to establish a stationary acoustic wave inside the crossflow. The mass flow rate through the differents holes proved to be significantly altered by the presence of the planar acoustic wave [4].

6.3. Discontinuous Galerkin methods for compressible multiphase flows

Participants: Vincent Perrier, Erwin Franquet.

We developed discontinuous Galerkin methods for compressible multiphase flows. This method is based on the method developed in [12] concerning the modelling of multiphase flows, and on the method developed in [36] concerning multiphase flows. In the method developed in [12], the exact expression of the continuous system is unclear, and as a consequence, the cell integral that naturally appear in the discontinuous Galerkin formulation is not clearly defined. In [2], [11], we developed an original analysis of the scheme [12] by mean of a stochastic process, which gave a unified framework for defining both the numerical scheme and the continuous limit. This was then applied to multiphase flows with phase transition in [10], [1]. Our method is currently being extended to interface flows with the maximum preserving limiter developed by [38].

7. Partnerships and Cooperations

7.1. National Initiatives

7.1.1. GIS Success

Participants: Vincent Perrier, Pascal Bruel.

We are presently participating in the CNRS GIS (Groupement d'Intérêt Scientifique) which is provisionally called "Super-calcul en Combustion et en Mécanique des Fluides dans les Géométries Complexes". This GIS intends to gather people working on HPC simulations for fluid mechanics. The GIS is led by CORIA (Rouen); the other members of this GIS are EM2C (Centrale Paris), I3M (Montpellier), LEGI (Grenoble), IMFT (Toulouse), CERFACS, and IFP-EN. The kick-off meeting will be held in Toulouse on March 2012.

7.2. European Initiatives

7.2.1. FP7 projects

Participants: Vincent Perrier [correspondant], Pascal Bruel.

Program: Propulsion

Project acronym: IMPACT-AE

Project title: Intelligent Design Methodologies for Low Pollutant Combustors for Aero-Engines

Duration: 01/11/2011 - 31/10/2015

Coordinator: Roll Royce Deutschland

Other partners:

- France: Insa of Rouen, ONERA, Snecma, Turbomeca.
- Germany: Rolls-Royce Deutschland, MTU Aeo Engine Gmbh, DLR, Technology Institute of Karlsruhe, University of Bundeswehr (Munich)
- Italy: AVIOPROP SRL, AVIO S.P.A., University of Florence
- United Kingdom: Rolls Royce PLC, Cambridge University, Imperial College od Science, Technology and Medecine, Loughborough University.

Abstract: The environmental benefits of low emissions lean burn technology in reducing NOx emissions up to 80only be effective when these are deployed to a large range of new aero-engine applications. While integrating methodologies for advanced engine architectures and thermodynamic cycles. It will support European engine manufacturers to pick up and keep pace with the US competitors, being already able to exploit their new low emission combustion technology to various engine applications with short turn-around times. Key element of the project will be the development and validation of design methods for low emissions combustors to reduce NOx and CO emissions by an optimization of the combustor aero-design process. Preliminary combustor design tools will be coupled with advanced parametrisation and automation tools. Improved heat transfer and NOx models will increase the accuracy of the numerical prediction. The advanced representation of low emission combustors and the capability to investigate combustor scaling effects allow an efficient optimisation of future combustors targeting a cut of combustor development time by 50work packages: WP1'Development of smart design methodologies for clean combustion' as central WP to deliver the new methodology for combustor design, WP2'Modelling and design of advanced combustor wall cooling concepts' for combustor liner design definition as key technology area, WP3'Technology validation by detailed flame diagnostics' to substantiate fuel injector design rules implemented into the design methodology and WP4'Methodology demonstration for efficient low NOx combustors' will validate the combustor design. The consortium consists of all major aero-engine manufactures in Europe, 7 universities and 3 research establishments with recognised experience in low emission combustion research and 10 SMEs. The contribution of our team is to create a direct numerical simulations (DNS) database relevant to the configuration of film cooling for subsequent improvement of RANS based simulations of isothermal and non isothermal wall flows with discrete mass transfer.

7.2.2. Major European Organizations with which Cagire has followed Collaborations

University of Ghent, Department of Flow, Heat and Combustion (Belgium)

Subject: this cooperation with E. Dick and Y. Moguen focuses on the improvement of the efficiency of numerical schemes used to simulate low Mach number flows.

7.3. International Initiatives

7.3.1. Visits of International Scientists

- Dr. A. Naïmanova, Institute of Mathematics, Almaty, Kazakhstan came for a one-month stay in September 2011.

8. Dissemination

8.1. Animation of the scientific community

Participants: Pascal Bruel, Erwin Franquet, Tarik Kousksou.

The team members have been invited to review for the following journals:

- Journal of Computational Physics
- International Journal for Numerical Methods in Fluids
- Journal of Mechanical Engineering Science
- Combustion and Flame
- AIAA Journal of Thermophysics and Heat Transfer
- Journal of Aerospace Engineering
- Computational Thermal Science

8.2. Teaching

Participants: Pascal Bruel [PB], Erwin Franquet [EF], Tarik Kousksou [TK].

Licence :

TP Transferts thermiques, 48h, L1, IUT-GTE-UPPA, Pau, France. [PB]

Programmation, 50h, L3, ENSGTI-UPPA, France [EF]

TP Composants, 40h, L3, ENSGTI-UPPA, France [EF]

Master :

An introduction to the numerical simulation of reacting flows, 15h, M2, ISAE-SupAéro, Toulouse, France. [PB]

Machines hydrauliques, 30h, M1, ENSGTI-UPPA, France [TK]

Machines aérauliques, 30h, M1, ENSGTI-UPPA, France [TK]

Thermo-économie, 30h M2, ENSGTI-UPPA, France [TK]

Modélisation des écoulements diphasiques, 30h, M1, ENSGTI-UPPA, France [TK]

TP systèmes, 50h, M1, ENSGTI-UPPA, France [TK]

Simulation industrielle, 40h, M1, ENSGTI-UPPA, France [EF]

Fluides compressibles, 20h, M1, ENSGTI-UPPA, France [EF]

Combustion industrielle, 30h, M1, ENSGTI-UPPA, France [EF]

Réseaux de chaleur, 4h, M2, ENSGTI-UPPA, France [EF]

Géothermie, 4h, M2, ENSGTI-UPPA, France [EF]

Biomasse, 4h, M2, ENSGTI-UPPA, France [EF]

9. Bibliography

Publications of the year

Articles in International Peer-Reviewed Journal

[1] E. FRANQUET, V. PERRIER. *Runge-Kutta Discontinuous Galerkin method for reactive multiphase flows*, 2011, submitted.

[2] E. FRANQUET, V. PERRIER. *Runge-Kutta Discontinuous Galerkin method for the approximation of Baer & Nunziato type multiphase models*, 2011, submitted.

[3] Y. MOGUEN, T. KOUSKSOU, P. BRUEL, J. VIERENDEELS, E. DICK. *Pressure-velocity coupling allowing acoustic calculation in low Mach number flow*, 2011, submitted.

[4] E. MOTHEAU, T. LEDERLIN, J.-L. FLORENCIANO, P. BRUEL. *LES investigation of the flow through an effusion-cooled aeronautical combustor model*, in "Flow Turbulence and Combustion", 2011, in press [*DOI :* 10.1007/s10494-011-9357-9].

[5] V. PERRIER. *A conservative method for the simulation of the isothermal Euler system with the van-der-Waals equation of state*, in "Journal of Scientific Computing", 2011, vol. 48, n^{o} 1-3, p. 296-303 [*DOI :* 10.1007/s10915-010-9415-9], http://hal.inria.fr/inria-00633917/en.

International Conferences with Proceedings

[6] Y. MOGUEN, E. DICK, J. VIERENDEELS, P. BRUEL. *Pressure-velocity coupling for unsteady low Mach number flow*, in "Proceedings of the 5 t h International Conference on Advanced Computational Methods in Engineering", M. HOGGE, R. V. KEER, E. DICK, B. MALENGIER, M. SLODICKA, E. BÉCHET, C. GUEUZAINE, L. NOELS, J.-F. REMACLE (editors), 2011, ISBN 978-2-9601143-1-7.

[7] Y. MOGUEN, T. KOUSKSOU, P. BRUEL, J. VIERENDEELS, E. DICK. *Rhie-Chow interpolation for low Mach number flow computation allowing small time steps*, in "Proceedings of the 6th International Symposium on Finite Volumes for Complex Applications", J. FORT, J. FÜRST, J. HALAMA, R. HERBIN, F. HUBERT (editors), Springer Verlag, 2011, ISBN 978-3-642-20670-2.

[8] Y. MOGUEN, T. KOUSKSOU, E. DICK, P. BRUEL. *On the role of numerical dissipation in unsteady low Mach number flows computations*, in "Proceedings of the 6th International Conference on Computational Fluid Dynamics", A. KUZMIN (editor), Springer Verlag, 2011, ISBN 978-3-642-17883-2.

[9] V. PERRIER, E. FRANQUET. *Runge–Kutta Discontinuous Galerkin method for multi–phase compressible flows*, in "International Conference on Computational Fluid Dynamics (ICCFD 2010)", Springer, 2011.

Conferences without Proceedings

[10] E. FRANQUET, V. PERRIER. *Runge–Kutta Discontinuous Galerkin method for reactive multiphase flows*, in "International Conference on numerical Methods For Multi-Material Fluid Flows (MULTIMAT 2011)", 2011.

[11] V. PERRIER, E. FRANQUET. *High order method for multiphase compressible flows with RKDG schemes*, in "European Workshop on High Order Nonlinear Numerical Methods for Evolutionary PDEs: Theory and Applications (HONOM 2011)", 2011.

References in notes

[12] R. ABGRALL, R. SAUREL. *Discrete equations for physical and numerical compressible multiphase mixtures*, in "Journal of Computational Physics", 2003, vol. 186, n^o 361-396.

[13] F. BASSI, A. CRIVELLINI, S. REBAY, M. SAVINI. *Discontinuous Galerkin solution of the Reynolds-averaged Navier-Stokes and k-omega turbulence model equations*, in "Computers & Fluids", 2005, vol. 34, n^o 4-5, p. 507-540.

[14] F. BASSI, S. REBAY. *A high-order accurate discontinuous finite element method for the numerical solution of the compressible Navier-Stokes equations*, in "J. Comput. Phys.", 1997, vol. 131, n^o 2, p. 267–279, http://dx.doi.org/10.1006/jcph.1996.5572.

[15] V. BILLEY, J. PERIAUX, B. STOUFFLET, A. DERVIEUX, L. FEZOUI, V. SELMIN. *Recent improvements in Galerkin and upwind Euler solvers and application to 3-D transonic flow in aircraft design*, in "Computer Methods in Applied Mechanics and Engineering", 1989, vol. 75, n^o 1-3, p. 409-414.

[16] B. COCKBURN, S. HOU, C.-W. SHU. *The Runge-Kutta local projection discontinuous Galerkin finite element method for conservation laws. IV. The multidimensional case*, in "Math. Comp.", 1990, vol. 54, n^o 190, p. 545–581, http://dx.doi.org/10.2307/2008501.

[17] B. COCKBURN, S. Y. LIN, C.-W. SHU. *TVB Runge-Kutta local projection discontinuous Galerkin finite element method for conservation laws. III. One-dimensional systems*, in "J. Comput. Phys.", 1989, vol. 84, n^o 1, p. 90–113.

[18] B. COCKBURN, C.-W. SHU. *TVB Runge-Kutta local projection discontinuous Galerkin finite element method for conservation laws. II. General framework*, in "Math. Comp.", 1989, vol. 52, n^o 186, p. 411–435, http://dx.doi.org/10.2307/2008474.

[19] B. COCKBURN, C.-W. SHU. *The Runge-Kutta local projection P^1-discontinuous-Galerkin finite element method for scalar conservation laws*, in "RAIRO Modél. Math. Anal. Numér.", 1991, vol. 25, n^o 3, p. 337–361.

[20] B. COCKBURN, C.-W. SHU. *The Runge-Kutta discontinuous Galerkin method for conservation laws. V. Multidimensional systems*, in "J. Comput. Phys.", 1998, vol. 141, n^o 2, p. 199–224, http://dx.doi.org/10.1006/jcph.1998.5892.

[21] S. S. COLIS. *Discontinuous Galerkin methods for turbulence simulation*, in "Proceedings of the Summer Program", Center for Turbulence Research, 2002.

[22] M. FEISTAUER, V. KUČERA. *On a robust discontinuous Galerkin technique for the solution of compressible flow*, in "J. Comput. Phys.", 2007, vol. 224, n^o 1, p. 208–221, http://dx.doi.org/10.1016/j.jcp.2007.01.035.

[23] R. J. GOLDSTEIN, E. ECKERT, W. E. IBELE, S. V. PATANKAR, T. W. SIMON, T. H. KUEHN, P. J. STRYKOWSKI, K. K. TAMMA, A. BAR-COHEN, J. V. R. HEBERLEIN, J. H. DAVIDSON, J. BISCHOF, F.

A. KULACKI, U. KORTSHAGEN, S. GARRICK. *Heat transfer - A review of 2000 literature*, in "International Journal of Heat and Mass Transfer", 2002, vol. 45, n^{o} 14, p. 2853-2957 [*DOI :* DOI: 10.1016/S0017-9310(02)00027-3].

[24] R. HARTMANN, P. HOUSTON. *Symmetric interior penalty DG methods for the compressible Navier-Stokes equations. I. Method formulation*, in "Int. J. Numer. Anal. Model.", 2006, vol. 3, n^{o} 1, p. 1–20.

[25] R. HARTMANN, P. HOUSTON. *Symmetric interior penalty DG methods for the compressible Navier-Stokes equations. II. Goal-oriented a posteriori error estimation*, in "Int. J. Numer. Anal. Model.", 2006, vol. 3, n^{o} 2, p. 141–162.

[26] C. JOHNSON, A. SZEPESSY, P. HANSBO. *On the convergence of shock-capturing streamline diffusion finite element methods for hyperbolic conservation laws*, in "Math. Comp.", 1990, vol. 54, n^{o} 189, p. 107–129, http://dx.doi.org/10.2307/2008684.

[27] H. LEE, J. PARK, J. LEE. *Flow visualization and film cooling effectiveness measurements around shaped holes with compound angle orientations*, in "Int. J. Heat Mass Transfer", 2002, vol. 45, p. 145-156.

[28] P. LESAINT, P.-A. RAVIART. *On a finite element method for solving the neutron transport equation*, in "Mathematical aspects of finite elements in partial differential equations (Proc. Sympos., Math. Res. Center, Univ. Wisconsin, Madison, Wis., 1974)", Math. Res. Center, Univ. of Wisconsin-Madison, Academic Press, New York, 1974, p. 89–123. Publication No. 33.

[29] F. LÖRCHER, G. GASSNER, C.-D. MUNZ. *An explicit discontinuous Galerkin scheme with local time-stepping for general unsteady diffusion equations*, in "J. Comput. Phys.", 2008, vol. 227, n^{o} 11, p. 5649–5670, http://dx.doi.org/10.1016/j.jcp.2008.02.015.

[30] R. MARGASON. *Fifty Years of Jet in Cross Flow Research*, in "NATO AGARD Conference", Winchester, UK, 1993, vol. CP-534, p. 1.1-1.41.

[31] A. MOST. *Étude numérique et expérimentale des écoulements pariétaux avec transfert de masse à travers une paroi multi-perforée*, Pau University, 2007.

[32] A. MOST, N. SAVARY, C. BÉRAT. *Reactive flow modelling of a combustion chamber with a multiperforated liner*, in "43rd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit", Cincinnati, OH, USA, AIAA Paper 2007-5003, 8-11 July 2007.

[33] E. MOTHEAU, T. LEDERLIN, P. BRUEL. *LES investigation of the flow through an effusion-cooled aeronautical combustor model*, in "8th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements", Marseille, France, June 2010, p. 872-877.

[34] C. PRIÈRE. *Simulation aux grandes échelles: application au jet transverse*, INP Toulouse, 2005.

[35] W. REED, T. HILL. *Triangular mesh methods for the neutron transport equation*, Los Alamos Scientific Laboratory, 1973, n^{o} LA-UR-73-479.

[36] S. RHEBERGEN, O. BOKHOVE, J. J. W. VAN DER VEGT. *Discontinuous Galerkin finite element methods for hyperbolic nonconservative partial differential equations*, in "J. Comput. Phys.", 2008, vol. 227, n^{o} 3, p. 1887–1922, http://dx.doi.org/10.1016/j.jcp.2007.10.007.

[37] S. SMITH, M. MUNGAL. *Mixing, structure and scaling of the jet in crossflow*, in "Journal of Fluid Mechanics", 1998, vol. 357, p. 83-122.

[38] X. ZHANG, Y. XIA, C.-W. SHU. *Maximum-Principle-Satisfying and Positivity-Preserving High Order Discontinuous Galerkin Schemes for Conservation Laws on Triangular Meshes*, in "Journal of Scientific Computing.", 2011, In press [*DOI :* 10.1007/S10915-011-9472-8].