Activity Report 2014

Team CAGIRE

Computational Approximation with discontinous Galerkin methods and compaRison with Experiments

RESEARCH CENTER
Bordeaux - Sud-Ouest

THEME
Numerical schemes and simulations

Table of contents

Team CAGIRE

Keywords: Fluid Dynamics, Direct Numerical Simulation, Finite Elements, Turbulence Modeling, Experiments, Internal Aerodynamic, Numerical Methods, Parallel Solver

Creation of the Team: 2011 June 01.

1. Members

Research Scientists

Pascal Bruel [Team leader, CNRS, Researcher, HdR]
Rémi Manceau [CNRS, Researcher, HdR]
Vincent Perrier [Inria, Researcher]

Faculty Member

Erwin Franquet [Univ. Pau, Associate Professor, until Dec 14]

Engineer

Benjamin Lux [Inria, since Nov 2014]

Post-Doctoral Fellow

Yann Moguen [Univ. Pau, until Nov 2014]

Administrative Assistants

Josette Baron [Inria, until May 2014]
Sylvie Embolla [Inria]
Nicolas Jahier [Inria]

Others

Hamza Belkhayat Zougari [Inria, Master 2/ISAE Supaéro student, from Jun 2014 until Nov 2014]
Simon Delmas [Univ. Pau, PhD student]
Jean-François Wald [Univ. Pau, PhD student]

2. Overall Objectives

2.1. Overall Objectives

This project aims at developping suitable tools for studying turbulent flows that are often encountered in internal aerodynamic such as the impinging jet or the 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] developed at LMAP and its metrology are 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 is experimentally 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 used in order to experimentally characterize the flowfield.

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

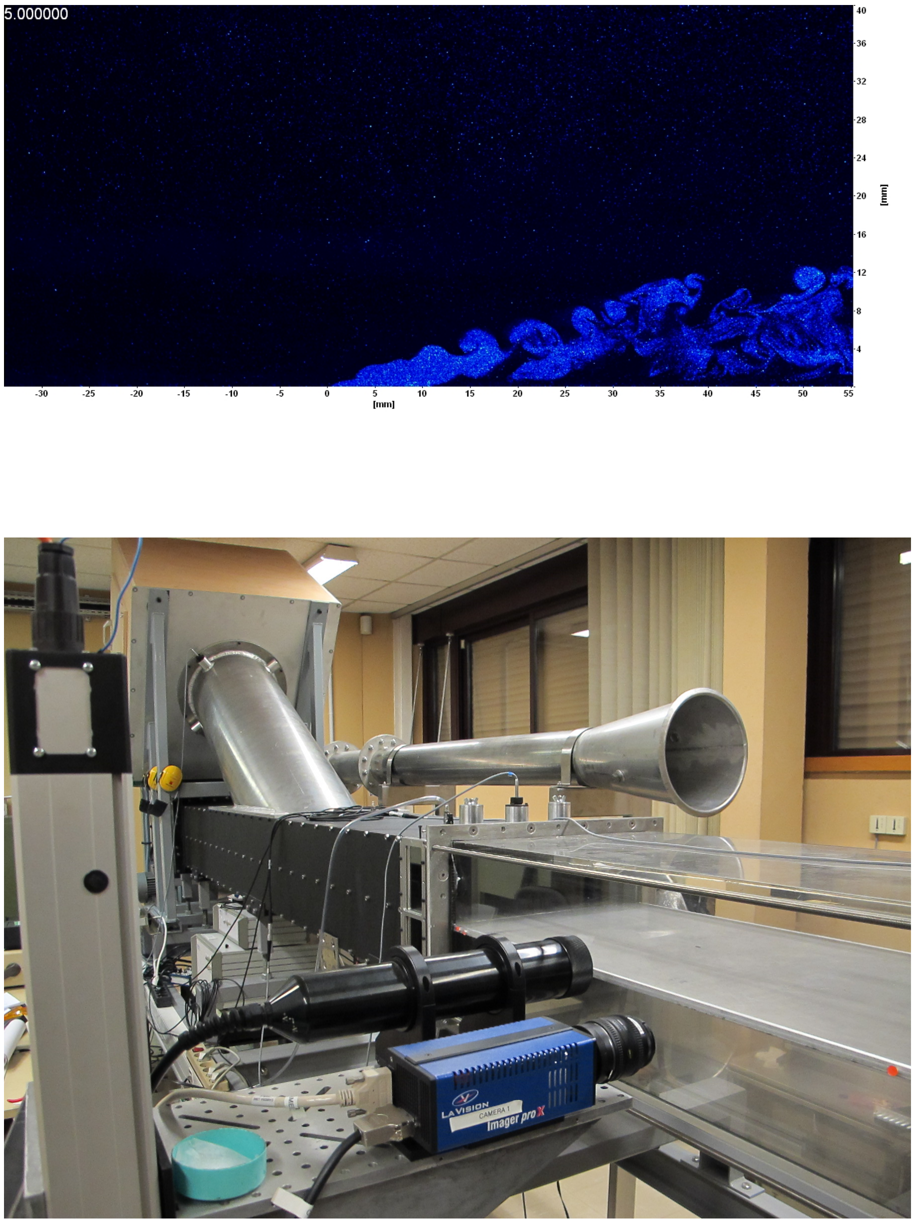

Figure 1. MAVERIC test facility: visualization of a single jet in crossflow (top) and overview of the test rig (bottom).

A close interaction during the course of the project between experiments, simulation and physical modelling represents the backbone of our methodology. From the simulation point of view, one of the short term aims is to perform a direct numerical simulation of an isothermal configuration of an inclined turbulent jet discharging into a turbulent crossflow. This is done in the framework of our participation in the IMPACT-AE EU funded program (http://www.impact-ae.eu/). The flows Mach number being small, preserving the accuracy of the specifically developed compressible solver for such flow regime represents quite a challenge. 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 quite 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 is mostly related to the capability of ensuring that the flow simulated and the flow experimentally investigated are as close as possible.

In short, the team members are presently developing their activity around the following axes:

- The development of a DNS tool able to simulate low Mach number turbulent flow and provide highly accurate simulations results.
- the development of low Mach schemes for unsteady flow simulations.
- The development of advanced turbulence RANS and hybrid RANS-LES turbulence models adapted to zero Mach and low Mach turbulent flows with a specific emphasis on the wall-flow interaction.
- The development of a test facility to generate high quality experimental data for the purpose of flow analysis, turbulence models and simulations results assessment.

3. Research Program

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 $Re^{3/4}$ per dimension. In addition, for internal flows, the viscous effects near the solid walls yield a scaling proportional to Re per dimension. The smallest scales may have a certain effect on the largest ones which implies that an accurate framework for the modelling and the computation of such turbulent flows must take into account all these scales of time and space fluctuations. This can be achieved either by solving directly the Navier-Stokes (NS) equations (Direct numerical simulations or DNS) or by first applying to them a filtering operation either in time or space. In the latter cases, the closure of the new terms that appear in the filtered equations due to the presence of the non-linear terms implies the recourse to a turbulence model before discretizing and then solving the set of resulting governing equations. Among these different methodologies, the Reynolds averaged Navier-Stokes (RANS) approach yields a system of equations aimed at describing the mean flow properties. The term mean is referring to an ensemble average which is equivalent to a time average only when the flow is statistically stationnary. In that case, the turbulence model aims at expressing the Reynolds stresses either through the solution of dedicated transport equations (second order modelling) or via the recourse to the concept of turbulent viscosity used to write and ad-hoc relation (linear or not) between the Reynolds stress and the mean velocity gradient tensor. If the filtering operation involves a convolution with a filter function in space of width δ, this corresponds to the large-eddy simulation (LES) approach. The structures of size below δ are filtered out while the bigger structures are directly resolved. The resulting set of filtered equations is again not closed and calls for a model aimed at providing a suitable expression for the subgrid scale stress tensor.

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 as well as hybrid methods that combine RANS and LES. By simulating the large scale structures while modelling the smallest ones supposed to be more isotropic, the LES, alone or combined with the most adanced RANS models such as the EB-RSM model [4] 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).

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 [18]. 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 [39] and first studied by Lesaint and Raviart [32]. 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 [22], [21], one dimensional systems [20], multidimensional scalar conservation laws [19], and multidimensional systems [23]. For the same system, we can also cite the work of [25], [30], 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 [17], 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 [16]. 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 [28]. The symmetric nature of the discretization is particularly well suited with mesh adaptation by means of the adjoint equation resolution [29]. Last, we note that the discontinuous Galerkin method was already successfully tested in [24] at Direct Numerical Simulation scale for very moderate Reynolds, and also by the Munz's team in Stuttgart [33], 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: efficient shock capturing term methods for supersonic flows, high order discretization of curved boundaries, or low Mach behaviour of these schemes. Another drawback of the discontinuous Galerkin methods is that they are very computationally costly, due to the accurate representation of the solution. Accordingly, a particular care must be taken on the implementation for being efficient.

3.3. Flow analysis and CFD assessment: experimental aspects

The capability of producing in-situ experimental data is another originality of our project. By carefully controlling the flow configuration and the type of data we are measuring, we are in situation of assessing in depth the quality of our simulations results over the complete spectrum of possible approaches ranging from DNS, RANS and Hybrid RANS-LES models that the team is developing or LES.

The flow configuration we have chosen is that of a jet in cross-flow since it features large scale coherent structures, flow separation, turbulence and wall-flow interaction.

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 [34]. 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 [40] served as reference for LES simulations [38]. 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 [31]. In 2007, Most [35] developed at UPPA a test facility for studying jet in crossflow issued from shaped holes. The hole shape was chosen as a 12.5 scale of the holes (i.e. at scale 1) drilled by laser in a combustion chamber. His preliminary 2-component PIV results have been used to test RANS simulations [36] and LES [37]. Later, in the framework of the KIAI FP7 European programme, Florenciano [26] upgraded the rig by implementing an acoustic forcing device of the crossflow stream and by performing phase-locked PIV measurements that were used to test the accuracy of LES results. Thus, this test facility is extensively used in the framework of the present project to investigate a 1-hole cylindrical inclined jet interacting with a turbulent crossflow. PIV and LDV metrology are the workhorses as far as metrology is concerned.

4. Application Domains

[2] Slanted jet

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.

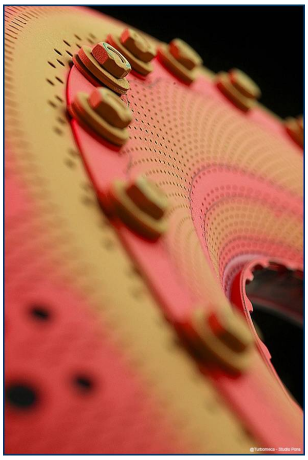

Figure 2. Effusion cooling of aeronautical combustion chambers: close view of a typical perforated chamber wall

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.

An important aspect that we began to adress in this project 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 be particularly useful to develop one or multidimensional models of wall-fluid interaction [27]. The large eddy simulation performed by Florenciano [26] clearly put into evidence the strong effect of the presence of an acoustic wave in the crossflow on the dynamics of the heat transfer coefficient at the wall.

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. New Software and Platforms

5.1. AeroSol

Participants: Hamza Belkhayat Zougari [Cagire], Simon Delmas [Cagire], Damien Genet [Bacchus], Francois Pellegrini [Bacchus], Vincent Perrier [Cagire, correspondant], Mario Ricchiuto [Bacchus].

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 work of the team Bacchus is focused on continuous finite elements methods, while the team Cagire is focused on discontinuous Galerkin methods. However, everything is done for sharing the largest part of code we can. More precisely, classes concerning IO, finite elements, quadrature, geometry, time iteration, linear solver, models and interface with PaMPAare used by both of the teams. This modularity is achieved by mean of template abstraction for keeping good performances.

The distribution of the unknowns is made with the software PaMPA, developed within the team Bacchus and the team Castor.

At the end of 2013, Aerosol had the following features

- **development environment** use of CMake for compilation (gcc, icc and xlc), CTest for automatic tests and memory checking, lcov and gcov for code coverage reports. Development of a CDash server for collecting the unitary tests and the memory checking. Beginning of the development of an interface for functional tests. Optional linking with HDF5, PAPI, with dense small matrices libraries (BLAS, Eigen)
- **In/Out** link with the XML library for handling with parameter files. Parallel reader for GMSH, with an embedded geometrical pre-partitioner. Writer on the VTK-ASCII legacy format (cell and point centered). Parallel output in vtu and pvtu (Paraview) for cell-centered visualization, and XDMF/HDF5 format for both cell and point centered visualization.
- **Quadrature formula** up to 11th order for Lines, Quadrangles, Hexaedra, Pyramids, Prisms, up to 14th order for tetrahedron, up to 21st order for triangles. Gauss-Lobatto type quadrature formula for lines, triangles, quadrangles and hexaedra.
- **Finite elements** up to fourth degree for Lagrange finite elements and hierarchical orthogonal finite element basis (with Dubiner transform on simplices) on lines, triangles, quadrangles, tetrahedra, prisms, hexaedra and pyramids. Finite element basis that are interpolation basis on Gauss-Legendre points for lines, quadrangles, and hexaedra, and triangle (only 1st and 2nd order)
- **Geometry** elementary geometrical functions for first order lines, triangles, quadrangles, prisms, tetrahedra, hexaedra and pyramids.
- **Time iteration** explicit Runge-Kutta up to fourth order, explicit Strong Stability Preserving schemes up to third order. Optimized CFL time schemes: SSP(2,3) and SSP(3,4). CFL time stepping. Implicit integration with BDF schemes from 2nd to 6th order
- **Linear Solvers** link with the external linear solver UMFPack, PETSc and MUMPS. Internal solver for diagonal and block-diagonal matrices.

- **Memory handling** discontinuous and continuous, sequential and parallel discretizations based on PaMPA for generic meshes, including hybrid meshes.
- **Models** Perfect gas Euler system, real gas Euler system (template based abstraction for a generic equation of state), scalar advection, Waves equation in first order formulation, generic interface for defining space-time models from space models. Diffusive models: isotropic and anisotropic diffusion, compressible Navier-Stokes.
- **Numerical schemes** continuous Galerkin method for the Laplace problem (up to fifth order) with non consistent time iteration or with direct matrix inversion. Explicit and implicit discontinuous Galerkin methods for hyperbolic systems, diffusive and advection-diffusion problems. Beginning of optimization by stocking the geometry for advection problems. SUPG and Residual disribution schemes.
- **Numerical fluxes** centered fluxes, exact Godunov' flux for linear hyperbolic systems, and Lax-Friedrich flux. Riemann solvers for Low Mach flows.
- **Boundary conditions** Periodic boundary conditions, time-dependent inlet and outlet boundary conditions.
- **Parallel computing** Mesh redistribution, computation of Overlap with PaMPA. collective asynchronous communications (PaMPA based). Asynchronous point to point communications. Tests on the cluster Avakas from MCIA, and on Mésocentre de Marseille, and PlaFRIM. Tuer-1 Turing (Blue-Gene).
- **C++/Fortran interface** Tests for binding fortran with C++.
- **Instrumentation** Aerosol can give some traces on memory consumption/problems with an interfacing with the PAPI library. Tests have also been performed with VTUNE and TAU.

This year, the following features were added

- **In/Out** Ability of saving the high order solution and restarting from it. Computation of volumic and probe statistics. Ability of saving averaged layer data in quad and hexa meshes. Ability of defining user defined output visualization variables.
- **Geometry** handling of high order meshes.
- **Time iteration** Newton method for stationary problems. Implicite unstationary time iterator non consistent in time for stationary problems. Implementation of in house GMRES and conjugate gradient based on Jacobian free iterations.
- **Models** scalar advection-diffusion model
- **Numerical schemes** Optimization of DG schemes for advection-diffusion problems: stocking of the geometry and use of BLAS for all the linear phases of the scheme.
- **Numerical fluxes** Development of a new numerical flux accurate for steady and unsteady computations.
- **Boundary conditions** Adiabatic wall and isothermal wall; Steger-Warming based boundary condition.
- **Instrumentation** Tests with Maqao and Scalasca (VIHPS workshop)
- **Validation** Poiseuille, Taylor-Green vortex. Laplace equation on a ring and Poiseuille flow on a ring. Implementation of volumic forcing based on wall dissipation.

6. New Results

6.1. DNS of a Taylor Green vortex

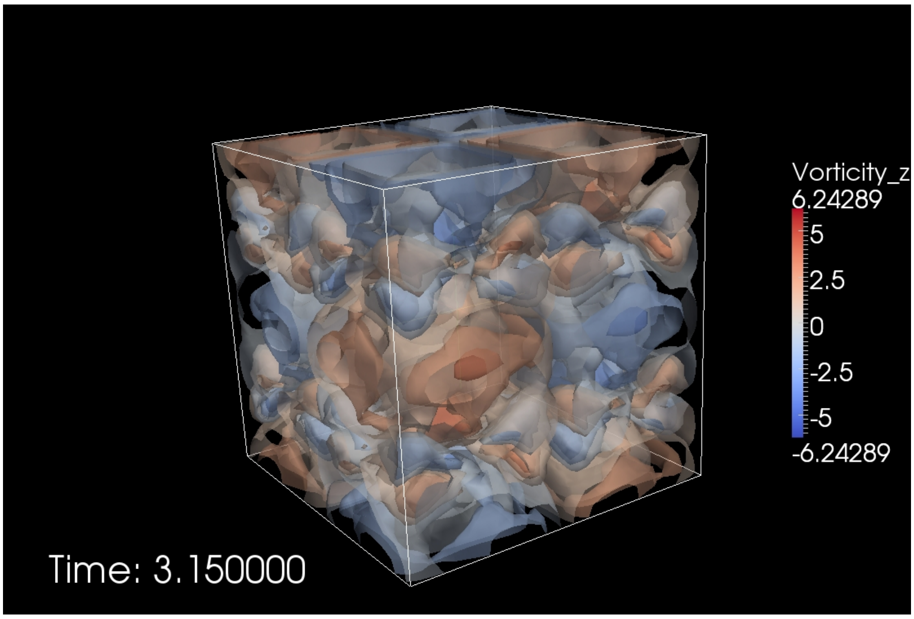

Figure 3. AeroSol simulation of a Taylor-Green vortex: snapshot of one component of the vorticity.

In 2014, we finished the validation of Navier-Stokes discretization with the discontinuous Galerkin method in the Aerosol library. The result of Figure 3 is a first validation in turbulence conditions. The Taylor-Green vortex case is part of the C3 (i.e. "difficult") test cases of the high order CFD workshop, see https://www.grc.nasa.gov/hiocfd/.

6.2. Low Mach number flows simulations issues

Our activity for developing schemes suitable for the simulation of low Mach number flows considers the two main techniques developed initially for dealing with either zero Mach number flows (pressure-velocity coupling) or compressible flows (density based approach). For both approaches, we concentrated this year on the specific difficulties related to unsteady flows simulations. For the methodology adressing the pressure-velocity coupling with a low-order discretization technique, we introduced an inertia term in the AUSM+ -up scheme. The resulting scheme, called AUSM-IT (IT for Inertia Term), was designed as an extension of the AUSM+ -up scheme allowing for full Mach number range calculations of unsteady flows including acoustic features. In line with the continuous asymptotic analysis, the AUSM-IT scheme satisfies the conservation of the discrete linear acoustic energy at first order in the low Mach number limit. Its capability to properly handle low Mach number unsteady flows, that may include acoustic waves or discontinuities was numerically illustrated [7].

As far as density based approach are concerned, an analysis of explicit RKDG schemes have been performed for unstationary acoustic waves propagating in a low Mach number flow. Classical cures of the unaccuracy of upwind schemes at low Mach number consist in using centered flux on the pressure. By a two scale asymptotic expansion of the scheme, we proved that this cure is a dead end for resolving unstationary acoustic waves, because it leads to a non dissipative scheme for the wave equations. We developed a dissipative term that can both stabilize the stationary incompressible equations, and the system of acoustic waves. The results with this new type of scheme have been presented in [8].

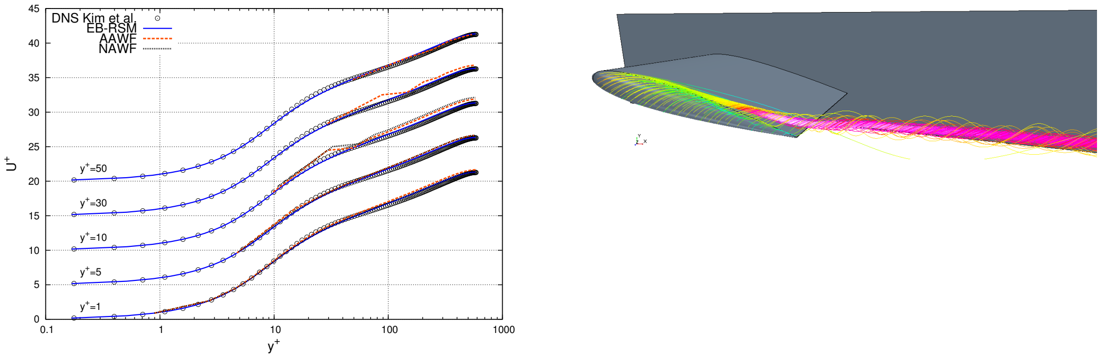

Figure 4. Left: Computation (Code_Saturne) of turbulent channel flow at 3 Reynolds numbers. Comparison with reference DNS of the results given by the EB-RSM integrated down to the wall (ItW, fine mesh) and the EB-RSM with analytical adaptive wall function (AAWF, 3 meshes). Right: EB-RSM computation (STARCCM+ code) of the wing-tip vortex generated by the flow around a NACA 0012 at 10 deg incidence. Visualisation of the streamlines colored with the streamwise vorticity.

6.3. Improving the flexibility of turbulence models for industrial applications

In collaboration with industrial partners (EDF and CD-Adapco) developing CFD codes (code_Saturne and STARCCM+, respectively), we are working on the flexibility and robustness of the EB-RSM, an advanced Reynolds-stress turbulence model. Indeed, the two main problems that slow down the spreading of the use of such low-Reynolds number models (i.e., integrating the equations down to solid boundaries) in the industry are the impossiblity to control the near-wall mesh quality in the whole domain of a complex industrial application and the occurence of numerical instabilities due to spurious relaminarizations in some configurations.

In order to address the first issue, we are working, in particular in the frame of the PhD thesis of J.-F. Wald, on the development of adaptive wall functions, i.e., non-homogeneous Dirichlet boundary conditions for the turbulent variables dependant on the size of the cell adjacent to the wall. These wall functions are based on the physical properties of turbulence in the different layers of the near-wall region (asymptotic behaviour in the viscous sublayer and log law in the equilibrium layer), such a way that the flow is correctly reproduced whatever the near-wall refinement of the mesh. Fig. 4 (left) shows that the reproduction of the mean velocity profile in turbulent channel flows obtained using a typical, industrial mesh ($y^+ = 50$) remains very close to the grid-converged solution.

The second issue, the numerical instabilities due to local, spurious relaminarization of the model, can be addressed by investigating the solutions of the dynamical system formed by the model equations in homogeneous situations. Equilibrium solution are intersections of the nullclines (the locus of steady solutions for individual equations) and the stability properties of these fixed points can be visualized using trajectories in the phase space. By investigating the dependance of these stability properties on the parameters of the model, it is possible to eliminate undesired stable fixed points and thus to avoid the appearance of spurious laminarization. Fig. 4 (right) shows the fully turbulent solution obtained with the modified model in a case where the original model exhibited a severe, unphysical relaminarization of the wing-tip vortex.

6.4. Assessment of the discontinuous Galerkin methods on curved meshes

The internship of Hamza Belkhayat-Zougari was concerned with the handling of high order curved meshes in the Aerosol library. During his internship, we developed new analytical solutions of the Laplace and of the Navier-Stokes equations on curved domains for emphasizing the limitation at second order of high order methods on straight meshes, and for assessing the right order on high order meshes. Example of order obtained on straight and curved meshes can be found on Figure 5.

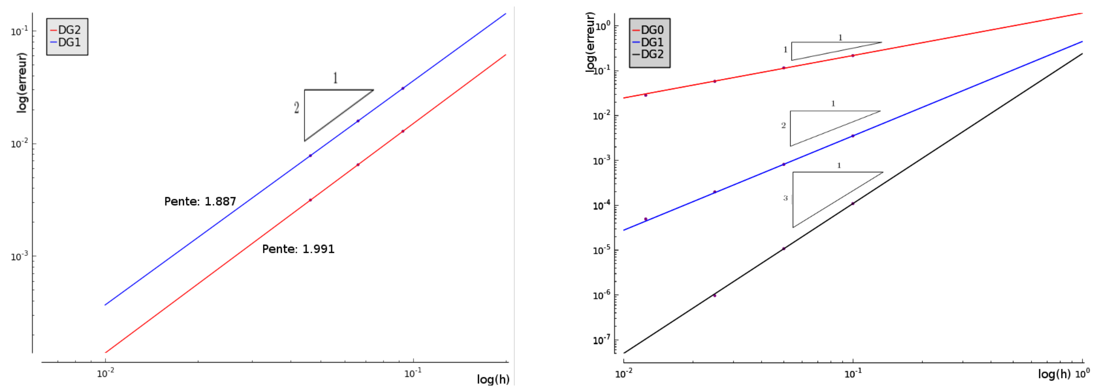

Figure 5. Convergence on a ring for the Laplace equation. Left: high order method on a straight mesh is limited to two. Right: third order accuracy can be recovered by using a second order mesh.

7. Bilateral Contracts and Grants with Industry

7.1. Bilateral Contracts with Industry

Collaborative research contract with EDF: "Nouveau modèle de turbulence Haut-Bas Reynolds avec prise en compte de la thermique active ou passive. (New high-low Reynolds number turbulence model accounting for active or passive heat transfer)" associated with the PhD thesis of J.-F. Wald.

7.2. Bilateral Grants with Industry

PhD grant (CIFRE) of J.-F. Wald, EDF, in progress.

8. Partnerships and Cooperations

8.1. Regional Initiatives

8.1.1. Predicting pressure losses in aeronautical fuel injectors

This is a 3-year programme, funded by Conseil Régional d'Aquitaine (call 2014) and two small-size companies, MMP (Gurmençon, France) and GDTECH (Bordes, France). A one-year post-doc will be recruited beginning of 2015. The objective is to investigate the possibility of using advanced RANS or hybrid RANS-LES approaches to better predict the pressure losses in injector.

8.2. National Initiatives

8.2.1. GIS Success

We are members of the CNRS GIS Success (Groupement d'Intérêt Scientifique) organised around the two major codes employed by the Safran group, namely AVBP and Yales 2.No specific activity has been devoted around those codes during 2014.

8.3. European Initiatives

8.3.1. FP7 & H2020 Projects

Participants: Vincent Perrier [responsible of the team contribution], Pascal Bruel [substitute], Simon Delmas [PhD].

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 contribution of our team is to create with AeroSol 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.

8.4. International Initiatives

8.4.1. Informal International Partners

- Collaboration [RM] with the M. Hadziabdic (International university of Sarajevo, Bosnia and Herzegovina) on the turbulence and heat transfer modelling of jets impinging on a heated, rotating disk.
- Collaboration [RM] with the A.T. Nguyen (University of Vietnam-Ho Chi Minh City) on the development of a new hybrid RANS/LES method based on temporal filtering.
- Collaboration [RM] with E. Juntasaro (King Mongkut's University of Technology North Bangkok, Thailand) on the modelling of transition to turbulence.
- Collaboration [RM] with S. Lardeau (CD-Adapco, London, UK) on the development of an industrial version of the EB-RSM model and its implementation in the commercial CFD software STAR-CCM+.
- Collaboration [PB, VP, YM] with E. Dick (University of Ghent, Belgium) on the development of schemes for the simulation of unsteady low Mach number flows.
- Collaboration [PB] with A. Allouhi, A. Jamil, Y. Mourad (Ecole Supérieure de Technologie of Fès, Marocco) on energy issues related to transition and phase change materials.

8.5. International Research Visitors

8.5.1. Visits of International Scientists

- June 2014 (5 days): Prof. Erik Dick from Ghent University (Belgium) concerning the development of low Mach number schemes.
- July 2014 (10 days) Dr. Paulo Correia from Evora University (Portugal) concerning the possibility of cooperating with the Cagire team.

8.5.2. Visits to International Teams

- University of Calabria (Italy): [YM] and [PB] stayed during three days there and met Dr Carmine de Bartolo, Fr Alessandra Nigro and Prof. Francesco Bassi (University of Bergame) to discuss the possibility of a future cooperation.
- University of Evora (Portugal): [PB] stayed there during five days paying back his visit to Dr Correia who came to Pau in July. Dr Correia is willing to work with the Cagire team on the topic of synthetic turbulence generation.

9. Dissemination

9.1. Promoting Scientific Activities

9.1.1. Scientific events organisation

9.1.1.1. Member of the organizing committee

Member [RM] of the steering committee of the Special Interest Group "Turbulence Modelling" (SIG-15) of ERCOFTAC (European Research COmmittee for Flow, Turbulence and Combustion) that organizes a series of international workshops dedicated to cross-comparisons of the results of turbulence models and experimental/DNS databases.

9.1.2. Scientific events selection

9.1.2.1. Member of the conference program committee

- Co-chair ([RM], with M. Visbal, Air Force Research Laboratory, Ohio, USA) of the symposium on DNS, LES and hybrid RANS/LES within the international conference FEDSM (Fluid Engineering Division Summer Meeting) of the ASME (American Society of Mechanical Engineers), organized in Chicago, in August 2014.
- Member [RM] of the scientific committee of the Intl Symp. Turbulence, Heat and Mass Transfer, Sarajevo, Bosnia and Herzegovina, 2015
- Member [RM] of the scientific committee of the Intl. Symp. Engineering Turbulence Modelling and Measurement, Marbella, Spain, 2014

9.1.2.2. Reviewer

- Turbo Expo ASME Gas Turbines Conference 2015 (Montreal) [PB]

9.1.3. Journal

9.1.3.1. Reviewer

This year, the team members have reviewed 33 papers for the following journals:

- Aerospace Science and Technology [PB]
- Combustion and Flame [PB]
- Computational Thermal Science [PB]
- Computers and Fluids [RM]

- Concurrency and Computation: practice and experience [PB]
- Flow Turbulence and Combustion [RM]
- Heat Transfer Engineering [RM]
- International Journal of Heat and Fluid Flow [RM]
- International Journal of Refrigeration [PB]
- Journal of Computational Physics [VP]
- Journal of Fluid Mechanics [RM]
- Journal of Petroleum Science and Engineering [PB]
- Journal of Propulsion Power [PB]
- Proceedings of the Combustion Institute [PB]

9.2. Teaching - Supervision - Juries

9.2.1. Teaching

- Master : [RM], Turbulence Modelling, 28h, École centrale de Lille/ENSI Poitiers/ISAE-ENSMA, Poitiers, France.
- Engineering School: [RM] Industrial codes for CFD, 12h, ISAE-ENSMA, Poitiers, France.
- Continuous Training : [RM] Simulation numérique de la turbulence en LES, 3h, EUROSAE, Paris, France.

9.2.2. Supervision

- PhD in progress: Simon Delmas, Simulation d’écoulements pariétaux génériques à bas nombre de Mach pour l’amélioration du refroidissement des chambres de combustion : développement et mise en œuvre de schémas de type Galerkine discontinu adaptés, University of Pau, started January 2013, Dir.:[PB] and Co-dir.: [VP].
- PhD in progress : Nurtoleu Shakhan, Modelling and simulation of supersonic jet in crossflow, University of Almaty (Kazakhstan), started October 2013, Dir.:Altyn Naïmanova and Co-dir.:[PB] (the thesis subject has been modified mid-2014).
- PhD in progress: Jean-François Wald, Modélisation de la turbulence avec traitement adaptatif des parois prenant en compte la thermique active ou passive, started October 2013, Dir.: [RM]

9.2.3. Juries

The participation in the following thesis juries is noted ("referee" in a French doctoral thesis jury is more or less equivalent to an external opponent in an Anglo-Saxon like PhD jury):

- PhD : Guao Wang, « Simulation numérique sur des feux de nappe de kérosène de grande échelle soumis à un vent traversier avec prise en compte d’un aéronef », University of Poitiers, France, 10 January 2014, [PB, referee].
- PhD : Julien Pilet, « Analyse du comportement moteur stabilisé en windmilling par couplage des modèles thermodynamiques et simulations numériques », University of Toulouse, France, 17 January 2014, [PB, referee].

9.3. Popularization

- Opération Forum des Métiers organisée par la Zone d'Activité Pédagogique d'Oloron Sainte Marie (64) , Salle Pierre Scohy, Oloron Sainte Marie (a stand was manned by [PB] during one day with the objective of explaining the activity of researcher to an audience of schoolboys/girls and high school students).

10. Bibliography

Publications of the year

Articles in International Peer-Reviewed Journals

[1] A. ALLOUHI, T. KOUSKSOU, A. JAMIL, P. BRUEL, Y. MOURAD, Y. ZERAOULI. *Solar driven cooling systems: An updated review*, in "Renewable and Sustainable Energy Review", April 2015, vol. 44, pp. 159–181 [*DOI :* 10.1016/J.RSER.2014.12.014], https://hal.inria.fr/hal-01107607

[2] J.-P. DUMAS, S. GIBOUT, L. ZALEWSKI, K. JOHANNES, E. FRANQUET, J.-P. BÉDÉCARRATS, P. TITTELEIN, F. KUZNIK. *Interpretation of calorimetry experiments to characterise phase change materials*, in "International Journal of Thermal Sciences", April 2014, vol. 78, pp. 48-55 [*DOI :* 10.1016/J.IJTHERMALSCI.2013.11.014], https://hal.archives-ouvertes.fr/hal-01016397

[3] T. KOUSKSOU, P. BRUEL, A. JAMIL, T. EL RHAFIKI, Y. ZERAOULI. *Energy storage: Applications and challenges*, in "Solar Energy Materials and Solar Cells", 2014, vol. 120, pp. 59-90, https://hal.inria.fr/hal-00930130

[4] R. MANCEAU. *Recent progress in the development of the Elliptic Blending Reynolds-stress model*, in "International Journal of Heat and Fluid Flow", 2015, 32 p. [*DOI :* 10.1016/J.IJHEATFLUIDFLOW.2014.09.002], https://hal.inria.fr/hal-01092931

[5] R. MANCEAU, R. PERRIN, M. HADZIABDIC, S. BENHAMADOUCHE. *Investigation of the interaction of a turbulent impinging jet and a heated, rotating disk*, in "Phys. Fluids", 2014, vol. 26, n^{o} 3, ACL, https://hal.archives-ouvertes.fr/hal-01051796

[6] Y. MOGUEN, P. BRUEL, V. PERRIER, E. DICK. *Non-reflective inlet conditions for the calculation of unsteady turbulent compressible flows at low Mach number*, in "Mechanics and Industry", March 2014, vol. 15, n^{o} 3, pp. 179-189 [*DOI :* 10.1051/MECA/2014027], https://hal.inria.fr/hal-01009979

[7] Y. MOGUEN, S. DELMAS, V. PERRIER, P. BRUEL, E. DICK. *Godunov-type schemes with an inertia term for unsteady full Mach number range flow calculations*, in "Journal of Computational Physics", January 2015, vol. 281, 35 p. [*DOI :* 10.1016/J.JCP.2014.10.041], https://hal.inria.fr/hal-01096422

International Conferences with Proceedings

[8] S. DELMAS, V. PERRIER, P. BRUEL. *A stable and accurate compressible low Mach scheme for unsteady flow calculation*, in "European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2014)", Barcelona, Spain, July 2014, https://hal.inria.fr/hal-01079763

[9] S. LARDEAU, R. MANCEAU. *Computations of canonical and complex flow configurations using a robust formulation of the elliptic-blending Reynolds-Stress model*, in "Proc. 10th ERCOFTAC Int. Symp. on Eng. Turb. Modelling and Measurements", Marbella, Spain, 2014, ACT, https://hal.archives-ouvertes.fr/hal-01051799

[10] Y. MOGUEN, P. BRUEL, E. DICK. *Solving low Mach number Riemann problems by momentum interpolation*, in "6th International Conference on Advanced Computational Methods in Engineering (ACOMEN 2014)", Ghent, Belgium, June 2014, https://hal.inria.fr/hal-01079741

[11] Y. MOGUEN, S. DELLACHERIE, P. BRUEL, E. DICK. *Momentum interpolation for quasi one-dimensional unsteady low Mach number flows with acoustics*, in "European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2014)", Barcelona, Spain, July 2014, https://hal.inria.fr/hal-01079747

[12] Y. MOGUEN, S. DELMAS, V. PERRIER, P. BRUEL, E. DICK. *Inertia terms for all Mach number Godunov-type schemes: behavior of unsteady solutions at low Mach number*, in "8th International Conference on Computational Fluid Dynamics", Chengdu, China, July 2014, https://hal.inria.fr/hal-01079755

[13] J.-F. WALD, S. BENHAMADOUCHE, R. MANCEAU. *Adaptive wall treatment for the elliptic blending Reynolds stress model*, in "Proc. 10th ERCOFTAC Int. Symp. on Eng. Turb. Modelling and Measurements", Marbella, Spain, 2014, ACT, https://hal.archives-ouvertes.fr/hal-01051800

Scientific Books (or Scientific Book chapters)

[14] R. ABGRALL, H. BEAUGENDRE, P. M. CONGEDO, C. DOBRZYNSKI, V. PERRIER, M. RICCHIUTO. *High Order Nonlinear Numerical Schemes for Evolutionary PDEs*, Lecture Notes in Computational Science and Engineering, Springer, January 2014, vol. 99, https://hal.inria.fr/hal-01092349

Other Publications

[15] R. MANCEAU. *La simulation numérique de la turbulence en LES*, 2014, Formation continue EUROSAE: La simulation numérique en mécanique des fluides compressibles, https://hal.inria.fr/hal-01092935

References in notes

[16] 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° 4-5, pp. 507-540

[17] 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° 2, pp. 267–279, http://dx.doi.org/10.1006/jcph.1996.5572

[18] 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° 1-3, pp. 409-414

[19] 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° 190, pp. 545–581, http://dx.doi.org/10.2307/2008501

[20] 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, pp. 90–113

[21] 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, pp. 411–435, http://dx.doi.org/10.2307/2008474

[22] 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, pp. 337–361

[23] 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, pp. 199–224, http://dx.doi.org/10.1006/jcph.1998.5892

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

[25] 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, pp. 208–221, http://dx.doi.org/10.1016/j.jcp.2007.01.035

[26] J.-L. FLORENCIANO. *Étude de la réponse d'un écoulement avec transfert pariétal de masse à un forçage acoustique*, Pau University, 2013

[27] 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, pp. 2853-2957 [*DOI :* DOI: 10.1016/S0017-9310(02)00027-3]

[28] 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, pp. 1–20

[29] 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, pp. 141–162

[30] 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, pp. 107–129, http://dx.doi.org/10.2307/2008684

[31] 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, pp. 145-156

[32] 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, n^o 33, pp. 89–123

[33] 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, pp. 5649–5670, http://dx.doi.org/10.1016/j.jcp.2008.02.015

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

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

[36] 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

[37] 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, pp. 872-877

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

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

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