Activity Report 2015

Team CAGIRE

Computational Approximation with discontinous Galerkin methods and compaRison with Experiments

Inria teams are typically groups of researchers working on the definition of a common project, and objectives, with the goal to arrive at the creation of a project-team. Such project-teams may include other partners (universities or research institutions).

RESEARCH CENTER
Bordeaux - Sud-Ouest

THEME
Numerical schemes and simulations

Table of contents

Team CAGIRE

Creation of the Team: 2011 June 01

Keywords:

Computer Science and Digital Science:
6.1.1. - Continuous Modeling (PDE, ODE)
6.2.1. - Numerical analysis of PDE and ODE
6.2.7. - High performance computing

Other Research Topics and Application Domains:
4.1.2. - Nuclear energy
5.2.3. - Aviation
5.2.4. - Aerospace

1. Members

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

Faculty Member
Jonathan Jung [Univ. Pau, Associate Professor, from Sep 2015]

Engineer
Benjamin Lux [Inria, until Oct 2016]

PhD Students
Simon Delmas [Univ. Pau, until January 2016]
Jean-François Wald [EDF, until April 2016]

Post-Doctoral Fellow
Yann Moguen [Univ. Pau, until May 2016]

Administrative Assistant
Nicolas Jahier [Inria]

2. Overall Objectives

2.1. Turbulent flows with complex interactions

This interdisciplinary project brings together researchers coming from different horizons and backgrounds (applied mathematics and fluid mechanics) who progressively elaborated a common vision of what should be the simulation tool of fluid dynamics of tomorrow. Our team focuses on wall bounded turbulent flows featuring complex phenomena such as aeroacoustics, hydrodynamic instabilities, wall roughness, buoyancy. Because such flows are exhibiting a multiplicity of time and scale fluctuations resulting from complex interactions, their simulation is extremely challenging. Even if various methods of simulation (DNS [1]) and turbulence modeling (RANS [2], LES [3], hybrid RANS-LES) are available and have been significantly improved over time, none

---

[1] Direct numerical simulation
[2] Reynolds averaged Navier-Stokes
[3] Large-eddy simulation

of them does satisfy all the needs encountered in industrial and environmental configurations. We consider that all these methods will be useful in the future in different situations or regions of the flow if combined in the same simulation in order to benefit from their respective advantages wherever relevant, while mutually compensating their known limitations. It will thus lead to a description of turbulence at widely varying scales in the computational domain, hence the name *multi-scale simulations*. For example, the RANS mode may extend throughout regions where turbulence is sufficiently close to equilibrium leaving to LES or DNS the handling of regions where large scale coherent structures are present. However, a considerable body of work is required to:

- Establish the behavior of the different types of turbulence modeling approaches when combined with high order discretization methods.
- Elaborate relevant and robust switching criteria between models, similar to error assessments used in automatic mesh refinement, but based on the physics of the flow in order to adapt on the fly the scale of resolution from one extreme of the spectrum to another (say from the Kolmogorov scale to the geometrical large scale, i.e., from DNS to RANS).
- Ensure a high level of accuracy and robustness of the resulting simulation tool to address a large range of flow configurations, i.e., from a generic lab scale geometry for validation to practical systems of interest of our industrial partners.

But the best multi-scale modeling and high order discretization methods are useless without the recourse to high performance computing (HPC) to bring the simulation time down to values compatible with the requirement of the end users. So, a significant part of our activity is devoted to the proper handling of the constantly evolving supercomputer architectures. The long-term objective of this project is to develop, validate, promote and transfer an original and effective approach for modeling and simulating generic flows representative of flow configurations encountered in the field of energy production and aeronautical propulsion. Our approach will be combining mesh (h) + turbulence model (m) + discretization order (p) adaptivity. This will be achieved by:

- Contributing to the development of new turbulence models.
- Improving high order numerical methods, and increasing their efficiency in the current High Performance Computing context.
- Developing experimental tools.

In that framework, in 2015, the team members developed their activity around the following axes:

- The development of the AeroSol library and the direct numerical simulations of single jets in crossflow including that experimentally studied on the team test facility MAVERIC.
- The development of low Mach schemes.
- The development of advanced turbulence RANS and hybrid RANS-LES turbulence models adapted to zero Mach flows with a specific emphasis on the wall-flow interaction.

3. Research Program

3.1. Computational fluid mechanics: modeling or not before discretizing ?

A typical continuous solution of the Navier Stokes equations at sufficiently high values of the Reynolds number is governed by a spectrum of time and space scales fluctuations closely connected with the turbulent nature of the flow. The term deterministic chaos employed by Frisch in its enlightening book [33] is certainly conveying most adequately the difficulty in analyzing and simulating this kind of flows. The broadness of the turbulence spectrum is directly controlled by the Reynolds number defined as the ratio between the inertial forces and the viscous forces. This number is not only useful to determine the transition from a laminar to a turbulent flow regime, it also indicates the range of scales of fluctuations that are present in the flow under consideration.

Typically, for the velocity field and far from solid walls, 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 play a crucial role in the dynamics of the largest ones which implies that an accurate framework for the computation of turbulent flows must take into account all these scales.

Thus, the usual practice to deal with turbulent flows is to choose between an a priori modeling (in most situations) or not (low Re number and rather simple configurations) before proceeding to the discretization step followed by the simulation runs themselves. If a modeling phase is on the agenda, then one has to choose again among the above mentioned variety of approaches.

As it is illustrated in Fig. 1, this can be achieved either by directly solving the Navier-Stokes equations (DNS) or by first applying a statistical averaging (RANS) or a spatial filtering operator to the Navier-Stokes equations (LES). The new terms brought about by the filtering operator have to be modeled. From a computational point of view, the RANS approach is the least demanding, which explains why historically it has been the workhorse in both the academic and the industrial sectors. It has permitted quite a substantial progress in the understanding of various phenomena such as turbulent combustion or heat transfer. Its inherent inability to provide a time-dependent information has led to promote in the last decade the recourse to either LES or DNS to supplement if not replace RANS. By simulating the large scale structures while modeling 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. At the same time, DNS was progressively applied to geometries of increasing complexity (channel flows with values of Re_τ multiplied by 10 during the last 15 years, jets, turbulent premixed flames, among many others), and proved to be a formidable tool that permits **(i)** to improve our knowledge on turbulent flows and **(ii)** to test (i.e., validate or invalidate) and improve the modeling 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, the high computational cost of LES or DNS makes necessary the use of highly accurate numerical schemes in order to optimize the use of computational resources.

To the noticeable exception of the hybrid RANS-LES modeling, which is not yet accepted as a reliable tool for industrial design, as mentioned in the preamble of the Go4hybrid European program [4], once chosen, a single turbulence model will (try to) do the job for modeling the whole flow. Thus, depending on its intrinsic strengths and weaknesses, the accuracy will be a rather volatile quantity strongly dependent on the flow configuration. The turbulence modeling and industrial design communities waver between the desire to continue to rely on the RANS approach, which is unrivaled in terms of computational cost, but is still not able to accurately represent all the complex phenomena; and the temptation to switch to LES, which outperforms RANS in many situations but is prohibitively expensive in high-Reynolds number wall-bounded flows. In order to account for the deficiencies of both approaches and to combine them for significantly improving the overall quality of the modeling, the hybrid RANS-LES approach has emerged during the last decade as a viable, intermediate way, and we are definitely inscribing our project in this innovative field of research, with an original approach though, connected with a time filtered hybrid RANS-LES and a systematic and progressive validation process against experimental data produced by the team.

3.2. Computational fluid mechanics: high order discretization on unstructured meshes and efficient methods of solution

All the methods considered in the project are mesh-based methods: the computational domain is divided into cells, that have an elementary shape: triangles and quadrangles in two dimensions, and tetrahedra, hexahedra, pyramids, and prisms 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.

---

[4] http://www.transport-research.info/web/projects/project_details.cfm?id=46810

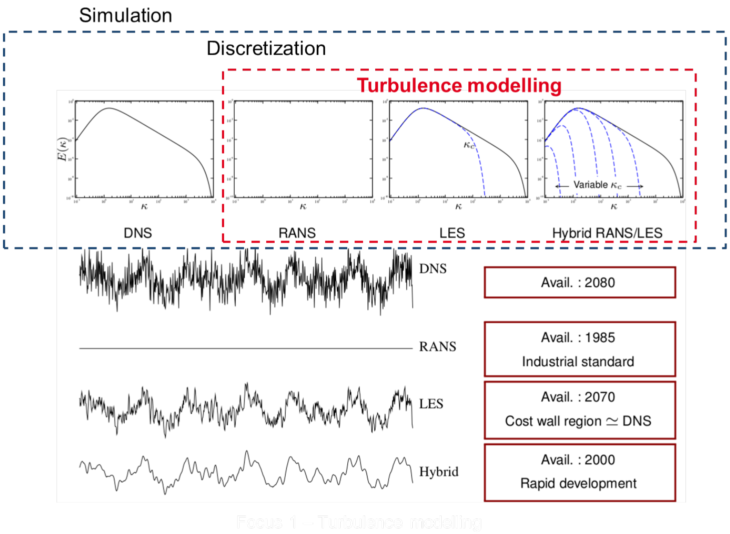

Figure 1. A schematic view of the different nested steps for turbulent flows simulation: from DNS to hybrid RANS-LES. The approximate dates at which the different approaches are or will be routinely used in the industry are indicated in the boxes on the right (extrapolations based on the present rate of increase in computer performances).

The AeroSol library developed in the team is based on discontinuous Galerkin methods. These methods were introduced by Reed and Hill [38] and first studied by Lesaint and Raviart [36]. 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 element nature of the scheme were progressively solved, for scalar conservation laws [29], [28], one dimensional systems [27], multidimensional scalar conservation laws [26], and multidimensional systems [30]. For the same system, we can also cite the work of [32], [35], which is slightly different: the stabilization is made by adding a nonlinear term, and the time integration is implicit.

Contrary to continuous Galerkin methods, the discretization of diffusive operators is not straightforward. This is due to the discontinuous approximation space, which does not fit well with the space function in which the diffusive system is well posed. A first stabilization was proposed by Arnold [20]. The first application of discontinuous Galerkin methods to Navier-Stokes equations was proposed in [24] by mean of a mixed formulation. Actually, this first attempt led to a non compact computation stencil, and was later proved to be not stable. A compactness improvement was made in [25], which was later analyzed, and proved to be stable in a more unified framework [21]. The combination with the $k - \omega$ RANS model was made in [23]. As far as Navier Stokes equations are concerned, we can also cite the work of [34], in which the stabilization is closer to the one of [21], the work of [37] on local time stepping, or the first use of discontinuous Galerkin methods for direct numerical simulation of a turbulent channel flow done in [31]. Discontinuous Galerkin methods are so popular because

- They can be developed for any order of approximation.
- The computational stencil of one given cell is limited to the cells with which it has a common face. This stencil does not depend on the order of approximation. This is a pro, compared for example with high order finite volumes, which require as more and more neighbors as the order increases.
- They can be developed for any kind of mesh, structured, unstructured, but also for aggregated grids

[22]. This is a pro compared not only with finite differences schemes, which can be developed only on structured meshes, but also compared with continuous finite elements methods, for which the definition of the approximation basis is not clear on aggregated elements.

- p-adaptivity is easier than with continuous finite elements, because neighboring elements having a different order are only weakly coupled.
- Upwinding is as natural as for finite volumes methods, which is a benefit for hyperbolic problems.
- As the formulation is weak, boundary conditions are naturally weakly formulated. This is a benefit compared with strong formulations, for example point centered formulation when a point is at the intersection of two kinds of boundary conditions.

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 many things to be improved, which include: efficient shock capturing methods for supersonic flows, high order discretization of curved boundaries, low Mach number behavior of these schemes and combination with second-moment RANS models. Another drawback of the discontinuous Galerkin methods is that they can be computationally costly, due to the accurate representation of the solution calling for a particular care of implementation for being efficient. We believe that this cost can be balanced by the strong memory locality of the method, which is an asset for porting on emerging many-core architectures.

3.3. Flow analysis and CFD assessment: experimental aspects

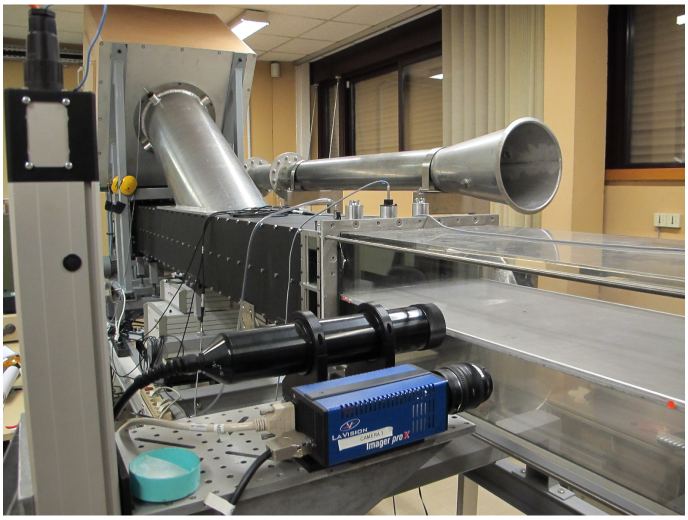

Figure 2. View of the team's test facility MAVERIC.

With the considerable and constant development of computer performance, many people were thinking at the turn of the 21st century that in the short term, CFD would replace experiments considered as too costly and not flexible enough. Simply flipping through scientific journals such as Journal of Fluid Mechanics, Combustion of Flame, Physics of Fluids or Journal of Computational Physics or through websites such that of Ercoftac [5] is sufficient to convince oneself that the recourse to experiments to provide either a quantitative description of complex phenomena or reference values for the assessment of the predictive capabilities of the physical modeling and of the related simulations is still necessary. The major change that can be noted though concerns

---

[5] http://www.ercoftac.org

the content of the interaction between experiments and CFD (understood in the broad sense). Indeed, LES or DNS assessment calls for the experimental determination of time and space turbulent scales as well as time resolved measurements and determination of single or multi-point statistical properties of the velocity field. Thus, the team methodology incorporates from the very beginning an experimental component that is in strong interaction with the physical modeling and the simulation activities. 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, LES, RANS and Hybrid RANS-LES models that the team is developing. 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. Thus, this test facility called MAVERIC (Fig. 2) is extensively used in the framework of the present project to investigate a 1-hole cylindrical inclined jet interacting with a turbulent crossflow. PIV [6] and LDV [7] are the workhorses as far as metrology is concerned.

4. Application Domains

4.1. Aeronautical combustion chambers

The combustion chamber of aeronautical engines is the system of practical interest we are interested in as far as propulsion devices are concerned. The MAVERIC test facility was developed by P. Bruel in that framework during the theses (CIFRE Turbomeca) of A. Most (2007) and J.-L. Florenciano (2013). The initial objective was to reproduce experimentally a simplified flow configuration (jet(s) in crossflow) representative of that encountered at the level of the effusion cooled aeronautical combustion chambers walls. The experimental data were used by Safran/Turbomeca to assess the predictive capability of LES simulations during our joint participation in the EU-FP7 KIAI program (2009-2013). Concerning DNS, the jet in crossflow configurations of our AeroSol based simulations which represent our contribution to the EU IMPACT-AE program (2011-2016) were chosen in partnership with Turbomeca who is leading the corresponding work package. Last but not least, tests aimed at demonstrating the feasibility of characterizing in situ by PIV the velocity field of flows emerging from different kinds of fuel nozzles were carried out at the Turbomeca premises in 2012 and 2013. Although our main present industrial partners are large companies, we are and will be actively targeting much smaller companies (SMEs) especially in the southwest part of France. In that respect, the partnership we just started with AD Industries which is manufacturing fuel nozzles as well as combustion chambers for business jet engines is emblematic of our involvement in such kind of partnership.

4.2. Power stations

The cooling of key components of power stations in case of emergency stops is a critical issue. R. Manceau has established a long term collaboration (4 PhD thesis) with the R & D center of EDF of Chatou, for the development of refined turbulence models in the in-house CFD code of EDF, Code_Saturne, in order to improve the physical description of the complex interaction phenomena involved in such applications. In the framework of the co-supervision of the PhD thesis (CIFRE EDF) of J.-F. Wald, strategies are developed to adapt the EB-RSM turbulence model to a local modification of the scale of description of the flow in the near-wall region: refined scale (fine mesh in the near-wall region) or coarse scale (with wall functions). Indeed, the complexity of the industrial geometries is such that a fine mesh along solid boundaries in the whole system is usually not possible/desirable.

5. Highlights of the Year

5.1. Highlights of the Year

First DNS simulation of a turbulent flow with AeroSol

---

[6] Particle image velocimetry

[7] Laser Doppler velocimetry

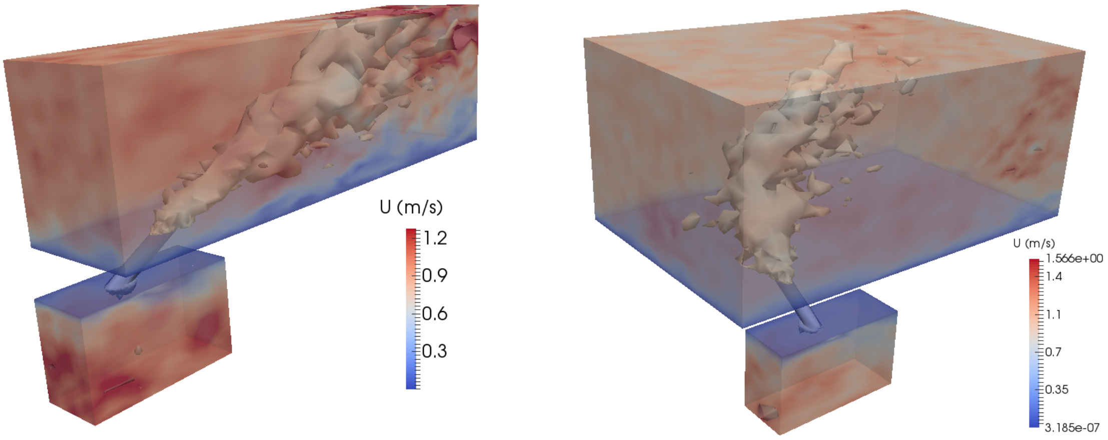

Figure 3. DNS of jets in crossflow (AeroSol-DG2): Examples of snapshots of instantaneous surfaces of the velocity norm. Left - With a 0-degree jet skidding (MAVERIC configuration). Right - With a 90-degree jet skidding.

In 2015, the first DNS of the configuration of a jet in turbulent crossflow have been carried out with the AeroSol library. Qualitativeley speaking, this represents the completion of the initial objective that the team was targeting in 2011 when it was created ! These computations were done within the IMPACT-AE project. The runs were using 1024 cores of the BlueGene /Q cluster Turing at IDRIS thanks to a 4400000-hour computing grant obtained in 2015. Examples of results obtained for the two flow configurations considered are presented in Fig. 3.

Implementation of the EB-RSM model into StarCCM+

In close collaboration with the R&D team of Adapco, the company that develops and sells the commercial CFD package StarCCM+, the EB-RSM model has been implemented in this code, starting from release 10.02. This constitutes a significant achievement that our models are made widely available to the engineering community.

6. New Software and Platforms

6.1. AeroSol

Participants: Simon Delmas [Cagire], Benjamin Lux [Cagire], Nikolaos Pattakos [Cardamom], Vincent Perrier [Cagire, correspondent], Mario Ricchiuto [Cardamom].

Developed since 2011 by V. Perrier in partnership with the Cardamom Inria team, the AeroSol library 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 element methods on hybrid and possibly curvilinear meshes.

The work of the Cardamom team is focused on continuous finite element methods, while we focus on discontinuous Galerkin methods. However, everything is done for sharing the largest possible part of code. The distribution of the unknowns is made with the software PaMPA, first developed within the Inria teams Bacchus and Castor, and currently maintained in the Tadaam team.

The generic features of the library are

- **High order**. It can be theoretically any order of accuracy, but the finite element basis, and quadrature formula are implemented for having up to a fifth order of accuracy.
- **Hybrid and curvilinear meshes**. AeroSol can deal with up to fifth order conformal meshes composed of lines, triangles, quadrangles, tetrahedra, hexahedra, prism, and pyramids.
- **Continuous and discontinuous discretization**. AeroSol deals with both continuous and discontinuous finite element methods.

We would like to emphasize three assets of this library:

- **Its development environment** For allowing a good collaborative work and a functional library, a strong emphasis has been put on the use of modern collaborative tools for developing our software. This includes the active use of a repository, the use of CMake for the compilation, the constant development of unitary and functional tests for all the parts of the library (using CTest), and the use of the continuous integration tool Jenkins for testing the different configurations of AeroSol and its dependencies. Efficiency is regularly tested with direct interfacing with the PAPI library or with tools like scalasca.
- **Its genericity** A lot of classes are common to all the discretization, for example classes concerning I/O, finite element functions, quadrature, geometry, time integration, linear solver, models and interface with PaMPA. Adding simple features (e.g. models, numerical flux, finite element basis or quadrature formula) can be easily done by writing the class, and declaring its use in only one class of the code.
- **Its efficiency** This modularity is achieved by means of template abstraction for keeping good performances. Dedicated efficient implementation, based on the data locality of the discontinuous Galerkin method has been developed. As far as parallelism is concerned, we use point-to-point communications, the HDF5 library for parallel I/O. The behavior of the AeroSol library at medium scale (1000 to 2000 cores) was studied in [19].

The AeroSol project fits with the first axis of the Bordeaux Sud Ouest development strategy, which is to build a coherent software suite scalable and efficient on new architectures, as the AeroSol library relies on several tools developed in other Inria teams, especially for the management of the parallel aspects.

At the end of 2014, 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. 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.
- **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. Handling of high order meshes.
- **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 Newton method for stationary problems. Implicit unstationary time iterator non consistent in time for stationary problems. Implementation of in house GMRES and conjugate gradient based on Jacobian free iterations.
- **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. Scalar advection-diffusion model.
- **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. 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** Centered fluxes, exact Godunov' flux for linear hyperbolic systems, and Lax-Friedrich flux. Riemann solvers for Low Mach flows. Numerical flux accurate for steady and unsteady computations.
- **Boundary conditions** Periodic boundary conditions, time-dependent inlet and outlet boundary conditions. Adiabatic wall and isothermal wall. Steger-Warming based boundary condition.
- **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. Tier-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. 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.

In 2015, N. Pattakos was hired in the team Cardamom, in order to improve the code architecture and for easing the installation of the library. The following features were added or improved:

- **Development environment** The use of CMake was strongly improved, which induced also easier test launching. Documentation, code cleaning and refactorization have also been led. The shared project of Plafrim was updated, and so was the joint Aerosol/Scotch/PaMPA project on the continuous integration platform. Integration of SPack for handling dependencies has begun. Interface with ESSL was fixed.
- **Multigrid** Development of p-multigrid methods. This includes also the possibility of beginning a computation iwth an order and to decrease or increase the order of approximation when restarting. For the p multigrid methods, V and W cycle have been developed, and restriction and prolongation opertors have also been developed. Implementation of h-multigrid has started, with the development of tests of the aggregation methods of PaMPA, and the definition of finite element basis on arbitrary cells.

- **Boundary conditions** Development of the Synthetic Eddy Method boundary condition.
- **Models** Linearized Euler equations, and Sutherland model for non isothermal diffusive flows. Shallow-water model.
- **Parallel computing** Weighted load balancing for hybrid meshes.
- **Validation** Turbulent channel flow.
- **Postprocessing** Development of high order projections over line postprocessing, possibility of stocking averaged data, such as the average flow and the Reynolds stresses.

7. New Results

7.1. DNS of a jet in crossflow

One main achievement of this year is to have done our first DNS computations at third order with the Aerosol software. Two configurations of jet in cross flow have been computed: one with a hole direction aligned with the main flow direction (Fig. 3-left), and another one with a 90-degree jet skidding (Fig. 3-right). The first case has been validated by using analytical models of jet trajectory, and has also been compared with experiments made with our experimental bench MAVERIC. The comparison of experiments and DNS showed a good agreement.

The DNS database includes:

- The instantaneous flow at the vertices of the mesh.
- the instantaneous flow at some probes.
- The mean flow.
- The value of the Reynolds stress tensor in all the degrees of freedom.

7.2. Extension of the momentum interpolation method to low Mach Rieman problems

In a previous study [9], the momentum interpolation (MI) method was considered as a guideline to develop a Godunov-like flux scheme called AUSM-IT and able to preserve the acoustic energy at the discrete level for a low-order finite volume approach. This year, the MI method has been successfully extended to the case of low Mach flows featuring discontinuities [8]. The undesirable dispersive effect directly connected to the upwinding of the MI formulation of the face velocity has been corrected (up to second-order errors) by using a central interpolation of momentum in the face velocity definition.

7.3. Main features of highly underexpanded jets

Despite the numerous studies dealing with underexpanded jets, many aspects of their structure were not clearly described, particularly when one seeks for quantitative predictions. Since such flow configuration may be of interest in case of the accidental boring of an aeronautical combustion chamber, an exhaustive review of the main experimental papers dealing with underexpanded jets has been carried out [5]. This study aimed at clarifying the characteristics which were well known, from those where there is clearly a lack of confidence. Curiously enough, such a work has never been done and no exhaustive review was available on such a topic.

7.4. Formulation of a reference EB-RSM model

The Elliptic Blending Reynolds Stress Model (EB-RSM), originally proposed by Manceau & Hanjalic in 2002, has been subject to various modifications by several authors during the last decade, mainly for numerical robustness reasons. We have revisited all these modifications from the theoretical standpoint and investigated in detail their influence on the reproduction of the physical mechanisms at the origin of the influence of the wall on turbulence. Theoretical arguments and comparison with DNS results led to the selection of a recommended formulation for the EB-RSM model [7].

7.5. Development of a new enrichment method

A complex issue in multi-scale simulations is the necessity to *enrich* the solution at the interface between a region described at coarse grain (e.g., using RANS) and a region described at fine grain (e.g., using LES). In order to rapidly generate realistic fluctuations at the beginning of the LES region, we have proposed [4] a method of volumic forcing, the so-called ALF (Anisotropic Linear Forcing). In an overlap region, a time-dependent volume force is introducted into the filtered equations of motion in order to amplify the turbulent fluctuations in order that the LES field satisfy the statistics of the RANS solution, a method that proved simple, efficient and computationnaly cheap.

7.6. A new criterion to analyse hybrid RANS/LES approaches

Most of the available hybrid RANS/LES methods are completely empirical or based on a formalism which is not applicable in practical application, due to a mismatch between the statistical average and the spatial filetring in inhomogeneous flows. The lack of clear formalism leads to limitations in terms of modeling of the unresolved turbulent motion. We have established a criterion [6] to assess the equivalence between hybrid RANS/LES methods, called *H-equivalence*, that makes it possible to view different hybrid methods as models for the same system of equations: as a consequence, empirical hybrid methods, such as the detached-eddy simulation (DES), can be interpreted as a model for the subfilter stress involved in the *temporally filtered* Navier-Stokes equations, which is an answer to the issue raised above about the formalism underlying such methods.

8. Bilateral Contracts and Grants with Industry

8.1. 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.

8.2. Bilateral Grants with Industry

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

9. Partnerships and Cooperations

9.1. Regional Initiatives

9.1.1. 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, AD Industrie (Gurmençon, France) and GDTECH (Bordes, France). A one-year post-doc [YM] started in May 2015. The objective is to investigate the possibility of using advanced RANS or hybrid RANS-LES approaches to better predict the pressure losses in aeronautical fuel nozzles.

9.2. National Initiatives

9.2.1. 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. Apart our participation in the annual meeting of the GIS technical comittee, no specific technical activity has been devoted around those codes during 2015.

9.3. European Initiatives

9.3.1. 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 - 31/05/2016

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 of Science, Technology and Medecine, Loughborough University.

Abstract: The environmental benefits of low emission lean burn technology in reducing NOx emissions up to 80% will only 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 emission 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.

9.4. International Initiatives

- April-June 2015: A. Javadi (PhD student) from Chalmers University, Gothenburg, Sweden (3 months).

9.4.1. Informal International Partners

- 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) related to solar driven cooling systems.
- Collaboration [PB] with A. Beketaeva and A. Naïmanova (Institute of Mathematics, Almaty, Kazakhstan) related to the simulation of supersonic flows.
- Collaboration [RM] with H. Nilsson and A. Javadi (University of Chalmers, Gothenburg, Sweden) on the development of RANS and hybrid RANS/LES for the turbomachinery computations.
- Collaboration [RM] with E. Juntasaro (King Mongkut’s TU, Bangkok, Thailand) about the modeling of bypass transition.

- Collaboration [RM] with Tran Thanh Tinh and Anh Thi NGuyen (TU Ho Chi Minh City, Viet Nam) on temporal hybrid RANS/LES.

9.5. International Research Visitors

9.5.1. Visits of International Scientists

- April-June 2015: A. Javadi (PhD student) from Chalmers University, Gothenburg, Sweden (3 months).
- November 2015: Prof. Erik Dick from Ghent University (Belgium) (4 days).
- November 2015: Dr. A. Naïmanova from the Institute of Mathematics (Ministry of Education), Almaty, Kazakhstan (4 weeks).
- November-December 2015: N. Shakhan (PhD student) from Al Farabi University, Almaty, Kazakhstan (7 weeks).

10. Dissemination

10.1. Promoting Scientific Activities

10.1.1. Scientific events organisation

10.1.1.1. Member of the organizing committees

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.

10.1.2. Scientific events selection

10.1.2.1. Member of the conference program committees

- Member [RM] of the scientific committee of the Intl Symp. Turbulence, Heat and Mass Transfer, Sarajevo, Bosnia and Herzegovina, 2015

10.1.2.2. Reviewer

This year, the team members have reviewed (12) contributions to the following conferences:

ECOS 2015 (Pau, France) (1) [PB] ASME GT Turbo Expo 2015 (Montréal, Canada) (2) [PB] ASME-GT Turbo Expo 2016 (Séoul, South Korea) (2) [PB] THMT-2015 (Sarajevo, Bosnia-Herzegovina) (5) [RM] NURETH (The Haag, The Netherlands) (2) [RM]

10.1.3. Journal

10.1.3.1. Reviewer - Reviewing activities

This year, the team members have reviewed (29) papers for the following journals:

- Aerospace Science and Technology (6) [PB]
- Combustion and Flame (5) [PB]
- Computers & Fluids (1) [VP]
- International Journal of Fluid Mechanics Research (2) [PB]
- International Communications in Heat and Mass Transfer (1)[YM]
- International Journal of Sustainable Aviation (1) [PB]
- Journal of Computational and Applied Mathematics (1) [YM]
- Journal of Computational Physics (2) [VP]
- Journal of Petroleum Science and Engineering (2) [PB]
- Journal of the Taiwan Institute of Chemical Engineers (1) [PB]
- International Journal of Heat and Fluid Flow (3) [RM]
- Flow Turbulence and Combustion (2) [RM]
- Journal of Fluid Mechanics (1) [RM]
- Heat Transfer Engineering (1) [RM]

10.1.4. Invited talks

- “A brief overview of Inria Cagire team activity”, CTA/ITA/IAE, Sao José dos Campos, Brazil, 26 November 2015. [PB]

10.1.5. Scientific expertise

V. Perrier is an expert for research for the "Région Île de France".

10.1.6. Research administration

V. Perrier is a member of the evaluation committee, which is in charge of assessing the calibre of research conducted at Inria and guaranteeing the quality of its hiring and internal promotions.

V. Perrier participated to the hiring committees for Young Graduate Scientists (CR2) in Inria Bordeaux and Inria Saclay. He participated also to the hiring committee for an assistant professor in Pau.

V. Perrier is member of the health, safety and working conditions comittee, in charge of watching the prevention in the Bordeaux Sud Ouest center.

V. Perrier is appointed member of the Scientific Applications committee of Pau University in charge of developing the scientific computing and high performance computing policy within Pau University.

V. Perrier is an elected member of the Mathematics and Application experts committee in Pau university, in charge of hiring the non permanent teachers, of forming the hiring comittees for assistant professors, and of ranking the proposition of invited professors. He was elected vice-chair of this comittee in 2015.

V. Perrier is the scientific responsible for the website of the Mathematics department in Pau.

10.2. Teaching - Supervision - Juries

10.2.1. 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.
- Master : [PB] “Fluid mechanics: Mach zero flows vs low Mach number flows”, 30h, M2, Al Faraby University, Almaty, Kazakhstan.
- Master : [PB], "An introduction to the numerical simulation of reacting flows", 15h, M2, ISAE-SupAéro, Toulouse, France.
- Licence : [JJ], "Sequences and functions of one variable", 48h45, L1 - Geo-sciences, Université de Pau et des Pays de l’Adour, Pau, France.
- Licence : [JJ], "Mathematics for the geo-sciences 2", 19h30, L2 - Chemistry, Université de Pau et des Pays de l’Adour, Pau, France.
- Licence : [JJ], "Sequences and series", 19h30, L2 - MIASHS, Université de Pau et des Pays de l’Adour, Pau, France.

10.2.2. Supervision

- PhD : Simon Delmas, Simulation numérique directe d’un jet en écoulement transverse à bas nombre de Mach en vue de l’amélioration du refroidissement par effusion des chambres de combustion aéronautiques, 16 December 2015, Sup.:[PB] and Co-sup.: [VP].
- 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, Sup.: [RM]

- PhD in progress : Nurtoleu Shakhan, Modelling and simulation of supersonic jet in crossflow, University of Almaty (Kazakhstan), started October 2013, Sup.:A. Naïmanova and Co-Sup.:[PB] (the thesis subject has been modified mid-2014).
- Young Engineer: Benjamin Lux, Implementation of h-p multigrid in Aerosol, Sup.: [VP]

10.2.3. 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 : G. Sempionato "Numerical study of premixed stratified flame using the b-theta flame wrinkling model with extinction limit", National Institute of Aerospace Research (INPE), Sao José dos Campos, Brazil, 25 november 2015, Sup.: W.M.C. Dourado. [PB]
- PhD : D. Lahbib « Modélisation aérodynamique et thermique des multiperforations en LES », University of Montpellier-2, France, 17 December 2015. Supervisor and co-supervisor: F. Nicoud and A. Dauptain.[PB, referee]
- PhD : A. Ghani « Simulation aux grandes échelles des instabilités de combustion transverses pour des flammes parfaitement prémélangées et swirlées diphasiques ». University of Toulouse, France, 17 September 2015. Supervisor: L. Gicquel. [PB]
- PhD : C. Koupper “Unsteady multi-component simulations dedicated to the impact of the combustion chamber on the turbine of aeronautical gas turbines”, Université de Toulouse, France, 11 May 2015 (Rapporteur). Supervisor and co-supervisor: L. Gicquel and P. Duchaine. [PB, referee]
- PhD : N. Petrova “Turbulence-chemistry interaction models for numerical simulation of aeronautical propulsion systems”, École Polytechnique, Palaiseau, France, 16 January 2015. Supervisor: V.A. Sabel’nikov. [PB, referee]

10.3. Popularization

- « Simulation d’écoulements turbulents : retour d’expérience de partenariats de recherche », Meeting "Nature & Technology" organized by the "Conseil départemental des Pyrénées Atlantiques", devoted to « La recherche scientifique au service de l’aéronautique », Parlement de Navarre, Pau, France, 12 November 2015. [PB]
- « Carrefour des Métiers» organized by the "Zone d’Activité Pédagogique d’Orthez", gymnase Blazy, Mourenx(64), France, 4 April 2015 (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).

11. Bibliography

Publications of the year

Doctoral Dissertations and Habilitation Theses

[1] S. Delmas. *Direct numerical simulation of a jet in crossflow at low Mach number in order to improve effusion cooling for combustion chambers*, Université de Pau et des pays de l’Adour, December 2015, https://hal.archives-ouvertes.fr/tel-01256238

Articles in International Peer-Reviewed Journals

[2] 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

[3] A. Beketaeva, P. Bruel, A. Naïmanova. *Vortical structures behind a transverse jet in a supersonic flow at high jet to crossflow pressure ratios*, in "Journal of Applied Mechanics and Technical Physics", December 2015, vol. 56, n^{o} 5, 12 p. , https://hal.inria.fr/hal-01253886

[4] B. De Laage De Meux, B. Audebert, R. Manceau, R. Perrin. *Anisotropic linear forcing for synthetic turbulence generation in large eddy simulation and hybrid RANS/LES modeling*, in "Physics of Fluids", 2015, vol. 27, 35 p. [*DOI :* 10.1063/1.4916019], https://hal.inria.fr/hal-01246100

[5] E. FRANQUET, V. Perrier, S. Gibout, P. Bruel. *Free underexpanded jets in a quiescent medium: A review*, in "Progress in Aerospace Sciences", August 2015, vol. 77, 29 p. [*DOI :* 10.1016/j.paerosci.2015.06.006], https://hal.inria.fr/hal-01247078

[6] C. Friess, R. Manceau, T. Gatski. *Toward an equivalence criterion for Hybrid RANS/LES methods*, in "Computers and Fluids", 2015, vol. 122, 14 p. [*DOI :* 10.1016/j.compfluid.2015.08.010], https://hal.inria.fr/hal-01246130

[7] 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

[8] Y. Moguen, P. Bruel, E. Dick. *Solving low Mach number Riemann problems by a momentum interpolation method*, in "Journal of Computational Physics", October 2015, vol. 298, 6 p. [*DOI :* 10.1016/j.jcp.2015.06.037], https://hal.inria.fr/hal-01247086

[9] 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

Invited Conferences

[10] J. Jung. *A low Mach correction for the Godunov scheme applied to the linear wave equation with porosity*, in "Low velocity flows", Paris, France, November 2015, https://hal.inria.fr/hal-01256455

[11] V. Perrier. *Discontinuous Galerkin methods for aerodynamic applications*, in "Méthode de Galerkin discontinue et ses applications", Paris, France, June 2015, https://hal.inria.fr/hal-01256443

International Conferences with Proceedings

[12] S. Benhamadouche, F. Dehoux, R. Manceau. *An elliptic blending differential flux model for natural, mixed and forced convection*, in "8th Int. Symp. Turbulence, Heat and Mass Transfer, Sarajevo, Bosnia-Herzegovina", Sarajevo, Bosnia and Herzegovina, 2015, https://hal.inria.fr/hal-01246134

[13] R. Manceau. *Investigation of rotating flows with separation using the elliptic-blending Reynolds-stress Model*, in "9th Int. Symp. Turb. Shear Flow Phenomena, Melbourne, Australi", Melbourne, Australia, 2015, https://hal.inria.fr/hal-01246146

[14] J.-F. Wald, S. Benhamadouche, R. Manceau. *Adaptive wall treatment for the elliptic blending Reynolds stress model*, in "36th IAHR World Congress, The Hague, the Netherlands", The Hague, Netherlands, 2015, https://hal.inria.fr/hal-01246140

Conferences without Proceedings

[15] S. Delmas, V. Perrier, P. Bruel. *Behaviour of discontinuous Galerkin methods for steady and unsteady compressible flow in the low Mach regime*, in "European Workshop on High Order Nonlinear Numerical Methods for Evolutionary PDEs: Theory and Applications (HONOM)", Trento, Italy, March 2015, https://hal.inria.fr/hal-01256440

Other Publications

[16] A. Bondesan, S. Dellacherie, H. Hivert, J. Jung, V. Lleras, C. Mietka, Y. Penel. *Study of a depressurisation process at low Mach number in a nuclear reactor core*, August 2015, working paper or preprint, https://hal.archives-ouvertes.fr/hal-01258397

[17] S. Dellacherie, J. Jung, P. Omnes, P.-A. Raviart. *Construction of modified Godunov type schemes accurate at any Mach number for the compressible Euler system*, November 2015, working paper or preprint, https://hal.archives-ouvertes.fr/hal-00776629

[18] J.-M. Hérard, J. Jung. *An interface condition to compute compressible flows in variable cross section ducts*, November 2015, working paper or preprint, https://hal.inria.fr/hal-01233251

References in notes

[19] D. Amenga-Mbengoue, D. Genet, C. Lachat, E. Martin, M. Mogé, V. Perrier, F. Renac, M. Ricchiuto, F. Rue. *Comparison of high order algorithms in Aerosol and Aghora for compressible flows*, in "ESAIM: Proceedings", December 2013, vol. 43, pp. 1-16, http://hal.inria.fr/hal-00917411

[20] D. N. Arnold. *An interior penalty finite element method with discontinuous elements*, in "SIAM journal on numerical analysis", 1982, vol. 19, n^o 4, pp. 742–760

[21] D. N. Arnold, F. Brezzi, B. Cockburn, L. D. Marini. *Unified analysis of discontinuous Galerkin methods for elliptic problems*, in "SIAM journal on numerical analysis", 2002, vol. 39, n^o 5, pp. 1749–1779

[22] F. Bassi, L. Botti, A. Colombo, D. D. Pietro, P. Tesini. *On the flexibility of agglomeration based physical space discontinuous Galerkin discretizations*, in "Journal of Computational Physics", 2012, vol. 231, n^o 1, pp. 45 - 65 [*DOI :* 10.1016/j.jcp.2011.08.018], http://www.sciencedirect.com/science/article/pii/S0021999111005055

[23] 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, pp. 507-540

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

[25] F. Bassi, S. Rebay, G. Mariotti, S. Pedinotti, M. Savini. *A high-order accurate discontinuous finite element method for inviscid and viscous turbomachinery flows*, in "Proceedings of the 2nd European Conference on Turbomachinery Fluid Dynamics and Thermodynamics", Technologisch Instituut, Antwerpen, Belgium, 1997, pp. 99–109

[26] 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$^{\text{o}}$ 190, pp. 545–581, http://dx.doi.org/10.2307/2008501

[27] 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$^{\text{o}}$ 1, pp. 90–113

[28] 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$^{\text{o}}$ 186, pp. 411–435, http://dx.doi.org/10.2307/2008474

[29] 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$^{\text{o}}$ 3, pp. 337–361

[30] 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$^{\text{o}}$ 2, pp. 199–224, http://dx.doi.org/10.1006/jcph.1998.5892

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

[32] M. Feistauer, V. Kučera. *On a robust discontinuous Galerkin technique for the solution of compressible flow*, in "J. Comput. Phys.", 2007, vol. 224, n$^{\text{o}}$ 1, pp. 208–221, http://dx.doi.org/10.1016/j.jcp.2007.01.035

[33] U. Frisch. *Turbulence: The Legacy of AN Kolmogorov*, Cambridge University Press, 1995

[34] 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$^{\text{o}}$ 1, pp. 1–20

[35] 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$^{\text{o}}$ 189, pp. 107–129, http://dx.doi.org/10.2307/2008684

[36] 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$^{\text{o}}$ 33, pp. 89–123

[37] 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$^{\text{o}}$ 11, pp. 5649–5670, http://dx.doi.org/10.1016/j.jcp.2008.02.015

[38] W. Reed, T. Hill. *Triangular mesh methods for the neutron transport equation*, Los Alamos Scientific Laboratory, 1973, n$^{\text{o}}$ LA-UR-73-479