



IN PARTNERSHIP WITH:
CNRS

**Université de Pau et des Pays de
l'Adour**

Activity Report 2018

Project-Team CAGIRE

Computational AGility for internal flows
simulations and compaRisons with
Experiments

IN COLLABORATION WITH: Laboratoire de mathématiques et de leurs applications (LMAP)

RESEARCH CENTER
Bordeaux - Sud-Ouest

THEME
Numerical schemes and simulations

Table of contents

1. Team, Visitors, External Collaborators	1
2. Overall Objectives	2
3. Research Program	3
3.1. The scientific context	3
3.1.1. Computational fluid mechanics: modeling or not before discretizing ?	3
3.1.2. Computational fluid mechanics: high order discretization on unstructured meshes and efficient methods of solution	4
3.1.3. Experimental fluid mechanics: a relevant tool for physical modeling and simulation development	6
3.2. Research directions	6
3.2.1. Boundary conditions	6
3.2.1.1. Generating synthetic turbulence	6
3.2.1.2. Stable and non reflecting boundary conditions	6
3.2.2. Turbulence models and model agility	7
3.2.2.1. Extension of zero-Mach models to the compressible system	7
3.2.2.2. Study of wall flows with and without mass or heat transfer at the wall: determination and validation of relevant criteria for hybrid turbulence models	7
3.2.2.3. Improvement of turbulence models	7
3.2.3. Development of an efficient implicit high-order compressible solver scalable on new architectures	7
3.2.3.1. Efficient implementation of the discontinuous Galerkin method	8
3.2.3.2. Implicit methods based on Jacobian-Free-Newton-Krylov methods and multigrid	8
3.2.3.3. Porting on heterogeneous architectures	8
3.2.3.4. Implementation of turbulence models in AeroSol and validation	9
3.2.4. Validation of the simulations: test flow configurations	9
4. Application Domains	10
4.1. Aeronautics	10
4.2. Power stations	10
4.3. Automotive propulsion	10
5. Highlights of the Year	11
6. New Software and Platforms	11
7. New Results	12
7.1. A density-based flux scheme scheme for simulating low Mach flows	12
7.2. A parameter free pressure based approach for simulating flows at all Mach	12
7.3. New models for conjugate heat transfer	13
8. Bilateral Contracts and Grants with Industry	13
8.1. Bilateral Contracts with Industry	13
8.2. Bilateral Grants with Industry	13
9. Partnerships and Cooperations	13
9.1. Regional Initiatives	13
9.1.1. SEIGLE	13
9.1.2. HPC scalable ecosystem	14
9.2. National Initiatives	14
9.2.1. GIS Success	14
9.2.2. ANR MONACO_2025	14
9.3. European Initiatives	15
9.4. International Initiatives	16
9.4.1. Inria International Partners	16
9.4.2. Participation in International Programs	16

9.5. International Research Visitors	16
10. Dissemination	16
10.1. Promoting Scientific Activities	16
10.1.1. Scientific Events Organisation	16
10.1.1.1. General Chair, Scientific Chair	16
10.1.1.2. Member of the Organizing Committees	16
10.1.2. Scientific Events Selection	16
10.1.3. Journal	17
10.1.3.1. Member of the Editorial Boards	17
10.1.3.2. Reviewer - Reviewing Activities	17
10.1.4. Invited Talks	17
10.1.5. Research Administration	17
10.2. Teaching - Supervision - Juries	17
10.2.1. Teaching	17
10.2.2. Supervision	18
10.2.3. Juries	18
10.3. Popularization	19
10.3.1. Internal or external Inria responsibilities	19
10.3.2. Interventions	19
10.3.3. Creation of media or tools for science outreach	19
11. Bibliography	19

Project-Team CAGIRE

Creation of the Team: 2011 June 01, updated into Project-Team: 2016 May 01

Keywords:

Computer Science and Digital Science:

- A6.1.1. - Continuous Modeling (PDE, ODE)
- A6.2.1. - Numerical analysis of PDE and ODE
- A6.2.7. - High performance computing
- A6.5. - Mathematical modeling for physical sciences
- A6.5.2. - Fluid mechanics

Other Research Topics and Application Domains:

- B4. - Energy
- B4.2. - Nuclear Energy Production
- B5.2.1. - Road vehicles
- B5.2.3. - Aviation
- B5.2.4. - Aerospace

1. Team, Visitors, External Collaborators

Research Scientists

- Pascal Bruel [Team leader, CNRS, Researcher, HDR]
- Rémi Manceau [CNRS, Senior Researcher, HDR]
- Vincent Perrier [Inria, Researcher]

Faculty Member

- Jonathan Jung [Univ de Pau et des pays de l'Adour, Associate Professor]

Post-Doctoral Fellow

- Enrique Gutierrez Alvarez [Inria, from Oct 2018]

PhD Students

- Al Hassan Afailal [IFPEN, from Sep 2017]
- Puneeth Bikkanahally Muni Reddy [Univ de Pau et des pays de l'Adour, from Oct 2018]
- Vladimir Duffal [CIFRE EDF, from Nov 2017]
- Gaetan Mangeon [CIFRE EDF, from Feb 2017]
- Gustave Sporshill [Dassault Aviation, from May 2018]
- Syed Mohd Saad Jameel [CIFRE PSA Group, from Feb 2017]

Technical staff

- Benjamin Lux [CNRS, until Jul 2018]

Administrative Assistant

- Sylvie Embolla [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 gradually elaborated a common vision of what should be the simulation tools for fluid dynamics of tomorrow. Our applications will be focused on wall bounded turbulent flows, featuring complex phenomena such as aeroacoustics, hydrodynamic instabilities, phase change processes, complex walls, buoyancy or localized relaminarization. Because such flows are exhibiting a multiplicity of time and length scales of 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 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 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 agile 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. Therefore, a significant part of our activity will be devoted to the proper handling of the constantly evolving supercomputer architectures. But even the best ever simulation library is useless if it is not disseminated and increasingly used by the CFD community as well as our industrial partners. In that respect, the significant success of the low-order finite volume simulation suite OpenFOAM ⁴ or the more recently proposed SU2 ⁵ from Stanford are considered as examples of quite successful dissemination stories that could be, if not followed, but at least considered as a source of inspiration. Our natural inclination though will be to promote the use of the library in direction of our present and future industrial and academic partners, with a special interest on the SMEs active in the highly competitive and strategic economical sectors of energy production and aerospace propulsion. Indeed, these sectors are experiencing a revolution of the entire design process especially for complex parts with an intimate mix between simulations and additive manufacturing (3D printing) processes in the early stages of the design process. For large companies, such as General Electric or Safran (co-developing the CFM Leap-1 engines with 3D printed fuel nozzles), as well as medium-size companies such as Aerojet Rocketdyne, this is a unique opportunity to reduce the duration and hence the cost of development of their systems, while preserving if not strengthening their capability of designing innovative components that cannot be produced by classical manufacturing processes. On the other hand, for the small companies of this sector, this may have a rather detrimental

¹Direct numerical simulation

²Reynolds-averaged Navier-Stokes

³Large-eddy simulation

⁴<http://www.openfoam.com>

⁵<http://su2.stanford.edu/>

effect on their competitiveness since their capability of mastering both these new manufacturing processes and advanced simulation approaches is far more limited. Thus, through our sustained direct (EDF, Turbomeca, PSA group, AD Industrie, Dassault Aviation) or indirect (European programs: WALLTURB, KIAI, IMPACT-AE, SOPRANO; ANR program MONACO_2025) partnership with different companies, we are able to identify relevant generic configurations, from our point of view of scientists, to serve as support for the development of our approach. This methodological choice was motivated by the desire to lead an as efficient as possible transfer activity, while maintaining a clear distinction between what falls within our field of competence of researchers and what is related to the development of their products by our industrial partners. 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/automotive propulsion. Our approach will be combining mesh (h) + turbulence model (m) + discretization order (p) agility. This will be achieved by:

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

Concerning applications, our objective are :

- To reinforce the long term existing partnership with industrial groups active in the sector of energy production and aeronautical/automotive propulsion, and the other European partners involved in the same European projects as we are.
- To consolidate and develop partnership with SMEs operating in the aeronautical sector.

3. Research Program

3.1. The scientific context

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

A typical continuous solution of the Navier-Stokes equations at sufficiently large values of the Reynolds number is governed by a wide spectrum of temporal and spatial scales closely connected with the turbulent nature of the flow. The term deterministic chaos employed by Frisch in his enlightening book [49] 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) and the smallest one (Kolmogorov scale) is proportional to $Re_t^{3/4}$ per dimension, where $Re_t^{3/4}$ is the turbulent Reynolds number, based on the length and velocity scales of the largest turbulent eddies. In addition, for internal flows, viscous effects near the solid walls yield a scaling proportional to Re_τ per dimension, where Re_τ is the friction Reynolds number. 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 the scales, which can lead to unrealistic computational costs in real-world applications. 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 itself. If a modeling phase is on the agenda, then one has to choose again among the above-mentioned variety of approaches. The different simulation options and their date of availability for high-Reynolds-number applications are illustrated in Fig. 1: simulation of turbulent flows can be achieved either by directly solving the Navier-Stokes equations (DNS) or by first applying to the equations a statistical averaging (RANS), a spatial filtering (LES), or a combination of these two operators (hybrid RANS/LES). The new terms

brought about by the 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, and it remains the standard approach nowadays for industrial design, except for very specific applications. 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, assumed more isotropic, LES proved to be quite a breakthrough to fully take advantage of the increasing power of computers to study complex flow configurations. At the same time, DNS was gradually applied to geometries of increasing complexity (channel flows with values of Re_τ multiplied by 45 during the last 30 years, jets, turbulent premixed flames, among many others), and proved to be a formidable tool to (i) improve our knowledge on turbulent flows and (ii) 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, due to the steady nature of the RANS equations, numerical accuracy is generally not ensured via the use of high-order schemes, but rather on careful grid convergence studies. In contrast, 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 ⁶, a turbulence model represents turbulent mechanisms in the same way in the whole flow. Thus, depending on its intrinsic strengths and weaknesses, accuracy will be a rather volatile quantity, strongly dependent on the flow configuration. For instance, RANS is perfectly suited to attached boundary layers, but exhibits severe limitations in massively-separated flow regions. Therefore, 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 limitations of the two approaches and to combine them for significantly improving the overall performance of the models, the hybrid RANS-LES approach has emerged during the last two decades as a viable, intermediate way, and we are definitely inscribing our project in this innovative field of research, with an original approach though, based on temporal filtering (Hybrid temporal LES, HTLES) rather than spatial filtering, and a systematic and progressive validation process against experimental data produced by the team.

3.1.2. 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. In the project, the numerical strategy is based on discontinuous Galerkin methods. These methods were introduced by Reed and Hill [60] and first studied by Lesaint and Raviart [56]. 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 gradually solved for scalar conservation laws [44], [43], one dimensional systems [42], multidimensional scalar conservation laws [41], and multidimensional systems [45]. For the same system, we can also cite the work of [48], [53], which is slightly different: the stabilization is made by adding a nonlinear term, and the time integration is implicit. In contrast 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 [34]. The first application of discontinuous Galerkin methods to Navier-Stokes equations was proposed in [39] by mean of a mixed

⁶<https://cordis.europa.eu/project/rcn/109107/factsheet/en>

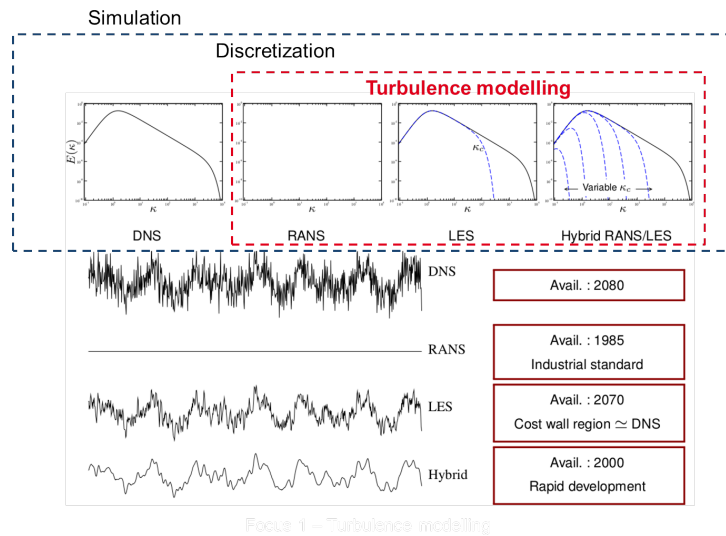


Figure 1. Schematic view of the different nested steps for turbulent flow simulation: from DNS to hybrid RANS-LES. The approximate dates at which the different approaches are or will be routinely used in the industry are indicated in the boxes on the right (extrapolations based on the present rate of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

formulation. Actually, this first attempt led to a non-compact computational stencil, and was later proved to be unstable. A compactness improvement was made in [40], which was later analyzed, and proved to be stable in a more unified framework [35]. The combination with the $k - \omega$ RANS model was made in [38]. As far as Navier-Stokes equations are concerned, we can also cite the work of [51], in which the stabilization is closer to the one of [35], the work of [57] on local time stepping, or the first use of discontinuous Galerkin methods for direct numerical simulation of a turbulent channel flow done in [46]. Discontinuous Galerkin methods became very 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, for which the number of neighbors required increases with the order of approximation.
- They can be developed for any kind of mesh, structured, unstructured, but also for aggregated grids [37]. This is a pro compared not only with finite-difference schemes, which can be developed only on structured meshes, but also compared with continuous finite-element 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 exists 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 closures. Another aspect that deserves attention is the computational cost of discontinuous Galerkin methods, 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.

3.1.3. Experimental fluid mechanics: a relevant tool for physical modeling and simulation development

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 and Flame*, *Physics of Fluids* or *Journal of Computational Physics* or through websites such that of *Ercoftac*⁷ 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 models and simulations is still necessary. The major change that can be noted though concerns 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 temporal and spatial 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 operated in strong interaction with the modeling and simulation activities.

3.2. Research directions

3.2.1. Boundary conditions

3.2.1.1. Generating synthetic turbulence

A crucial point for any multi-scale simulation able to locally switch (in space or time) from a coarse to a fine level of description of turbulence, is the enrichment of the solution by fluctuations as physically meaningful as possible. Basically, this issue is an extension of the problem of the generation of realistic inlet boundary conditions in DNS or LES of subsonic turbulent flows. In that respect, the method of anisotropic linear forcing (ALF) we have developed in collaboration with EDF proved very encouraging, by its efficiency, its generality and simplicity of implementation. So, it seems natural, on the one hand, to extend this approach to the compressible framework and to implement it in *AeroSol*. On the other hand, we shall concentrate (in cooperation with EDF R&D in Chatou in the framework of a the CIFRE PhD of V. Duffal) on the theoretical link between the local variations of the scale of description of turbulence (e.g. a sudden variations in the size of the time filter) and the intensity of the ALF forcing, transiently applied to promote the development of missing fluctuating scales.

3.2.1.2. Stable and non reflecting boundary conditions

In aerodynamics, and especially for subsonic computations, handling inlet and outlet boundary conditions is a difficult issue. A significant amount of work has already been performed for second-order schemes for Navier-Stokes equations, see [59], [62] and the huge number of papers citing it. On the one hand, we believe that decisive improvements are necessary for higher-order schemes: indeed, the less dissipative the scheme is, the worse impact have the spurious reflections. For this purpose, we will first concentrate on the linearized Navier-Stokes system, and analyze the way to impose boundary conditions in a discontinuous Galerkin framework with a similar approach as in [50]. We will also try to extend the work of [63], which deals with Euler equations, to the Navier-Stokes equations.

⁷<http://www.ercoftac.org>

3.2.2. Turbulence models and model agility

3.2.2.1. Extension of zero-Mach models to the compressible system

We shall develop in parallel our multi-scale turbulence modeling and the related adaptive numerical methods of AeroSol. Without prejudice to methods that will be on the podium in the future, a first step in this direction will be to extend to a compressible framework the continuous temporal hybrid RANS/LES method we have developed up to now in a Mach zero context.

3.2.2.2. Study of wall flows with and without mass or heat transfer at the wall: determination and validation of relevant criteria for hybrid turbulence models

In the targeted application domains, turbulence/wall interactions and heat transfer at the fluid-solid interface are physical phenomena whose numerical prediction is at the heart of the concerns of our industrial partners. For instance, for a jet engine manufacturer, being able to properly design the configuration of the cooling of the walls of its engine combustion chamber in the presence of thermoacoustic instabilities is based on the proper identification and a thorough understanding of the major mechanisms that drive the dynamics of the parietal transfer. Our objective is to take advantage of our analysis, experimental and computational tools to actively participate in the improvement of the collective knowledge of such kind of transfer. The flow configurations dealt with from the beginning of the project are those of subsonic, single-phase impinging jets or JICF (jets in crossflow) with the possible presence of an interacting acoustic wave. The issue of conjugate heat transfer at the wall will be also gradually investigated. The existing switchover criteria of the hybrid RANS/LES models will be tested on these flow configurations in order to determine their domain of validity. In parallel, the hydrodynamic instability modes of the JICF will be studied experimentally and theoretically (in cooperation with the SIAME laboratory) in order to determine the possibility to drive a change of instability regime (e.g., from absolute to convective) and thus to propose challenging flow conditions that would be relevant for the setting-up of an hybrid LES/DNS approach aimed at supplementing the hybrid RANS/LES approach.

3.2.2.3. Improvement of turbulence models

The production and subsequent use of DNS (AeroSol library) and experimental (MAVERIC bench) databases dedicated to the improvement of the physical models is a significant part of our activity. In that respect, our present capability of producing in-situ experimental data for simulation validation and flow analysis is clearly a strongly differentiating mark of our project. The analysis of the DNS and experimental data produced make the improvement of the hybrid RANS/LES approach possible. Our hybrid temporal LES (HTLES) method has a decisive advantage over all other hybrid RANS/LES approaches since it relies on a well-defined time-filtering formalism. This feature greatly facilitates the proper extraction from the databases of the various terms appearing in transport equations obtained at the different scales involved (e.g. from RANS to LES). But we would not be comprehensive in that matter if we were not questioning the relevance of any simulation-experiment comparisons. In other words, a central issue is the following question: are we comparing the same quantities between simulations and experiment? From an experimental point of view, the questions to be raised will be, among others, the possible difference in resolution between the experiment and the simulations, the similar location of the measurement points and simulation points, the acceptable level of random error associated to the necessary finite number of samples. In that respect, the recourse to uncertainty quantification techniques will be advantageously considered.

3.2.3. Development of an efficient implicit high-order compressible solver scalable on new architectures

As the flows simulated are very computationally demanding, we will maintain our efforts in the development of AeroSol in the following directions:

- Efficient implementation of the discontinuous Galerkin method.
- Implicit methods based on Jacobian-Free-Newton-Krylov methods and multigrid.
- Porting on heterogeneous architectures.
- Implementation of models.

3.2.3.1. Efficient implementation of the discontinuous Galerkin method

In high-order discontinuous Galerkin methods, the unknown vector is composed of a concatenation of the unknowns in the cells of the mesh. An explicit residual computation is composed of three loops: an integration loop on the cells, for which computations in two different cells are independent, an integration loop on boundary faces, in which computations depend on data of one cell and on the boundary conditions, and an integration loop on the interior faces, in which computations depend on data of the two neighboring cells. Each of these loops is composed of three steps: the first step consists in interpolating data at the quadrature points; the second step in computing a nonlinear flux at the quadrature points (the physical flux for the cell loop, an upwind flux for interior faces or a flux adapted to the kind of boundary condition for boundary faces); and the third step in projecting the nonlinear flux on the degrees of freedom.

In this research direction, we propose to exploit the strong memory locality of the method (i.e., the fact that all the unknowns of a cell are stocked contiguously). This formulation can reduce the linear steps of the method (interpolation on the quadrature points and projection on the degrees of freedom) to simple matrix-matrix product which can be optimized. For the nonlinear steps, composed of the computation of the physical flux on the cells and of the numerical flux on the faces, we will try to exploit vectorization.

3.2.3.2. Implicit methods based on Jacobian-Free-Newton-Krylov methods and multigrid

For our computations of the IMPACT-AE project, we have used explicit time stepping. The time stepping is limited by the CFL condition, and in our flow, the time step is limited by the acoustic wave velocity. As the Mach number of the flow we simulated in IMPACT-AE was low, the acoustic time restriction is much lower than the turbulent time scale, which is driven by the velocity of the flow. We hope to have a better efficiency by using time implicit methods, for using a time step driven by the velocity of the flow.

Using implicit time stepping in compressible flows is particularly difficult, because the system is fully nonlinear, such that the nonlinear solving theoretically requires to build many times the Jacobian. Our experience in implicit methods is that the building of a Jacobian is very costly, especially in three dimensions and in a high-order framework, because the optimization of the memory usage is very difficult. That is why we propose to use a Jacobian-free implementation, based on [55]. This method consists in solving the linear steps of the Newton method by a Krylov method, which requires Jacobian-vector product. The smart idea of this method is to replace this product by an approximation based on a difference of residual, therefore avoiding any Jacobian computation. Nevertheless, Krylov methods are known to converge slowly, especially for the compressible system when the Mach number is low, because the system is ill-conditioned. In order to precondition, we propose to use an aggregation-based multigrid method, which consists in using the same numerical method on coarser meshes obtained by aggregation of the initial mesh. This choice is driven by the fact that multigrid methods are the only one to scale linearly [64], [65] with the number of unknowns in term of number of operations, and that this preconditioning does not require any Jacobian computation.

Beyond the technical aspects of the multigrid approach, which is challenging to implement, we are also interested in the design of an efficient aggregation. This often means to perform an aggregation based on criteria (anisotropy of the problem, for example) [58]. To this aim, we propose to extend the scalar analysis of [66] to a linearized version of the Euler and Navier-Stokes equations, and try to deduce an optimal strategy for anisotropic aggregation, based on the local characteristics of the flow. Note that discontinuous Galerkin methods are particularly well suited to h-p aggregation, as this kind of methods can be defined on any shape [37].

3.2.3.3. Porting on heterogeneous architectures

Until the beginning of the 2000s, the computing capacities have been improved by interconnecting an increasing number of more and more powerful computing nodes. The computing capacity of each node was increased by improving the clock speed, the number of cores per processor, the introduction of a separate and dedicated memory bus per processor, but also the instruction level parallelism, and the size of the memory cache. Even if the number of transistors kept on growing up, the clock speed improvement has flattened since the mid 2000s [61]. Already in 2003, [52] pointed out the difficulties for efficiently using the biggest clusters: "While these super-clusters have theoretical peak performance in the Teraflops range, sustained performance

with real applications is far from the peak. Salinas, one of the 2002 Gordon Bell Awards was able to sustain 1.16 Tflops on ASCI White (less than 10% of peak)." From the current multi-core architectures, the trend is now to use many-core accelerators. The idea behind many-core is to use an accelerator composed of a lot of relatively slow and simplified cores for executing the most simple parts of the algorithm. The larger the part of the code executed on the accelerator, the faster the code may become. Therefore, it is necessary to work on the heterogeneous aspects of computations. These heterogeneities are intrinsic to our computations and have two sources. The first one is the use of hybrid meshes, which are necessary for using a locally-structured mesh in a boundary layer. As the different cell shapes (pyramids, hexahedra, prisms and tetrahedra) do not have the same number of degrees of freedom, nor the same number of quadrature points, the execution time on one face or one cell depends on its shape. The second source of heterogeneity are the boundary conditions. Depending on the kind of boundary conditions, user-defined boundary values might be needed, which induces a different computational cost. Heterogeneities are typically what may decrease efficiency in parallel if the workload is not well balanced between the cores. Note that heterogeneities were not dealt with in what we consider as one of the most advanced work on discontinuous Galerkin on GPU [54], as only straight simplicial cell shapes were addressed. For managing at best our heterogeneous computations on heterogeneous architectures, we propose to use the execution runtime StarPU [36]. For this, the discontinuous Galerkin algorithm will be reformulated in terms of a graph of tasks. The previous tasks on the memory management will be useful for that. The linear steps of the discontinuous Galerkin methods require also memory transfers, and one issue consists in determining the optimal task granularity for this step, i.e. the number of cells or face integrations to be sent in parallel on the accelerator. On top of that, the question of which device is the most appropriate to tackle such kind of tasks is to be discussed.

Last, we point out that the combination of shared-memory and distributed-memory parallel programming models is better suited than only the distributed-memory one for multigrid, because in a hybrid version, a wider part of the mesh shares the same memory, therefore making a coarser aggregation possible.

These aspects will benefit from a particularly stimulating environment in the Inria Bordeaux Sud Ouest center around high-performance computing, which is one of the strategic axes of the center.

3.2.3.4. Implementation of turbulence models in AeroSol and validation

We will gradually insert models developed in research direction 3.2.2.1 in the AeroSol library in which we develop methods for the DNS of compressible turbulent flows at low Mach number. Indeed, due to its formalism based on temporal filtering, the HTLES approach offers a consistent theoretical framework characterized by a continuous transition from RANS to DNS, even for complex flow configurations (e.g. without directions of spatial homogeneity). As for the discontinuous Galerkin method available presently in AeroSol, it is the best suited and versatile method able to meet the requirements of accuracy, stability and cost related to the local (varying) level of resolution of the turbulent flow at hand, regardless of its complexity. The first step in this direction was taken in 2017 during the internship of Axelle Perraud, who has implemented a turbulence model (k - ω -SST) in the Aerosol library.

3.2.4. Validation of the simulations: test flow configurations

To supplement whenever necessary the test flow configuration of MAVERIC and apart from configurations that could emerge in the course of the project, the following configurations for which either experimental data, simulation data or both have been published will be used whenever relevant for benchmarking the quality of our agile computations:

- The impinging turbulent jet (simulations).
- The ORACLES two-channel dump combustor developed in the European projects LES4LPP and MOLECULES.
- The non reactive single-phase PRECCINSTA burner (monophasic swirler), a configuration that has been extensively calculated in particular with the AVBP and Yales2 codes.
- The LEMCOTEC configuration (monophasic swirler + effusion cooling).
- The ONERA MERCATO two-phase injector configuration provided the question of confidentiality of the data is not an obstacle.

- Rotating turbulent flows with wall interaction and heat transfer.
- Turbulent flows with buoyancy.

4. Application Domains

4.1. Aeronautics

Cagire is presently involved in studies mainly related to:

- The combustion chamber wall: the modelling, the simulation and the experimentation of the flow around a multiperforated plate representative of a real combustion chamber wall are the three axes we have been developing during the recent period. The continuous improvement of our in-house test facility Maveric is also an important ingredient to produce our own experimental validation data for isothermal flows. For non-isothermal flows, our participation in the EU funded program Soprano will be giving us access to non-isothermal data produced by Onera.
- The flow around airfoils: the modelling of the turbulent boundary layer has been for almost a century a key issue in the aeronautics industry. However, even the more advanced RANS models face difficulties in predicting the influence of pressure gradients on the development of the boundary layer. A main issue is the reliability of the modelling hypotheses, which is crucial for less conservative design. One of the technological barriers is the prediction of the flow in regimes close to the edge of the flight domain (stall, buffeting, unsteady loads) when the boundary layer is slowed down by an adverse pressure gradient. This is the subject of the CIFRE PhD thesis of Gustave Sporschill, started in 2018, in collaboration with Dassault Aviation.

4.2. Power stations

R. Manceau has established a long term collaboration (4 CIFRE PhD theses in the past, 2 ongoing) 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 :

- The prediction of heat transfer in fluid and solid components is of major importance in power stations, in particular, nuclear power plants. Either for the thermohydraulics of the plenum or in the study of accidental scenarii, among others, the accurate estimation of wall heat transfer, mean temperatures and temperature fluctuations are necessary for the evaluation of relevant thermal and mechanical design criteria. The PhD thesis (CIFRE EDF) of G. Mangeon is dedicated to the development of relevant RANS models for these industrial applications.
- Moreover, the prediction of unsteady hydrodynamic loadings is a key point for operating and for safety studies of PWR power plants. Currently, the static loading is correctly predicted by RANS computations but when the flow is transient (as, for instance, in Reactor Coolant Pumps, due to rotor/stator interactions, or during operating transients) or in the presence of large, energetic, coherent structures in the external flow region, the RANS approach is not sufficient, whereas LES is still too costly for a wide use in industry. This issue constitutes the starting point of the just-started PhD thesis (CIFRE EDF) of Vladimir Duffal.

4.3. Automotive propulsion

- The engine (underhood) compartment is a key component of vehicle design, in which the temperature is monitored to ensure the effectiveness and safety of the vehicle, and participates in 5 to 8% of the total drag and CO2 emissions. Dimensioning is an aerodynamic and aerothermal compromise, validated on a succession of road stages at constant speed and stopped phases (red lights, tolls, traffic jam). Although CFD is routinely used for forced convection, state-of-the-art turbulence models are not able to reproduce flows dominated by natural convection during stopped phases, with a Rayleigh

number of the order of 10^{10} , such that the design still relies on costly, full-scale, wind tunnel experiments. This technical barrier must be lifted, since the ambition of the PSA group is to reach a *full digital design of their vehicles in the 2025 horizon*, i.e., to almost entirely rely on CFD. This issue is the focus of the ongoing PhD thesis (CIFRE PSA) of S. Jameel, supervised by R. Manceau, and also a part of the ANR project MONACO_2025 described in section 9.2.2.

- The Power & Vehicles Division of IFPEN co-develops a CFD code to simulate the internal flow in a spark-ignition engine, in order to provide the automotive industry with tools to optimize the design of combustion engines. The RANS method, widely used in the industry, is not sufficiently reliable for quantitative predictions, and is only used as a tool to qualitatively compare different geometries. On the other hand, LES provides more detailed and accurate information, but at the price of a CPU cost unaffordable for daily use in the industry. Therefore, IFPEN aims at developing the hybrid RANS/LES methodology, in order to combine the strengths of the two approaches. The PhD thesis of Hassan Afailal, co-supervised by Rémi Manceau, is focused on this issue.

5. Highlights of the Year

5.1. Highlights of the Year

ANR MONACO_2025

The ANR MONACO_2025 project started in March 2018. The consortium of this project, coordinated by [RM], consists in an academic partner, the institute PPrime of Poitiers, and two industrial partners, PSA and EDF. It is focused on the development of a CFD methodology for transient, buoyancy-affected turbulent flows, that are crucial for the two industrial partners. Four PhD students, Saad Jameel (CIFRE PSA grants), Puneeth Reddy (ANR grant), Gaëtan Mangeon (CIFRE EDF) and Vladimir Duffal (CIFRE EDF) are involved in this project, which plays a major role in the active collaboration among these students.

A new industrial partner

A collaboration started in 2018 with a new industrial partner, Dassault Aviation, via the CIFRE PhD of Gustave Sporschill supervised by Rémi Manceau.

A new regional initiative

Cagire is part of the 3-year program HPC scalable ecosystem funded by Région Nouvelle-Aquitaine in the framework of its 2018 call.

HTLES in the commercial code CONVERGE

In the framework of the IFPEN PhD thesis of Al Hassan Afailal (supervision by Rémi Manceau), the hybrid RANS/LES method developed in the project-team CAGIRE has been implemented in the commercial software CONVERGE (<https://convergecf.com>).

6. New Software and Platforms

6.1. AeroSol

KEYWORD: Finite element modelling

FUNCTIONAL DESCRIPTION: The AeroSol software is a high order finite element library written in C++. The code has been designed so as to allow for efficient computations, with continuous and discontinuous finite elements methods on hybrid and possibly curvilinear meshes. The work of the team CARDAMOM (previously 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 PaMPA are 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 TADAAM (and previously in Bacchus) and the team Castor.

NEWS OF THE YEAR: In 2018, the following points were addressed in AeroSol

- * A 6 month CNRS contract, led by Vincent Perrier was obtained in the team Cagire, concentrated on the quality of the code. A two-year Inria Hub, led by H elo ise Beaugendre was obtained in the team Cardamom. On both of these contracts, Benjamin Lux was hired (January-June in Cagire team, and since October in Cardamom team). The library has kept on benefiting from the work of Florent Pruvost (Inria Hub HPCLib), on the continuous integration and packaging aspects.
- * The CNRS contract, aiming at improving the code resulted in the successful porting from the inria gforge to the inria gitlab, with a functional pipeline of code assessment (based on Jenkins) and code quality assessment (based on sonarqube).
- * Installation was simplified. A fully automatic installation script based on spack was developed.
- * Development of a true documentation policy, based on a wiki. About half of the functional tests were documented and updated. Doxygen documentation was improved.
- * An API was developed for the AeroSol library. The mesh reading, and parallel distribution was refactored.
- * Update of the test case interface was updated by using this API for being more convenient. About half of the functional tests are now using this interface.
- * Refactoring of the xml parameter file.
- * Hyperbolized models, based on the hyperbolization of advection-diffusion models were added.
- * Handling of nonconservative hyperbolic models.
- * Improvement of mesh adaptation
- * Beginning of implementation of droplet model for icing.
- * Add the possibility of using several matrices, with a different number of variables
 - Participants: Benjamin Lux, Damien Genet, Dragan Amenga Mbengoue, Hamza Belkhatay Zougari, Mario Ricchiuto, Maxime Mog e, Simon Delmas and Vincent Perrier
 - Contact: Vincent Perrier

7. New Results

7.1. A density-based flux scheme scheme for simulating low Mach flows

Participants: Pascal Bruel, Jonathan Jung, Vincent Perrier.

The topic dealt with concerns acoustic computations in low Mach number flows with density based solvers. For ensuring a good resolution of the low Mach number base flow, a scheme able to deal with stationary low Mach number flows is necessary. Previously proposed low Mach number fixes have been tested with acoustic computations. Numerical results prove that they are not accurate for acoustic computations. The issues raised with acoustic computations with low Mach number fixes were studied and a new scheme has been developed, in order to be accurate not only for steady low Mach number flows, but also for acoustic computations. Numerical tests evidenced the improvement of the proposed scheme with respect to the state of the art [9].

7.2. A parameter free pressure based approach for simulating flows at all Mach

Participant: Pascal Bruel.

A pressure-correction algorithm developed in close partnership with Prof. E. Dick (Ghent University, Belgium) and Dr. Y. Moguen (UPPA, France) has been developed and extensively tested for a wide range of compressible fluid flow regimes. It proved to be well-suited to simulate flows at all levels of Mach number with smooth and discontinuous flow field changes, by providing a precise representation of convective transport and acoustic propagation. A co-located finite volume space discretization is used with the AUSM flux splitting. It is demonstrated that two ingredients are essential for obtaining good quality solutions: the presence of an inertia term in the transporting velocity expression; a velocity difference diffusive term in the face pressure expression, with a correct Mach number scaling to recover the hydrodynamic and acoustic low Mach number limits. To meet these two requirements, a new flux scheme, named MIAU, for Momentum Interpolation with Advection Upstream splitting has been proposed (one journal paper submitted in 2018).

7.3. New models for conjugate heat transfer

Participant: Rémi Manceau.

New models valid in the near-wall region have been proposed for both the turbulent heat flux and the dissipation rate of the temperature variance in the framework of the EDF CIFRE PhD thesis of G. Mangeon. The purpose is to extend the Elliptic Blending approaches developed in the team to all possible boundary conditions for the temperature: imposed wall-temperature, imposed heat flux or conjugate heat transfer, which is of primary importance for applications in the nuclear industry. The new full model (which associates the two above-mentioned models) is the first one to satisfy all the near-wall budgets and, consequently, the asymptotic behavior of all the quantities. These results have been presented at two international symposia [16], [15].

8. Bilateral Contracts and Grants with Industry

8.1. Bilateral Contracts with Industry

- EDF: "Advanced modelling of heat transfer for industrial configurations with or without accounting of the solid wall", contract associated to the PhD thesis of Gaëtan Mangeon
- EDF: "Hybrid RANS/LES modelling for unsteady loadings in turbulent flows", contract associated to the PhD thesis of Vladimir Duffal
- IFPEN: "3D simulation of non-reactive internal aerodynamics of spark-ignition engines using an hybrid RANS/LES method", contract associated to the PhD thesis of Hassan Al Afailal
- PSA: "Turbulence modelling in the mixed and natural convection regimes in the context of automotive applications", contract associated to the PhD thesis of Saad Jameel.

8.2. Bilateral Grants with Industry

- EDF (Cifre PhD grant): "Advanced modelling of heat transfer for industrial configurations with or without accounting of the solid wall", PhD student: Gaëtan Mangeon
- EDF (Cifre PhD grant): "Hybrid RANS/LES modelling for unsteady loadings in turbulent flows", PhD student: Vladimir Duffal
- IFPEN (PhD grant): "3D simulation of non-reactive internal aerodynamics of spark-ignition engines using an hybrid RANS/LES method", PhD student: Hassan Al Afailal
- PSA (Cifre PhD grant): "Turbulence modelling in the mixed and natural convection regimes in the context of automotive applications", PhD student: Saad Jameel.
- Dassault Aviation (Cifre PhD grant): "Amélioration des modèles pour la turbulence. Applications à la prédiction des écoulements aérodynamiques.", PhD student: Gustave Sporschill.

9. Partnerships and Cooperations

9.1. Regional Initiatives

9.1.1. SEIGLE

Participants: Enrique Gutierrez Alvarez, Jonathan Jung, Vincent Perrier.

SEIGLE means "Simulation Expérimentation pour l'Interaction de Gouttes Liquides avec un Ecoulement fortement compressible". It is a 3-year program which has started since October 2017 and was funded by Régional Nouvelle-Aquitaine, ISAE-ENSMA, CESTA and Inria. The interest of understanding aerodynamic mechanisms and liquid drops atomization is explained by the field of applications where they play a key role, specially in the new propulsion technologies through detonation in the aerospace as well as in the securities field. The SEIGLE project was articulated around a triptych experimentation, modeling and simulation. An experimental database will be constituted. It will rely on a newly installed facility (Pprime), similar to a supersonic gust wind tunnel/ hypersonic from a gaseous detonation tube at high pressure. This will allow to test modeling approaches (Pprime / CEA) and numerical simulation (Inria / CEA) with high order schemes for multiphasic compressible flows, suitable for processing shock waves in two-phase media. Enrique Gutierrez Alvarez (post-doctoral) joined the team in October 2018 to work on this project.

9.1.2. HPC scalable ecosystem

Participants: Jonathan Jung, Vincent Perrier, [A two-year Post-doc starting in 2019 or 2020].

HPC scalable ecosystem is a 3-year program funded by Région Nouvelle-Aquitaine (call 2018), Airbus, CEA-CESTA, University of Bordeaux, INRA, ISAE-ENSMA and Inria. A two-year post-doc will be hired in 2019 or 2020. The objective is to extend the prototype developed in [47] to high order (discontinuous Galerkin) and non-reactive diffusive flows in 3d. The same basis will be developed in collaboration with Pprime for WENO based methods for reactive flows.

9.2. National Initiatives

9.2.1. GIS Success

Participant: Pascal Bruel.

We are members of the CNRS GIS Success (Groupement d'Intérêt Scientifique) organised around two of the major CFD codes employed by the Safran group, namely AVBP and Yales2. No scientific activity has been devoted around those codes during 2018, but a 2-day Yales2 training session held in Rouen was attended by Pascal Bruel. Some technical activity was also devoted to install Yales2 at Evora University (Portugal) in the framework of the on-going informal scientific cooperation with Dr. P. Correia.

9.2.2. ANR MONACO_2025

Participant: Rémi Manceau.

The ambition of the MONACO_2025 project, coordinated by Rémi Manceau, is to join the efforts made in *two different industrial sectors* in order to tackle the industrial simulation of transient, turbulent flows affected by buoyancy effects. It brings together two academic partners, the project-team Cagire hosted by the university of Pau, and the institute Pprime of the CNRS/ENSMA/university of Poitiers (PPRIME), and R&D departments of two industrial partners, the PSA group and the EDF group, who are major players of the automobile and energy production sectors, respectively.

- The main **scientific objective** of the project is to make a breakthrough in *the unresolved issue* of the modelling of turbulence/buoyancy interactions in transient situations, within the continuous hybrid RANS/LES paradigm, which consists in preserving a computational cost compatible with industrial needs by relying on statistical approaches where a fine-grained description of the turbulent dynamics is not necessary. The transient cavity flow experiments acquired during MONACO_2025 will provide the partners and the scientific community with *an unrivalled source of knowledge* of the physical mechanisms that must be accounted for in turbulence models.
- The main **industrial objective** is *to make available computational methodologies* to address dimensioning, reliability and security issues in buoyancy-affected transient flows. It is to be emphasized that such problems are *not tackled using CFD at present in the industry*. At the end of MONACO_2025, a panel of methodologies, ranging from simple URANS to sophisticated hybrid

model based on improved RANS models, will be evaluated in transient situations, against the dedicated cavity flow experiments and a real car underhood configuration. This final benchmark exercise will form a *decision-making tool* for the industrial partners, and will thus pave the way towards high-performance design of low-emission vehicles and highly secure power plants. In particular, the project is in line with the *Full Digital 2025 ambition*, e.g., the declared ambition of the PSA group to migrate, within the next decade, to a design cycle of new vehicles nearly entirely based on CAE (computer aided engineering), without recourse to expensive full-scale experiments.

9.3. European Initiatives

9.3.1. FP7 & H2020 Projects

9.3.1.1. SOPRANO

Participants: Rémi Manceau, Pascal Bruel, [A one-year Post-doc starting in January 2019].

Topic: MG-1.2-2015 - Enhancing resource efficiency of aviation

Project acronym: SOPRANO

Project title: Soot Processes and Radiation in Aeronautical inNOvative combustors

Duration: 01/09/2016 - 31/08/2020

Coordinator: SAFRAN

Other partners:

- France: CNRS, CERFACS, INSA Rouen, SAFRAN SA, Snecma SAS, Turbomeca SA.
- Germany: DLR, GE-DE GmbH, KIT, MTU, RRD,
- Italy: GE AVIO SRL, University of Florence
- United Kingdom: Rolls Royce PLC, Imperial College of Science, Technology and Medicine, Loughborough University.

Abstract: For decades, most of the aviation research activities have been focused on the reduction of noise and NO_x and CO₂ emissions. However, emissions from aircraft gas turbine engines of non-volatile PM, consisting primarily of soot particles, are of international concern today. Despite the lack of knowledge toward soot formation processes and characterization in terms of mass and size, engine manufacturers have now to deal with both gas and particles emissions. Furthermore, heat transfer understanding, that is also influenced by soot radiation, is an important matter for the improvement of the combustor's durability, as the key point when dealing with low-emissions combustor architectures is to adjust the air flow split between the injection system and the combustor's walls. The SOPRANO initiative consequently aims at providing new elements of knowledge, analysis and improved design tools, opening the way to: • Alternative designs of combustion systems for future aircrafts that will enter into service after 2025 capable of simultaneously reducing gaseous pollutants and particles, • Improved liner lifetime assessment methods. Therefore, the SOPRANO project will deliver more accurate experimental and numerical methodologies for predicting the soot emissions in academic or semi-technical combustion systems. This will contribute to enhance the comprehension of soot particles formation and their impact on heat transfer through radiation. In parallel, the durability of cooling liner materials, related to the walls air flow rate, will be addressed by heat transfer measurements and predictions. Finally, the expected contribution of SOPRANO is to apply these developments in order to determine the main promising concepts, in the framework of current low-NO_x technologies, able to control the emitted soot particles in terms of mass and size over a large range of operating conditions without compromising combustor's liner durability and performance toward NO_x emissions.

In the SOPRANO project, our objective is to complement the experimental (ONERA) and LES (CERFACS) work by RANS computations of the flow around a multiperforated plate, in order to build a database making possible a parametric study of mass, momentum and heat transfer through the plate and the development of multi-parameter-dependent equivalent boundary conditions. This year, we attended the two ITR meetings in Stuttgart (Germany) and Bordes (France) and recruited a post-doc bound to start by mid-january 2019.

9.4. International Initiatives

9.4.1. Inria International Partners

9.4.1.1. Informal International Partners

- Institute of Mathematics, Almaty, Kazakhstan:
Collaboration with Drs A. Beketaeva and A. Naimanova for the RANS simulations of a supersonic jet in crossflow configuration for a wide range of pressure ratio (1 journal paper submitted in 2018). The low-Mach preconditioning of an in-house ENO based compressible flow solver was also further investigated. [PB] (one 13-day stay in Almaty).
- Collaboration with P. Correia (University of Evora, Portugal) related to the development of enhanced boundary conditions for the simulations of Mach zero flows with the artificial compressibility method and low Mach flows with a pressure-based approach. [PB] (two 5-day stays in Evora).
- Collaboration with S. Lardeau (Siemens Industry Software Computational Dynamics, Nuremberg, Germany) on the EB-RSM model and hybrid RANS/LES model for industrial applications. [RM]

9.4.2. Participation in International Programs

Participants: Pascal Bruel, Jonathan Jung, Rémi Manceau, Vincent Perrier.

- ECOS-Sud A17A07 project with UNC (Cordoba, Argentina): this is the first year of this project devoted to the simulations of the wind around aerial fuel tank. Jonathan Jung and Pascal Bruel spent several weeks at UNC in the framework of this project.

9.5. International Research Visitors

- Dr. Juan Pablo Saldia (University of Cordoba, Argentina) spent one month in the team in the framework of the A17A07 Ecos-Sud project.

10. Dissemination

10.1. Promoting Scientific Activities

10.1.1. Scientific Events Organisation

10.1.1.1. General Chair, Scientific Chair

- 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. Organization of the next workshop to be held at the Jožef Stefan Institute in Ljubljana, Slovenia in 2019.
- Scientific chair of the mini-symposium on Hybrid RANS/LES methods of the Fluids Engineering Division Summer Meeting (FEDSM) of ASME held in Montreal in 2018 [RM].

10.1.1.2. Member of the Organizing Committees

- Organizer and scientific chair of the mini-symposium on numerical schemes for compressible flows at low Mach number at CANUM 2018 [JJ].

10.1.2. Scientific Events Selection

10.1.2.1. Member of the Conference Program Committees

- Intl Symp. Turbulence, Heat and Mass Transfer [RM]

10.1.3. Journal

10.1.3.1. Member of the Editorial Boards

- Visualization of Mechanical Processes [PB]
- Advisory Board of International Journal of Heat and Fluid Flow [RM]
- Advisory Board of Flow, Turbulence and Combustion [RM]

10.1.3.2. Reviewer - Reviewing Activities

During 2018, the team members reviewed 22 papers for the following journals:

- Aerospace Science and Technology (3) [PB]
- AIAA Journal (2) [PB,RM]
- Computational Thermal Sciences (1) [PB]
- Computers and Fluids (3) [RM,VP(2)]
- Energy and Buildings (1) [PB]
- Flow, Turbulence and Combustion (3) [RM]
- International Journal of Heat and Fluid flow (1) [RM]
- International Journal of Heat and Mass Transfer (2) [PB]
- Journal of Buildings Engineering (1) [PB]
- Journal of Petroleum Science and Engineering (1) [PB]
- Journal of Scientific Computing (1) [VP]
- Nuclear Engineering and Design (1) [RM]
- Physics of Fluids (1) [RM]
- Theoretical and Computational Fluid Dynamics (1) [RM]

10.1.4. Invited Talks

- P. Bruel [18]
- R. Manceau [10]

10.1.5. Research Administration

- Co-responsible for the organisation of the LMAP seminar of Mathematics and their Applications [JJ].
- Member of the LMAP council [JJ, PB].
- Member of the IPRA research federation scientific council [RM].

10.2. Teaching - Supervision - Juries

10.2.1. Teaching

Licence : [JJ], Research and innovation, 1h50, L1, Université de Pau et des Pays de l'Adour, Pau, France.

Licence : [JJ], Descriptive statistical, 24h, L1 - MIASHS, Université de Pau et des Pays de l'Adour, Pau, France.

Licence : [JJ], Scientific computing, 19h, L2 - Informatique, Université de Pau et des Pays de l'Adour, Pau, France.

Master : [JJ], Data analysis, 68h25, M1 - GP, Université de Pau et des Pays de l'Adour, Pau, France.

Master : [JJ], Tools for scientific computing, 48h75, M1 - MMS-MSID, Université de Pau et des Pays de l'Adour, Pau, France.

Master : [JJ,VP], Finite volume methods for hyperbolic systems, 52h50, M2 - MMS , Université de Pau et des Pays de l'Adour, Pau, France.

Master : "Turbulence modelling" (in English), 27h30, M2 - International Master program Turbulence, Université de Poitiers/Ecole centrale de Lille, France. [RM]

Eng. 3 : "Industrial codes for CFD" (in English), 12h30, 3rd year of engineering school (M2), ENSMA, Poitiers, France. [RM]

Eng. 3 : "Advanced physics-Turbulence modelling for CFD", 16h, 3rd year of engineering school (M2), ENSGTI, France. [RM]

10.2.2. Supervision

- PhD in progress : Puneeth Bikkanahally Muni Reddy, "Modelling turbulent flows in natural convection regimes using hybrid RANS-LES approaches, UPPA, October 2018, Rémi Manceau.
- PhD in progress : Gaëtan Mangeon, "Advanced modelling of heat transfer for industrial configurations with or without accounting of the solid wall", UPPA, February 2017, Rémi Manceau.
- PhD in progress : Vladimir Duffal, "Hybrid RANS/LES modelling for unsteady loadings in turbulent flows", UPPA, November 2017, Rémi Manceau.
- PhD in progress : Hassan Al Afailal: "3D simulation of non-reactive internal aerodynamics of spark-ignition engines using an hybrid RANS/LES method", September 2017, Rémi Manceau.
- PhD in progress Saad Jameel : "Turbulence modelling in the mixed and natural convection regimes in the context of automotive applications", UPPA, February 2017, Rémi Manceau.
- PhD in progress : Gustave Sporschill, "Amélioration des modèles pour la turbulence. Applications à la prédiction des écoulements aérodynamiques", UPPA, May 2018, Rémi Manceau.

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):

- François Delassaux, "Modélisation instationnaire de l'aérodynamique externe automobile", Conservatoire National des Arts et Métiers, 20 December 2018. Supervisor: I. Mortazavi [RM, Referee]
- Loïc Jecker, "Prévision de la transition bypass à l'aide d'un modèle à énergie cinétique laminaire basé sur la dynamique des modes de Klebanoff", ONERA, 15 November 2018. Supervisor: G. Casalis [RM, Referee]
- PhD: R. Bizzari "Modélisation aérodynamique et thermique des plaques multiperforées en LES", Université de Toulouse, France, 5 November 2018. Supervisors: T. Poinso and A. Dauptain. [PB]
- PhD: D. Maestro "Large eddy simulation of the interactions between flames and thermal phenomena : application to wall heat transfer and combustion control" Université de Toulouse, France, 27 September 2018. Supervisors: T. Poinso et G. Staffelbach. [PB, Referee]
- Cédric Uribe, "Développement d'une approche ZDES à deux équations de transport et application turbomachines", Sorbonne Université/ONERA, 24 September 2018. Supervisor: G. Gerolymos [RM, Referee]
- PhD: Valentin Bonnifet, "Prédiction du phénomène de tremblement sur un profil d'aile avec une approche LES de type PANS-RSM", Sorbonne Université, 19 September 2018. Supervisor: I. Vallet [RM, Referee]
- PhD: F. Guillois "Simulation d'une zone de mélange turbulente issue de l'instabilité de Richtmyer-Meshkov à l'aide d'un modèle à fonction de densité de probabilité – Analyse du transport de l'énergie turbulente", Université de Lyon, France, 7 September 2018. Supervisors: S. Simoëns and V.A. Sabel'nikov. [PB, Referee]
- PhD: Q. Douasbin "Acoustic waves in combustion devices : interactions with flames and boundary conditions", Université de Toulouse, France, 30 March 2018. Supervisors: T. Poinso and L. Selle. [PB, Referee]
- PhD: B. P. Trevisan "Estudo experimental da interação turbulência, combustão e acústica aplicada a motores aeroespaciais", INPE, São José dos Campos, Brazil, 28 February 2018. Supervisor: W.M.C. Dourado. [PB]

10.3. Popularization

10.3.1. Internal or external Inria responsibilities

- Vincent Perrier is a member of the CUMI-R.
- Vincent Perrier is a member of the CDT, in charge of the evaluation of software projects at the Inria Bordeaux center.
- Vincent Perrier is an elected member of the CLHSCT.
- Vincent Perrier is an elected member of the Inria evaluation committee.⁸
- Vincent Perrier is a member of the CT3-Num committee of Pau University, in charge of managing the computing resources and projects at Pau University.

10.3.2. Interventions

- «Forum des Métiers» organized by Collège Pierre Emmanuel, Pau (64), France, 9 February 2018. A stand was manned during one day with the objective of explaining the activity of researcher to an audience of middle school students. [PB]
- «Savoir en Partage», organized by Lacq Odyssee. [PB [22], JJ and VP [27], [24], RM [29], [28]]
- «Café des Sciences». [PB [23], JJ, RM [30], VP [25]]
- «Fête de la Science - Journée Portes Ouvertes Centre Inria BSO», Talence, France, 13 October 2018. [PB]
- «10 ans du centre Inria BSO: célébration de 10 ans de recherche et de transfert», Talence, France, 28 September 2018. [JJ and VP]

10.3.3. Creation of media or tools for science outreach

- «Science on tourne». [PB, JJ, RM, VP]
<http://www.cestdanslaire.fr/fr/page/science-on-tourne>

11. Bibliography

Major publications by the team in recent years

- [1] 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*, in "Mathematical Models and Methods in Applied Sciences", November 2016 [DOI : 10.1142/S0218202516500603], <https://hal.archives-ouvertes.fr/hal-00776629>
- [2] J.-L. FLORENCIANO, P. BRUEL. *LES fluid-solid coupled calculations for the assessment of heat transfer coefficient correlations over multi-perforated walls*, in "Aerospace Science and Technology", 2016, vol. 53, 13 p. [DOI : 10.1016/J.AST.2016.03.004], <https://hal.inria.fr/hal-01353952>
- [3] E. FRANQUET, V. PERRIER. *Runge-Kutta discontinuous Galerkin method for the approximation of Baer and Nunziato type multiphase models*, in "Journal of Computational Physics", February 2012, vol. 231, n^o 11, pp. 4096-4141 [DOI : 10.1016/J.JCP.2012.02.002], <https://hal.inria.fr/hal-00684427>
- [4] C. FRIESS, R. MANCEAU, T. GATSKI. *Toward an equivalence criterion for Hybrid RANS/LES methods*, in "Computers and Fluids", 2015, vol. 122, pp. 233-246 [DOI : 10.1016/J.COMPFLUID.2015.08.010]

⁸<https://www.inria.fr/en/institute/organisation/committees/evaluation-committee>

- [5] J.-M. HÉRARD, J. JUNG. *An interface condition to compute compressible flows in variable cross section ducts*, in "Comptes Rendus Mathématique", February 2016 [DOI : 10.1016/J.CRMA.2015.10.026], <https://hal.inria.fr/hal-01233251>
- [6] R. MANCEAU. *Recent progress in the development of the Elliptic Blending Reynolds-stress model*, in "Int. J. Heat Fluid Fl.", 2015, vol. 51, pp. 195-220, <http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.09.002>
- [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>

Publications of the year

Articles in International Peer-Reviewed Journals

- [8] V. BOUTROUCHE, E. FRANQUET, S. SERRA, R. MANCEAU. *Influence of the turbulence model for channel flows with strong transverse temperature gradients*, in "International Journal of Heat and Fluid Flow", April 2018, vol. 70, pp. 79-103, <https://hal.inria.fr/hal-01944199>
- [9] P. BRUEL, S. DELMAS, J. JUNG, V. PERRIER. *A low Mach correction able to deal with low Mach acoustics*, in "Journal of Computational Physics", February 2019, vol. 378, pp. 723-759 [DOI : 10.1016/J.JCP.2018.11.020], <https://hal.inria.fr/hal-01953424>

Invited Conferences

- [10] R. MANCEAU. *Modélisation des transferts thermiques turbulents (conférence plénière)*, in "26e congrès français de thermique", Pau, France, May 2018, <https://hal.inria.fr/hal-01944227>
- [11] R. MANCEAU. *Progress in Hybrid Temporal LES (plenary lecture)*, in "6th Symposium on Hybrid RANS-LES Methods", Strasbourg, France, Progress in Hybrid RANS-LES Modelling. Papers contributed to the 6th Symp. Hybrid RANS-LES Methods, 26-28 September 2016, Strasbourg, France. Notes on Numerical Fluid Mechanics and Multidisciplinary Design, September 2018, vol. 137, <https://hal.inria.fr/hal-01391899>

International Conferences with Proceedings

- [12] A. G. FILIPPINI, S. DE BRYE, V. PERRIER, F. MARCHE, M. RICCHIUTO, D. LANNES, P. BONNETON. *UHAINA : A parallel high performance unstructured near-shore wave model*, in "Journées Nationales Génie Côtier - Génie Civil", La Rochelle, France, Editions Paralia, May 2018 [DOI : 10.5150/JNGCGC.2018.006], <https://hal.inria.fr/hal-01824108>
- [13] M. LORINI, C. DOBRZYNSKI, V. PERRIER, M. RICCHIUTO. *A Discontinuous Galerkin Immersed Boundary Method Using Unstructured Anisotropic Mesh Adaptation and Penalization Techniques*, in "13th World Congress in Computational Mechanics", New York, United States, July 2018, <https://hal.inria.fr/hal-01824109>
- [14] M. LORINI, C. DOBRZYNSKI, V. PERRIER, M. RICCHIUTO. *Preliminary results of a Discontinuous Galerkin immersed boundary method combining penalisation and anisotropic adaptation*, in "6th European Conference on Computational Mechanics (ECCM 6)/7th European Conference on Computational Fluid Dynamics (ECFD 7)", Glasgow, United Kingdom, June 2018, <https://hal.inria.fr/hal-01824099>

- [15] G. MANGEON, S. BENHAMADOUICHE, J.-F. WALD, R. MANCEAU. *Modelling of the dissipation rate of the temperature variance*, in "ETMM12 - 12th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements", Montpellier, France, September 2018, <https://hal.inria.fr/hal-01944242>
- [16] G. MANGEON, S. BENHAMADOUICHE, J.-F. WALD, R. MANCEAU. *Unifying the near-wall treatment of the turbulent heat fluxes for all kinds of temperature boundary conditions with the Elliptic Blending approach*, in "THMT-18 - 9th International Symposium on Turbulence, Heat and Mass Transfer", Rio De Janeiro, Brazil, July 2018, <https://hal.inria.fr/hal-01944249>

Conferences without Proceedings

- [17] A. H. AFAILAL, J. GALPIN, A. VELGHE, R. MANCEAU. *A Hybrid Temporal RANS-LES Method for internal combustion engine applications*, in "Large-Eddy Simulation for Internal Combustion Engines", Rueil-Malmaison, France, December 2018, <https://hal.inria.fr/hal-01970768>
- [18] P. BRUEL. *An efficient pressure-based methodology for low Mach flow simulations*, in "CAIA 2018: 5° Congreso Argentino de Ingeniería Aeronáutica", Córdoba, Argentina, November 2018, <https://hal.archives-ouvertes.fr/hal-01953245>
- [19] J. JUNG, V. PERRIER. *A low Mach correction able to deal with low Mach acoustic and free of checkerboard modes*, in "ECCM-ECFD Conferences 2018 - 6th European Conference on Computational Mechanics - 7th European Conference on Computational Fluid Dynamics", Glasgow, United Kingdom, June 2018, <https://hal.inria.fr/hal-01953376>
- [20] V. PERRIER, J. JUNG. *A low Mach correction able to deal with low Mach acoustic and free of checkerboard modes*, in "CANUM 2018 - 44e Congrès National d'Analyse Numérique", Cap d'Agde, France, May 2018, vol. 228, pp. 2525 - 2615, <https://hal.inria.fr/hal-01960122>
- [21] V. PERRIER, A. MAZAHERI. *Symmetrizable first order formulation of Navier-Stokes equations and numerical results with the discontinuous Galerkin method*, in "6th European Conference on Computational Mechanics (ECCM 6)/7th European Conference on Computational Fluid Dynamics (ECFD 7)", Glasgow, France, June 2018, <https://hal.inria.fr/hal-01953594>

Scientific Popularization

- [22] P. BRUEL. *Transition énergétique: d'où partons-nous ?*, May 2018, Opération "Savoir en partage" avec Lacq Odyssée. Lycée du Pays de Soule. Chéraute. France, <https://hal.archives-ouvertes.fr/hal-01953296>
- [23] P. BRUEL. *Transition énergétique: d'où partons-nous ?*, May 2018, Café des Sciences. Médiathèque de Cambo les Bains. France, <https://hal.archives-ouvertes.fr/hal-01953284>
- [24] J. JUNG, V. PERRIER. *Le calcul à haute performance : aujourd'hui: enjeux et applications*, May 2018, Savoir en partage, Lycée Louis Barthou, <https://hal.inria.fr/hal-01960160>
- [25] J. JUNG, V. PERRIER. *Le calcul à haute performance : aujourd'hui: enjeux et applications*, June 2018, Café des Sciences, Médiathèque des Gaves, Oloron Sainte-Marie, <https://hal.inria.fr/hal-01960216>
- [26] J. JUNG, V. PERRIER. *Le calcul à haute performance : aujourd'hui: enjeux et applications*, November 2018, Recherche et Innovations, UPPA, Pau, <https://hal.inria.fr/hal-01953396>

- [27] J. JUNG, V. PERRIER. *Le calcul à haute performance aujourd'hui : enjeux et applications*, May 2018, Savoir en partage, Collège de Salies-de-Béarn, France, <https://hal.inria.fr/hal-01953388>
- [28] R. MANCEAU. *La simulation numérique en physique*, May 2018, Savoir en partage, Lycée Albert Camus, Mourenx, France, <https://hal.inria.fr/hal-01944279>
- [29] R. MANCEAU. *Le numérique en physique*, June 2018, Café des sciences, collège du Vic-Bilh, Lembeye, France, <https://hal.inria.fr/hal-01944266>
- [30] R. MANCEAU. *Le numérique en physique*, June 2018, Café des sciences, médiathèque de Billère, France, <https://hal.inria.fr/hal-01944273>

Other Publications

- [31] V. DUFFAL, B. DE LAAGE DE MEUX, R. MANCEAU. *Hybrid RANS/LES modelling of unsteady turbulent loads in hydraulic pumps*, May 2018, Code_Saturne user meeting, Poster, <https://hal.inria.fr/hal-01944333>
- [32] J. JUNG. *A low Mach correction able to deal with low Mach acoustic and free of checkerboard modes*, May 2018, Séminaire, Groupe de travail de l'ENS Rennes, Rennes, France, <https://hal.inria.fr/hal-01953411>
- [33] G. MANGEON, S. BENHAMADOUCHE, R. MANCEAU, J.-F. WALD. *Modeling of the dissipation rate of the temperature variance*, May 2018, Code_Saturne user meeting, Poster, <https://hal.inria.fr/hal-01944358>

References in notes

- [34] 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
- [35] 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
- [36] C. AUGONNET, S. THIBAUT, R. NAMYST, P.-A. WACRENIER. *StarPU: A Unified Platform for Task Scheduling on Heterogeneous Multicore Architectures*, in "Concurr. Comput. : Pract. Exper.", February 2011, vol. 23, n^o 2, pp. 187–198, <http://dx.doi.org/10.1002/cpe.1631>
- [37] 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>
- [38] 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
- [39] 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>

- [40] 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
- [41] B. COCKBURN, S. HOU, C.-W. SHU. *The Runge-Kutta local projection discontinuous Galerkin finite element method for conservation laws. IV. The multidimensional case*, in "Math. Comp.", 1990, vol. 54, n^o 190, pp. 545–581, <http://dx.doi.org/10.2307/2008501>
- [42] 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
- [43] 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>
- [44] 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
- [45] 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>
- [46] S. S. COLIS. *Discontinuous Galerkin methods for turbulence simulation*, in "Proceedings of the Summer Program", Center for Turbulence Research, 2002
- [47] M. ESSADKI, J. JUNG, A. LARAT, M. PELLETIER, V. PERRIER. *A task-driven implementation of a simple numerical solver for hyperbolic conservation laws*, in "ESAIM: Proceedings and Surveys", January 2017, vol. 63, pp. 228–247 [DOI : 10.1051/PROC/201863228], <https://hal.archives-ouvertes.fr/hal-01439322>
- [48] 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>
- [49] U. FRISCH. *Turbulence: The Legacy of AN Kolmogorov*, Cambridge University Press, 1995
- [50] M. GILES. *Non-Reflecting Boundary Conditions for Euler Equation Calculation*, in "The American Institute of Aeronautics and Astronautics Journal", 1990, vol. 42, n^o 12
- [51] 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
- [52] A. JAMESON, M. FATICA. *Using Computational Fluid Dynamics for Aerodynamics*, in "National Research Council Workshop on "The Future of Supercomputing"", 2003

- [53] 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>
- [54] A. KLÖCKNER, T. WARBURTON, J. BRIDGE, J. HESTHAVEN. *Nodal discontinuous Galerkin methods on graphics processors*, in "Journal of Computational Physics", 2009, vol. 228, n^o 21, pp. 7863 - 7882 [DOI : 10.1016/J.JCP.2009.06.041], <http://www.sciencedirect.com/science/article/pii/S0021999109003647>
- [55] D. KNOLL, D. KEYES. *Jacobian-free Newton-Krylov methods: a survey of approaches and applications*, in "Journal of Computational Physics", 2004, vol. 193, n^o 2, pp. 357 - 397 [DOI : 10.1016/J.JCP.2003.08.010], <http://www.sciencedirect.com/science/article/pii/S0021999103004340>
- [56] P. LESAIN, 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, pp. 89–123. Publication No. 33
- [57] 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>
- [58] A. C. MURESAN, Y. NOTAY. *Analysis of Aggregation-Based Multigrid*, in "SIAM J. Sci. Comput.", March 2008, vol. 30, n^o 2, pp. 1082–1103, <http://dx.doi.org/10.1137/060678397>
- [59] T. POINSOT, S. LELE. *Boundary conditions for direct simulations of compressible viscous flows*, in "Journal of Computational Physics", 1992, vol. 101, pp. 104-129
- [60] W. REED, T. HILL. *Triangular mesh methods for the neutron transport equation*, Los Alamos Scientific Laboratory, 1973, n^o LA-UR-73-479
- [61] H. SUTTER. *The free lunch is over: A fundamental turn toward concurrency in software*, in "Dr. Dobb's Journal", 2005
- [62] K. W. THOMPSON. *Time-dependent boundary conditions for hyperbolic systems, {II}*, in "Journal of Computational Physics", 1990, vol. 89, n^o 2, pp. 439 - 461 [DOI : 10.1016/0021-9991(90)90152-Q], <http://www.sciencedirect.com/science/article/pii/002199919090152Q>
- [63] I. TOULOPOULOS, J. A. EKATERINARIS. *Artificial boundary conditions for the numerical solution of the Euler equations by the discontinuous galerkin method*, in "Journal of Computational Physics", 2011, vol. 230, n^o 15, pp. 5974 - 5995 [DOI : 10.1016/J.JCP.2011.04.008], <http://www.sciencedirect.com/science/article/pii/S0021999111002324>
- [64] P. WESSELING. *An introduction to multigrid methods*, Pure and applied mathematics, J. Wiley, Chichester, New York, 1992, <http://opac.inria.fr/record=b1088946>
- [65] I. YAVNEH. *Why Multigrid Methods Are So Efficient*, in "Computing in Science and Engg.", November 2006, vol. 8, n^o 6, pp. 12–22, <http://dx.doi.org/10.1109/MCSE.2006.125>

-
- [66] J. VAN DER VEGT, S. RHEBERGEN. *hp-Multigrid as Smoother algorithm for higher order discontinuous Galerkin discretizations of advection dominated flows: Part I. Multilevel analysis*, in "Journal of Computational Physics", 2012, vol. 231, n^o 22, pp. 7537 - 7563 [DOI : 10.1016/j.jcp.2012.05.038], <http://www.sciencedirect.com/science/article/pii/S0021999112003129>