

Inria

IN PARTNERSHIP WITH:
CNRS

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

Activity Report 2019

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 parameter free pressure based approach for simulating flows at all Mach	12
7.2. Simulation of a jet in a supersonic cross-flow	12
7.3. Experiments and simulations related to flows over aerial tanks	12
7.4. A density-based numerical flux for high order simulation of low Mach flows with acoustics	13
7.5. Conjugate heat transfer with different fluid-solid physical properties: a new elliptic blending second-moment closure	13
7.6. Eddy-viscosity models sensitized to buoyancy effects for predicting natural convection flows	14
7.7. Development and validation of an improved HTLES approach	14
7.8. Compact WENO, positivity preserving stabilization of discontinuous Galerkin methods	14
8. Bilateral Contracts and Grants with Industry	14
8.1. Bilateral Contracts with Industry	14
8.2. Bilateral Grants with Industry	14
9. Partnerships and Cooperations	15
9.1. Regional Initiatives	15
9.1.1. SEIGLE	15
9.1.2. HPC scalable ecosystem	15
9.2. National Initiatives	15

9.2.1. GIS Success	15
9.2.2. ANR MONACO_2025	15
9.3. European Initiatives	16
9.4. International Initiatives	17
9.4.1. Informal International Partners	17
9.4.2. Participation in International Programs	18
9.5. International Research Visitors	18
10. Dissemination	18
10.1. Promoting Scientific Activities	18
10.1.1. Member of the Organizing Committees	18
10.1.2. Journal	18
10.1.2.1. Member of the Editorial Boards	18
10.1.2.2. Reviewer - Reviewing Activities	18
10.1.3. Invited Talks	19
10.1.4. Leadership within the Scientific Community	19
10.1.5. Scientific Expertise	19
10.1.6. Research Administration	19
10.2. Teaching - Supervision - Juries	19
10.2.1. Teaching	19
10.2.2. Supervision	20
10.2.3. Juries	20
11. Bibliography	20

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 [Team leader, 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 Fellows

- Enrique Gutierrez Alvarez [Inria, Post-Doctoral Fellow]
- Franck Mastrippolito [CNRS, Post-Doctoral Fellow]

PhD Students

- Al Hassan Afailal [IFPEN, PhD Student]
- Puneeth Bikkanahally Muni Reddy [Univ de Pau et des pays de l'Adour, PhD Student]
- Vladimir Duffal [EDF, PhD Student]
- Gaetan Mangeon [EDF, PhD Student]
- Gustave Sporshill [Dassault Aviation, PhD Student]
- Jameel Syed Mohd Saad [PSA, PhD Student]

Intern and Apprentice

- Mauricio Garcia Zulch [Inria, until Apr 2019]

Administrative Assistant

- Sylvie Embolla [Inria, Administrative Assistant]

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 [46] 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 [57] and first studied by Lesaint and Raviart [53]. 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 [41], [40], one dimensional systems [39], multidimensional scalar conservation laws [38], and multidimensional systems [42]. For the same system, we can also cite the work of [45], [50], 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 [31]. The first application of discontinuous Galerkin methods to Navier-Stokes equations was proposed in [36] by mean of a mixed

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

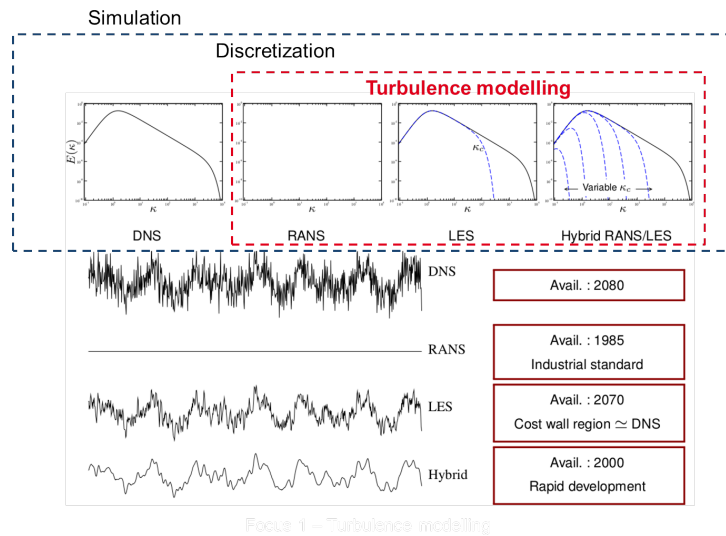


Figure 1. Schematic view of the different nested steps for turbulent flow simulation: from DNS to hybrid RANS-LES. The approximate dates at which the different approaches are or will be routinely used in the industry are indicated in the boxes on the right (extrapolations based on the present rate of 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 [37], which was later analyzed, and proved to be stable in a more unified framework [32]. The combination with the $k - \omega$ RANS model was made in [35]. As far as Navier-Stokes equations are concerned, we can also cite the work of [48], in which the stabilization is closer to the one of [32], the work of [54] on local time stepping, or the first use of discontinuous Galerkin methods for direct numerical simulation of a turbulent channel flow done in [43]. 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 [34]. 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 [56], [59] 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 [47]. We will also try to extend the work of [60], 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 [52]. 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 [61], [62] 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) [55]. To this aim, we propose to extend the scalar analysis of [63] 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 [34].

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 [58]. Already in 2003, [49] 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 [51], 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 [33]. 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 Afaïl, co-supervised by Rémi Manceau, is focused on this issue.

5. Highlights of the Year

5.1. Highlights of the Year

5.1.1. R. Manceau, new Cagire team leader

After having taken over the responsibility of the team since its creation, Pascal Bruel, 59, has decided this year to hand over the reins! After consultation, Inria's management has appointed Remi Manceau as the new head of the Cagire team as of 18 November 2019.

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 2019, the following points were addressed in AeroSol

- *Update, documentation, and wiki for the test case
- *exact solution of Riemann problem and exact Godunov solver
- *Development of Droplet model, and of a Baer and Nunziato diphasic model.
- *Beginning of implementation of eddy viscosity models (k-epsilon, Spalart-Almarras) turbulence models.
- *Add the possibility of mesh dependent data (for example, a flow computed by AeroSol with the Euler system) for being used as input for another model (e.g. advection of droplets within this flow). This feature is used also for wall distance for turbulent models.
- *Penalization problems, with single core mesh adaptation was merged in the master branch.

*Improvements of PETSc usage: possibility of solving linear problems that are not of size $nvar$, usage of MUMPS LU solver through PETSc.

*Interfacing with SLEPc for solving eigenvalues and eigenvectors problems.

*High order visualization based on GMSH.

*Beginning of interfacing with PARMMG for parallel mesh adaptation.

*Clean of warning, error messages, etc...

- Participants: Benjamin Lux, Damien Genet, Mario Ricchiuto, Vincent Perrier, H elo ise Beaugendre, Subodh Madhav Joshi, Christopher Poette, Marco Lorini, Jonathan Jung and Enrique Gutierrez Alvarez
- Partner: BRGM
- Contact: Vincent Perrier

7. New Results

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

Participant: Pascal Bruel.

The latest version of the pressure-correction algorithm developed in the last years in close partnership with Prof. E. Dick (Ghent University, Belgium) and Dr. Y. Moguen (UPPA, France) has been published this year [14]. Beyond its promotion [25], future efforts will be directed towards its extension to reacting single phase flows.

7.2. Simulation of a jet in a supersonic cross-flow

Participant: Pascal Bruel.

In our joint work, published this year with our Kazakh colleagues [10], we have focused on detailing the vortical patterns present in the structure of a sonic jet injected normally into a supersonic crossflow. Such a generic flow configuration is considered to feature some prominent characteristics encountered in propulsion system based on supersonic combustion (scramjet). A critical pressure ratio value beyond which new vortical structures are appearing has been evidenced. Contacts have been taken with ONERA to have access to their experimental database which is much more developed than the one we had access to so far. Some specific adaptation will have to be conducted though, in order to adapt the simulation to the slightly different flow configuration considered by ONERA.

7.3. Experiments and simulations related to flows over aerial tanks

Participant: Pascal Bruel.

These results have been obtained in the framework of the cooperation with the National University of C ordoba (Argentina).

- Predicting the pressure loads produced by the atmospheric wind flow over a cylindrical vertical tank
Pressure distributions obtained using different RANS turbulence models were compared with experimental data obtained in wind tunnel tests for a tank with a closed roof. We have considered a flat shape and a conical roof of 25 degrees. Combinations of aspect ratios equal to 0.5, 1 and 2, and Reynolds numbers equal to 250000, 290000 and 340000 were simulated. The results of the numerical model were compared with those obtained in experimental tests in a wind tunnel of the atmospheric boundary layer using a rigid tank model. We have worked to obtain a stable atmospheric boundary layer in the complete domain using the correct boundary conditions for the implemented RANS models. After that, we have developed numerical simulations for the flow around the tanks inside the atmospheric boundary layer. These results have been published in [17].
- Studying the interaction of a wall with the flow around two cylinders arranged in tandem

For this configuration, the cylinders were immersed in a flow with a boundary layer profile at a subcritical Reynolds number ($Re=10000$). The three-dimensional transient turbulent flow around the cylinders was simulated numerically using the SAS turbulence model. The effects of wake interference due to both the proximity between the cylinders and their position with respect to the wall were examined through the values of drag, lift and pressure coefficients. The details of the flow fields in the near wake of the cylinders were also studied. The results were compared with experimental and numerical results reported in the literature, and with the case of a single cylinder near a wall. These results have been published in [15]. In parallel, a specific test section for the team's Maveric test facility has been developed. It gives the possibility to accommodate wall mounted cylinder(s) that represent a scaled down version of real horizontal tanks. The objective here is to generate validation data. Particle image velocimetry (PIV) measurements have been carried out during the 1-month stay of Mauro Grioni in Pau in September 2019.

- Simulating the effects of explosions on liquid fuel storage tanks.

The fast release of energy in explosive processes produces intense shock waves (blast waves). The interaction of these waves with obstacles such as tanks can be extremely destructive. As a first step towards the full simulation of a blast wave with a tank, we have studied the capabilities of OpenFOAM to simulate a blast wave. The numerical results were compared in a cylindrical configuration with the analytical solution provided by the Sedov theory. A special attention has been paid to evaluate the influence of the reconstruction functions in the Euler flux (Kurganov scheme) on the numerical results. The predicted position and velocity of the generated shock wave as well as the pressure jump and its evolution behind the shock were in good agreement with their theoretical counterparts. These results have been published in [16].

7.4. A density-based numerical flux for high order simulation of low Mach flows with acoustics

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. These results have been published in [12].

7.5. Conjugate heat transfer with different fluid-solid physical properties: a new elliptic blending second-moment closure

Participants: Rémi Manceau, Gaëtan Mangeon.

Our approach to model the wall/turbulence interaction, based on Elliptic Blending, was successfully applied to flows with standard thermal boundary conditions at the walls [11]. However, Conjugate Heat Transfer, which couples fluid and solid domains, are particularly challenging for turbulence models. We have developed an innovative model, the Elliptic Blending Differential Flux Model, to account for the influence of various wall thermal boundary conditions on the turbulent heat flux and the temperature variance. An assessment of this new model in Conjugate Heat Transfer has been performed for several values of fluid-solid thermal diffusivity and conductivity ratios. A careful attention is paid to the discontinuity of the dissipation rate associated with the temperature variance at the fluid-solid interface. The analysis is supported by successful comparisons with Direct Numerical Simulations [30].

7.6. Eddy-viscosity models sensitized to buoyancy effects for predicting natural convection flows

Participants: Rémi Manceau, Saad Jameel.

Eddy-viscosity turbulence models have been sensitized to the effects of buoyancy, in order to improve the prediction in natural convection flows. The approach extends in a linear way the constitutive relations for the Reynolds stress and the turbulent heat flux, in order to account for the anisotropic influence of buoyancy. The novelty of this work involves the buoyancy extension applied to two very different eddy-viscosity models, which leads to encouraging results for the highly challenging case of the differentially heated vertical channel [27], [22].

7.7. Development and validation of an improved HTLES approach

Participants: Rémi Manceau, Vladimir Duffal, Hassan Afailal, Franck Mastrippolito, Pascal Bruel.

The HTLES (hybrid temporal LES) approach, developed by the team, has been improved by introducing shielding functions and an internal consistency constraint to enforce the RANS behavior in the near-wall regions [18]. The influence of the underlying closure model was studied by applying HTLES to two RANS models: the $k-\omega$ SST and the BL- v^2/k . The validation, carried out using two different solvers, Code_Saturne (collaboration with EDF) and Converge-CFD (collaboration with IFPEN), encompassed different type of internal flows: channel, periodic hill, internal combustion engines (in fixed position and with a moving piston), jet in crossflow [21], [9]. The robustness of HTLES to grid coarsening and the accuracy of the predictions at a reduced numerical cost compared to LES was demonstrated. Notably, the capacity of HTLES to provide information on velocity and pressure temporal fluctuations at the wall was assessed, offering a cost-saving alternative to LES to predict unsteady loads. One of the major limitation of HTLES, i.e., the fact that it is based on statistically averaged quantities, which are challenging to estimate in non-stationary flows, has been removed, by approximating the statistical average by a Dynamic Temporal Filter.

7.8. Compact WENO, positivity preserving stabilization of discontinuous Galerkin methods

Participant: Vincent Perrier.

Jointly with Alireza Mazaheri (NASA Langley) and Chi Wang Shu, we have developed a compact WENO stabilization that moreover ensures the positivity of physical quantities. The work was published in [13].

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.

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 [44] 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. This year, the evaluation of the capability of the compressible module of Yales2 has started.

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: Pascal Bruel, Rémi Manceau, Franck Mastrippolito.

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-NOx 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 NOx 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. Franck Matrippolito, the post-doc recruited by mid-january 2019, performed simulations aimed at reproducing the experiment of ONERA Toulouse carried out in the same workpackage. The configuration is that of an effusion plate with a gyration angle of 90 degrees and the turbulence model is EBRSM. Franck presented his results in October 2019 during the ITR meeting in Florence (Italy).

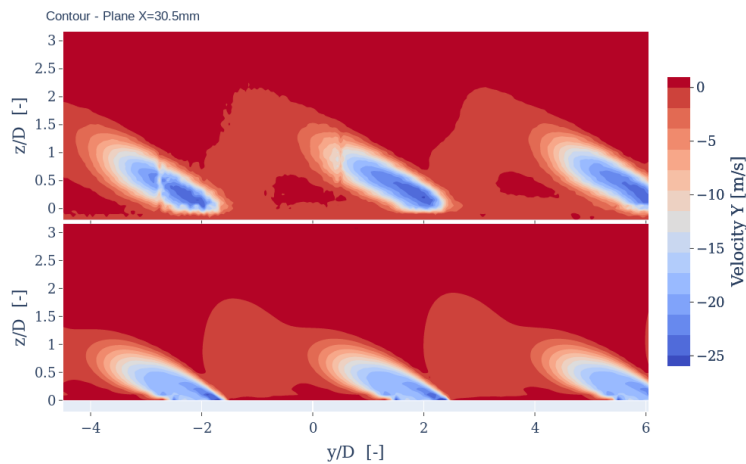


Figure 2. Simulation of the ONERA SOPRANO configuration: example of experimental (top) vs numerical (bottom) results concerning the mean velocity field.

9.4. International Initiatives

9.4.1. Informal International Partners

- Institute of Mathematics and Mathematical Modelling, Almaty, Kazakhstan
Participant: Pascal Bruel.

Collaboration with Drs A. Beketaeva and A. Naïmanova for the RANS simulations of a supersonic jet in crossflow configuration for a wide range of pressure ratio ([10]). This year, Pascal Bruel spent two weeks in Almaty in the framework of this partnership.

- University of Evora, Evora, Portugal

Participant: Pascal Bruel.

Collaboration with Dr. P. Correia related this year to the partial rewriting of a Fortran code implementing a pressure-based approach for simulating low Mach flows as well as to the promotion of such a pressure-based approach ([25]). This year, Pascal Bruel spent 5 days in Evora in the framework of this partnership.

- University of Ghent, Ghent, Belgium

Participant: Pascal Bruel.

Collaboration with Prof. E. Dick related to the development and the promotion of a pressure-based approach for simulating low Mach and all-Mach flows. ([25], [14])

9.4.2. Participation in International Programs

- National University of Córdoba (UNC), Córdoba, Argentina: ECOS-Sud A17A07 project

Participant: Pascal Bruel.

2019 was the second year of this project devoted to the simulations of the wind around aerial fuel tanks and related experiments. Pascal Bruel spent two weeks at UNC in the framework of this project.

9.5. International Research Visitors

- Prof. Sergio Elaskar (2 weeks) and PhD student Mauro Grioni (1 month) from University of Córdoba (Argentina) visited the team in the framework of the A17A07 Ecos-Sud project.
- Dr. Paulo Correia from University of Evora spent two weeks in the team in May 2019.

9.5.1. Visits of International Scientists

9.5.1.1. Internships

Mauricio Garcia Zulch from Chile spent 3 months in the team.

10. Dissemination

10.1. Promoting Scientific Activities

10.1.1. Member of the Organizing Committees

- Organizer and scientific chair of the mini-symposium "Numerical method for multi-scale fluid problems" at ICIAM 2019 [JJ].

10.1.2. Journal

10.1.2.1. 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]

10.1.2.2. Reviewer - Reviewing Activities

During 2019, the team members reviewed papers for the following journals:

- AIAA Journal [PB, RM]
- Computer & Fluids [RM]
- Int. J. Heat Fluid Flow [RM]
- Journal of Computational Physics [JJ]
- J. Hydr. Res. [RM]
- Physics of Fluids [FM]
- Phys. Rev. Fluids [RM]
- SIAM Journal on Scientific Computing [VP]

10.1.3. Invited Talks

- J. Jung [19]
- R. Manceau [20]

10.1.4. Leadership within the Scientific Community

- Rémi Manceau co-organizes the activities of the Special Interest Group 15 (Turbulence modelling) of ERCOFTAC (European Research Community on Flow, Turbulence and Combustion) as a member of the Steering Committee. The main activity of this group in 2019 was the organization of a workshop in Ljubljana, Slovenia.
- Rémi Manceau coordinates the ANR Project MONACO_2025, a 4-year project started in 2018. The partners are: the institute PPrime, PSA Group and EDF.

10.1.5. Scientific Expertise

- Evaluation of one Ecos Sud project [PB]

10.1.6. 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].
- 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 Inria evaluation committee, and member of the board. 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.

10.2. Teaching - Supervision - Juries

10.2.1. Teaching

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

Licence : [JJ], Scientific computing, 40.5h, L2 - Informatique, Université de Pau et des Pays de l'Adour, Pau, France.

Licence : [JJ], Numerical analysis for vectorial problems, 33.75h, L2 - Mathematics, 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], Finite volume methods for hyperbolic systems, 15h, Master ANEDP, ENS, Casablanca, Maroc.

Master: [VP], Numerical analysis of PDE 1, Master MMS, Pau.

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]

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

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):

- Thomas Kaiser, "Impact of flow rotation on flame dynamics and hydrodynamic stability", University of Toulouse (France), 31 January 2019. Supervisor: T. Poinso [PB, Referee].
- Joao Rodrigo Andrade, "Spectral analysis of the turbulent energy cascade and the development of a novel nonlinear subgrid-scale model for large-eddy simulation", Universidade Federale de Uberlândia (Brazil) and University of Lille (France), 27 March 2019. Supervisors: A. S. Neto, G. Mompean and R.L. Thompson [RM, Referee]
- Adithya Ramanathan Krishnan, "Explicit algebraic subfilter scale modeling for DES-like methods and extension to variable density flows", University of Aix-Marseille 3 April 2019. Supervisor: P. Sagaut [RM, Referee]
- Benjamin Lorendeau, "Amélioration des performances via un parallélisme multi-niveaux sur un code CFD en maillages non structurés", University of Bordeaux (France), 16 December 2019. Supervisor: E. Jeannot [PB, Referee]

11. Bibliography

Major publications by the team in recent years

- [1] P. BRUEL, S. DELMAS, J. JUNG, V. PERRIER. *A low Mach correction able to deal with low Mach acoustics*, in "Journal of Computational Physics", 2019, vol. 378, pp. 723–759, <https://hal.inria.fr/hal-01953424>
- [2] 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>
- [3] 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>

- [4] 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>
- [5] 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]
- [6] J.-M. HÉRARD, J. JUNG. *An interface condition to compute compressible flows in variable cross section ducts*, in "Comptes Rendus Mathématiques", February 2016 [DOI : 10.1016/J.CRMA.2015.10.026], <https://hal.inria.fr/hal-01233251>
- [7] 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>
- [8] 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

- [9] A. H. AFAILAL, J. GALPIN, A. VELGHE, R. MANCEAU. *Development and validation of a hybrid temporal LES model in the perspective of applications to internal combustion engines*, in "Oil & Gas Science and Technology - Revue d'IFP Energies nouvelles", 2019, vol. 74, 56 p. [DOI : 10.2516/OGST/2019031], <https://hal-ifp.archives-ouvertes.fr/hal-02162596>
- [10] A. BEKETAEVA, P. BRUEL, A. Z. NAIMANOVA. *Detailed Comparative Analysis of Interaction of a Supersonic Flow with a Transverse Gas Jet at High Pressure Ratios*, in "Technical Physics / Zhurnal Tekhnicheskoi Fiziki", October 2019, vol. 64, n^o 10, pp. 1430-1440 [DOI : 10.1134/S1063784219100049], <https://hal.inria.fr/hal-02373910>
- [11] S. BENHAMADOUCHE, I. AFGAN, R. MANCEAU. *Numerical simulations of flow and heat transfer in a wall-bounded pin matrix*, in "Flow, Turbulence and Combustion", 2019 [DOI : 10.1007/s10494-019-00046-8], <https://hal.archives-ouvertes.fr/hal-02179021>
- [12] 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>
- [13] A. MAZAHERI, C.-W. SHU, V. PERRIER. *Bounded and compact weighted essentially nonoscillatory limiters for discontinuous Galerkin schemes: Triangular elements*, in "Journal of Computational Physics", October 2019, vol. 395, pp. 461-488 [DOI : 10.1016/J.JCP.2019.06.023], <https://hal.archives-ouvertes.fr/hal-02321400>
- [14] Y. MOGUEN, P. BRUEL, E. DICK. *A combined momentum-interpolation and advection upstream splitting pressure-correction algorithm for simulation of convective and acoustic transport at all levels of Mach number*, in "Journal of Computational Physics", May 2019, vol. 384, pp. 16-41 [DOI : 10.1016/J.JCP.2019.01.029], <https://hal.inria.fr/hal-02064848>

Articles in National Peer-Reviewed Journals

- [15] M. GRIONI, S. ELASKAR, A. MIRASSO, P. BRUEL. *Flow interference between circular cylinders in tandem arrangement near to a plane wall*, in "Mecánica Computacional", November 2019, vol. 37, n^o 26, pp. 1065-1074, <https://hal.inria.fr/hal-02376562>
- [16] L. GUTIÉRREZ MARCANTONI, S. ELASKAR, J. TAMAGNO, P. BRUEL. *Simulation of blast waves using OpenFOAM*, in "Mecánica Computacional", November 2019, vol. 37, n^o 26, pp. 1075-1084, <https://hal.inria.fr/hal-02376633>
- [17] J. SALDÍA, G. KRAUSE, S. ELASKAR, P. BRUEL. *Modelizacion numerica de cargas de viento sobre un tanque de almacenamiento de combustible*, in "Mecánica Computacional", November 2019, vol. 37, n^o 27, pp. 1163-1175, <https://hal.inria.fr/hal-02376676>

Articles in Non Peer-Reviewed Journals

- [18] R. MANCEAU. *Hybrid temporal LES: Development and applications*, in "ERCOFTAC Bulletin", 2019, forthcoming, <https://hal.archives-ouvertes.fr/hal-02344854>

Invited Conferences

- [19] J. JUNG. *Quelques applications des mathématiques au centre Inria de Bordeaux Sud-Ouest*, in "Septième journée des Sciences de l'Ingénieur", Casablanca, Morocco, June 2019, <https://hal.inria.fr/hal-02283149>
- [20] R. MANCEAU. *Modelling of turbulent natural convection (keynote lecture)*, in "16th ERCOFTAC SIG15 Workshop on Modelling of wall bounded turbulent natural convection", Ljubljana, Slovenia, Jozef Stefan Institute (IJS), October 2019, <https://hal.archives-ouvertes.fr/hal-02319389>

International Conferences with Proceedings

- [21] V. DUFFAL, B. DE LAAGE DE MEUX, R. MANCEAU. *Development and validation of a hybrid RANS/LES approach based on temporal filtering*, in "ASME - JSME - KSME Joint Fluids Engineering Conference 2019", San Francisco, CA, United States, Proc. ASME - JSME - KSME Joint Fluids Engineering Conference 2019, Jul 2019, San Francisco, CA, United States., 2019, <https://hal.archives-ouvertes.fr/hal-02128250>
- [22] J. SYED MOHD SAAD, R. MANCEAU, V. HERBERT. *Sensitization of eddy-viscosity models to buoyancy effects for predicting natural convection flows*, in "HEFAT 2019 - 14th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics", Wicklow, Ireland, Proc. HEFAT 2019 - 14th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics, 2019, <https://hal.archives-ouvertes.fr/hal-02129153>

Conferences without Proceedings

- [23] J. JUNG, V. PERRIER. *Low Mach corrections and checkerboard modes*, in "ICIAM 2019 - International Congress on Industrial and Applied Mathematics", Valencia, Spain, July 2019, <https://hal.inria.fr/hal-02283173>
- [24] J. JUNG, V. PERRIER. *Low Mach schemes for acoustics*, in "ICIAM 2019 - International Congress on Industrial and Applied Mathematics", Valencia, Spain, July 2019, <https://hal.archives-ouvertes.fr/hal-02321427>

- [25] Y. MOGUEN, P. BRUEL, P. CORREIA, E. DICK. *Un algorithme pour la simulation du transport convectif et de la propagation acoustique à tout nombre de Mach*, in "24ème Congrès Français de Mécanique", Brest, France, August 2019, <https://hal-univ-pau.archives-ouvertes.fr/hal-02361166>
- [26] V. PERRIER, A. MAZAHERI. *Symmetrizable first order formulation of Navier-Stokes equations and numerical results with the discontinuous Galerkin method*, in "HONOM 2019 - European Workshop on High order nonlinear numerical methods for evolutionary PDEs: Theory and Applications", Madrid, Spain, April 2019, <https://hal.archives-ouvertes.fr/hal-02321418>
- [27] J. SYED MOHD SAAD, R. MANCEAU, V. HERBERT. *A buoyancy extension for eddy-viscosity models for the natural convection regime*, in "17th European Turbulence Conference (ETC-2019)", Torino, Italy, September 2019, <https://hal.archives-ouvertes.fr/hal-02305788>

Other Publications

- [28] P. BRUEL. *Pressure-based methodologies for zero Mach and low Mach flow simulations*, September 2019, Workshop « Non Linear Phenomena and Dynamics of Flame Propagation », <https://hal.inria.fr/hal-02379004>
- [29] V. DUFFAL, R. MANCEAU, B. DE LAAGE DE MEUX. *Hybrid RANS/LES modelling of unsteady turbulent loads in hydraulic pumps. A hybrid approach based on temporal filtering.*, May 2019, Code_Saturne user meeting, Poster, <https://hal.archives-ouvertes.fr/hal-02305766>
- [30] G. MANGEON, S. BENHAMADOUCHE, R. MANCEAU, J.-F. WALD. *Conjugate heat transfer: A big challenge for RANS modeling*, May 2019, Code_Saturne user meeting, Poster, <https://hal.archives-ouvertes.fr/hal-02128551>

References in notes

- [31] 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
- [32] 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
- [33] 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>
- [34] 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>
- [35] 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

- [36] 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>
- [37] 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
- [38] 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>
- [39] 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
- [40] 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>
- [41] 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
- [42] 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>
- [43] S. S. COLIS. *Discontinuous Galerkin methods for turbulence simulation*, in "Proceedings of the Summer Program", Center for Turbulence Research, 2002
- [44] 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>
- [45] 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>
- [46] U. FRISCH. *Turbulence: The Legacy of AN Kolmogorov*, Cambridge University Press, 1995
- [47] 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
- [48] 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

- [49] A. JAMESON, M. FATICA. *Using Computational Fluid Dynamics for Aerodynamics*, in "National Research Council Workshop on "The Future of Supercomputing"", 2003
- [50] 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>
- [51] 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>
- [52] 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>
- [53] 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
- [54] 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>
- [55] 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>
- [56] T. POINSOT, S. LELE. *Boundary conditions for direct simulations of compressible viscous flows*, in "Journal of Computational Physics", 1992, vol. 101, pp. 104-129
- [57] W. REED, T. HILL. *Triangular mesh methods for the neutron transport equation*, Los Alamos Scientific Laboratory, 1973, n^o LA-UR-73-479
- [58] H. SUTTER. *The free lunch is over: A fundamental turn toward concurrency in software*, in "Dr. Dobbs's Journal", 2005
- [59] 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>
- [60] 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>
- [61] P. WESSELING. *An introduction to multigrid methods*, Pure and applied mathematics, J. Wiley, Chichester, New York, 1992, <http://opac.inria.fr/record=b1088946>

- [62] 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>
- [63] 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>