Activity Report 2012

Team CAGIRE

Computational Approximation with discontinous Galerkin methods and compaRison with Experiments

RESEARCH CENTER
Bordeaux - Sud-Ouest

THEME
Computational models and simulation

Table of contents

Team CAGIRE

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

Creation of the Team: June 01, 2011 .

1. Members

Research Scientists

Pascal Bruel [Team leader, Researcher @CNRS, HdR]

Vincent Perrier [Researcher @Inria]

Faculty Members

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

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

Engineer

Maxime Mogé

Post-Doctoral Fellow

Yann Moguen [Since November 12, 2012]

Administrative Assistant

Josy Baron [shared with another team]

2. Overall Objectives

2.1. Overall Objectives

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

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

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

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

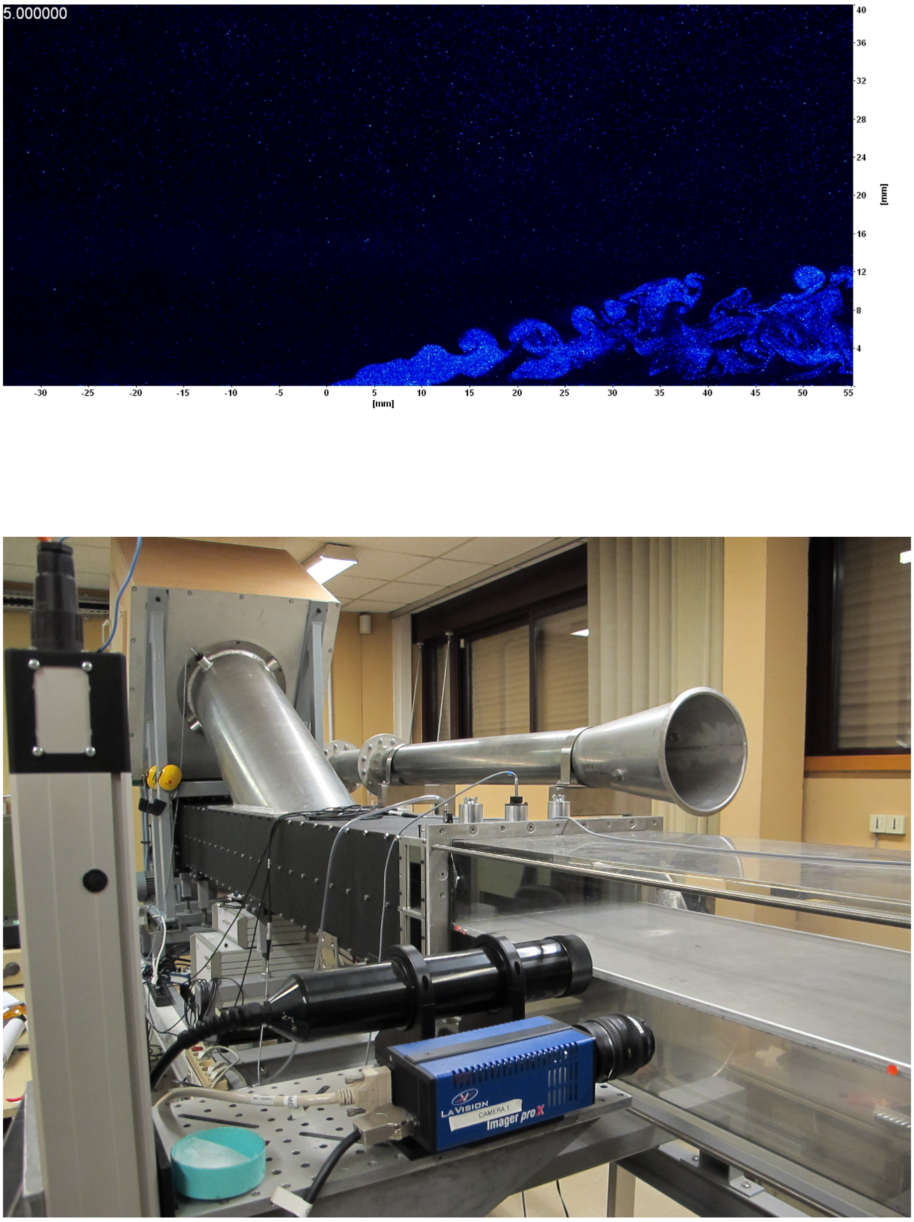

Figure 1. MAVERIC test facility: visualization of a single jet in crossflow (top) and overview of the test rig (bottom).

3. Scientific Foundations

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

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

3.2. Computational fluid mechanics: numerical methods

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

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

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

The finite volumes methods have imposed themselves in the last thirty years in the context of aerodynamic. They intrinsically contain an upwinding mechanism, so that they are naturally stable for linear as much as for nonlinear convective flows. The extension to diffusive flows has been done in [11]. Whereas the extension to second order with the MUSCL method is widely spread, the extension to higher order has always been a strong drawback of finite volumes methods. For such an extension, reconstruction methods have been developed (ENO, WENO). Nevertheless, these methods need to use a stencil that increases quickly with the order, which induces problems for the parallelisation and the efficiency of the implementation. Another natural extension of finite volume methods are the so-called discontinuous Galerkin methods. These methods are based on the Galerkin' idea of projecting the weak formulation of the equations on a finite dimensional space. But on the contrary to the conforming finite elements method, the approximation space is composed of functions that are continuous (typically: polynomials) inside each cell, but that are discontinuous on the sides. The discontinuous Galerkin methods are currently very popular, because they can be used with many sort of partial differential equations. Moreover, the fact that the approximation is discontinuous allows to use modern mesh adaptation (hanging nodes, which appear in non conforming mesh adaptation), and adaptive order, in which the high order is used only where the solution is smooth.

Discontinuous Galerkin methods where introduced by Reed and Hill [31] and first studied by Lesaint and Raviart [24]. The extension to the Euler system with explicit time integration was mainly led by Shu, Cockburn and their collaborators. The steps of time integration and slope limiting were similar to high order ENO schemes, whereas specific constraints given by the finite elements nature of the scheme were progressively solved, for scalar conservation laws [15], [14], one dimensional systems [13], multidimensional scalar conservation laws [12], and multidimensional systems [16]. For the same system, we can also cite the work of [18], [22], which is slightly different: the stabilisation is made by adding a nonlinear stabilisation term, and the time integration is implicit. Then, the extension to the compressible Navier-Stokes system was made by Bassi and Rebay [10], first by a mixed type finite element method, and then simplified by means of lifting operators. The extension to the $k-\omega$ RANS system was made in [9]. Another type of discontinuous Galerkin method for Navier Stokes is the so-called Symmetric Interior Penalty (SIP) method. It is used for example by Hartmann and Houston [20]. The symmetric nature of the discretization is particularly well suited with mesh adaptation by means of the adjoint equation resolution [21]. Last, we note that the discontinuous Galerkin method was already successfully tested in [17] at Direct Numerical Simulation scale for very moderate Reynolds, and also by Munz'team in Stuttgart [25], with local time stepping.

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

3.3. Experimental aspects

A great deal of experiments has been devoted to the study of jet in crossflow configurations. They essentially differ one from each other by the hole shape (cylindrical or shaped), the hole axis inclination, the way by which the hole is fed, the characteristics of the crossflow and the jet (turbulent or not, isothermal or not), the number of holes considered and last but not least the techniques used to investigate the flow. A good starting point to assess the diversity of the studies carried out is given by [26]. For inclined cylindrical holes, the experimental database produced by Gustafsson and Johansson [2] represents a sound reference base and for normal injection, the work by [32] served as reference for LES simulations [30]. For shaped holes, the studies are less numerous and are aimed at assessing the influence of the hole shape on various flow properties such as the heat transfer at the wall [23]. In 2007, A. Most developed at UPPA a test facility for studying jet in crossflow issued from shaped holes [27]. The hole shape was chosen as a 12.5 scale of the holes (i.e. at scale 1) drilled by laser in a

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

combustion chamber. His preliminary 2-component PIV results have been used to test RANS simulations [28] and LES [29]. This test facility will be used in the framework of the present project to investigate a 1-hole plane i.e. an isolated jet in crossflow. PIV and LDV metrology will be used.

4. Application Domains

4.1. Effusion cooling of aeronautical combustion chambers walls

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

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

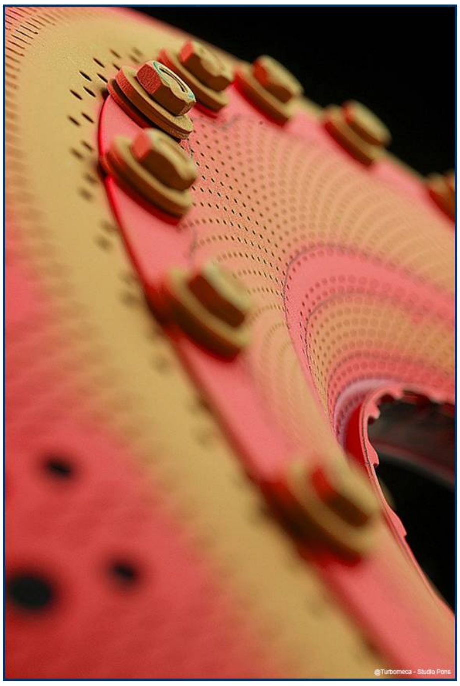

Figure 2. Effusion cooling of aeronautical combustion chambers: close view of a typical perforated chamber wall

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

- The jets of cool air should not mix too much with the internal flow. Otherwise it will decrease the efficiency of the combustion.
- The jets should be as much stable as possible when submitted to waves emitted in the combustion chamber, e.g. acoustic waves induced by combustion instabilities. Otherwise the jets may not cool enough the walls of the combustion chamber which can then undergoes severe damages.

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

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

A last aspect, that will not be dealt with in this project, but that could be dealt with in the future, is the interaction between the flow and the wall. The aim is to understand the effect of coupling between the heat propagation in the wall and the flow near the wall. A careful study of this interaction can allow to determine the exchange coefficients, and so the efficiency of the cooling by the jet. Such determination may particularly useful to develop one or multidimensional models of wall-fluid interaction [19].

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

5. Software

5.1. AeroSol

Participants: Dragan Amenga-Mbengoué [Bacchus], Damien Genet [Bacchus], Maxime Mogé, Francois Pellegrini [Bacchus], Vincent Perrier [correspondant], Francois Rué [Bacchus], Mario Ricchiuto [Bacchus].

The software AeroSol is jointly developed in the team Bacchus and the team Cagire. It is a high order finite element library written in C++. The code design has been carried for being able to perform efficient computations, with continuous and discontinuous finite elements methods on hybrid and possibly curvilinear meshes. The distribution of the unknowns is made with the software PaMPA, developed within the team Bacchus and the team Pumas. This year, Dragan Amenga-Mbengoué was recruited on the ANR Realfluids, and François Rué (Service Experimentation et Développement) joined the team Bacchus for working on Aerosol.

At the end of 2011, Aerosol had the following features

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

This year, the following features were added

- **development environement** development of a CDash server for collecting the unitary tests and memory checking. Beginning of the development of an interface for functional tests.
- **General structure** Parts of the code were abstracted in order to allow for parallel development: Linear solvers (template type abstraction for generic linear solver external library), Generic integrator classes (integrating on elements, on faces with handling neighbour elements, or for working on Lagrange points of a given element), models (template abstraction for generic hyperbolic systems), equations of state (template-based abstraction for a generic equation of state).
- **In/Out** Parallel GMSH reader, cell and point centered visualization based on VTK-legacy formats. XML paraview files on unstructured meshes (vtu), and parallel XML based files (pvtu).
- **Quadrature formula** Gauss-Lobatto type quadrature formula.
- **Finite elements** Hierarchichal orthogonal finite element basis on lines, triangles (with Dubiner transform). Finite element basis that are interpolation basis on Gauss-Legendre points for lines, quadrangles, and hexaedra. Lagrange, and Hierarchical orthogonal finite elements basis for hexaedra, prisms and tetrahedra.
- **Geometry** elementary geometrical functions for first order three dimensional shapes: hexaedra, prisms, and tetrahedra.
- **Time iteration** CFL time stepping, optimized CFL time schemes: SSP(2,3) and SSP (3,4)
- **Linear Solvers** Internal solver for diagonal matrices. Link with the external solvers PETSc and MUMPS.
- **Memory handling** parallel degrees of freedom handling for continuous and discontinuous approximations
- **Numerical schemes** Discontinuous Galerkin methods for hyperbolic systems. SUPG and Residual Distribution schemes.
- **Models** Perfect gas Euler system, real gas Euler system, scalar advection, Waves equation in first order formulation, generic interface for defining space-time models from space models.
- **Numerical fluxes** centered fluxes, exact Godunov' flux for linear hyperbolic systems, and Lax-Friedrich flux.
- **Parallel computing** Mesh redistribution, computation of Overlap with PaMPA. collective asynchronous communications (PaMPA based). Tests on the cluster Avakas from MCIA, and on Mésocentre de Marseille, and PlaFRIM.
- **C++/Fortran interface** Tests for binding fortran with C++.

6. New Results

6.1. Low Mach number flows simulations issue

The time-step dependency and the scaling of the pressure-velocity coupling suitable for unsteady calculations of low Mach number flows including acoustic features has been identified in the Momentum Interpolation approach. It has been shown that the proper form of the inertia term in the transporting velocity definition is related to the time-step independency of the steady state. The suitable scaling of the pressure gradient dissipation has been used to suggest a modification of AUSM+-up that allows acoustic simulations of low Mach number flows. The accuracy improvement when the solution is compared to the one of the original AUSM+-up scheme indicates that the scaling identified in the Momentum Interpolation approach can be applied with advantage to Godunov-type schemes [3].

6.2. Experimental results

The MAVERIC test facility has been significantly upgraded with the acquisition of a complete GPU-based system (hardware+software) that speeds up by a factor of 10 the processing of the PIV data. The strong sensitivity of the flow topology to the presence of an acoustic standing wave in the cross-flow has been clearly evidenced. The presently available measurements give already the possibility of extracting numerous velocity profiles for a future fruitful LES assessment. The dedicated 1-jet experiment for DNS assessment will start at the beginning of 2013 [8].

7. Partnerships and Cooperations

7.1. Regional Initiatives

7.1.1. Boundary conditions for DNS (6 months of post-doct funded by Conseil régional d'Aquitaine)

Although DNS is mostly used in simplified geometries, issues remain for properly imposing boundary conditions. Indeed, considering for example an inflow boundary condition (BC), a number of variables depending on the subsonic or supersonic nature of the flow must be suitably imposed. As far as the velocity is concerned, it is highly desirable to prescribe boundary conditions with statistics which will match as much as possible those encountered in practice while controlling the reflective nature of the boundary. This can be highly beneficial to drastically reduce the computational domain, thus reducing the computational time. It has to be checked though that the best identified methodology suitable for the continuous problem is still compatible with the methods of resolution adopted to solve the related discrete problem.The long-term objective is to develop, implement and test an efficient method to prescribe boundary conditions for the DNS simulation of a jet in cross-flow. The focus here will be made on the constraints brought about by the compressible and low Mach nature of the flow. Accordingly, the successful low Mach number compressible laminar flow simulation will be considered as the criterion of success of the post-doc. Project: The activity will begin by properly identifying the different sets of physical inlet/outlet physical boundary conditions that are relevant for the low Mach compressible nature of the flow to be simulated; In that framework, a specific analysis of the popular Navier-Stokes characteristic boundary condition (NSCBC) will be carried out in the context of a low Mach number viscous flow. Second, the compatibility of these NSCBC's with the finite element DG formulation retained in the Aerosol library will be investigated in depth in order to identify any potential incompatibility and the way to overcome it, if necessary. Then, the methodology for combining these BC's with the various flux schemes and methods of solution of Aerosol will be developed. The programming of the proposed methodology in Aerosol will be carried out in a parallel environment. Then, a set of unitary tests will be defined and progressively addressed. Last, the simulation of a laminar low-Mach jet in cross-flow configuration will be carried out.Yann Moguen has been recruited on November 2012 to take up that post-doct position. The Conseil régional d'Aquitaine 6-month funding is supplemented by funding from the European programme IMPACT-AE so that the total duration of the post-doct will be 12 months.

7.1.2. Low Mach number aspects for DG schemes (18 months of thesis funded by Conseil général des Pyrénées Atlantiques)

In the litterature, the targeted direct numerical simulation (DNS) of a jet in a subsonic crossflow at low Mach number has been carried out by solving the zero Mach number Navier Stokes equations i.e. without acoustics. The reader is referred to the work by Muppidi and Mahesh (2007) or by Bagheri et al. (2009). Such an approach is acceptable since in a real combustion chamber, the Mach number is rarely above 0.3 and as long as thermo-acoustic instabilities are not to be dealt with. However, in the present project, it has been decided to adopt a compressible framework in order to be able to study in the future the interaction of a jet with a crossflow where a standing acoustic wave is present which corresponds to the configuration presently studied in the framework of the EU funded KIAI programme Workpackage 3.1). To the best of our knowledge, no DNS of an inclined turbulent JICF with a DG based compressible flow solver has been carried out so far. So a thesis work breakdown on that topic has been established as follows:

- Year 1: Understanding the industrial and contractual context. Asymptotic analysis for small Mach numbers of the continuous problem. Study of the various alternatives for discretization schemes at low Mach number. Establishing the link with schemes adapted for zero Mach number flows. Writing of the corresponding thesis chapter; Writing a communication for an international symposium. Participating in a summer school on numerical simulation.
- Year 2: Implementation of the schemes which exhibit a satisfactory asymptotic behavior at low mach number. Carrying out a DNS of an isothermal single jet in cross flow configuration with and without yaw angle in the framework of the IMPACT-AE programme. Analysis of the results, comparison with existing experimental data available in the team. Writing of the corresponding thesis chapter. Writing and submission of a journal paper.
- Year 3: Improvement of the schemes if necessary. Carrying out the DNS of a cold jet in a hot crossflow configuration with and without yaw angle in the framework of the IMPACT-AE programme. Analysis of the results. Writing of the corresponding thesis chapter. Thesis defense.

Thus a thesis proposal has been established and submitted to the Conseil Général des Pyrénées Atlantiques who agreed to fund 18 months of this thesis. The remaining 18 months will be funded through the European programme IMPACT-AE. The recruitment procedure was launched in June 2012 for a provisional starting date in January 2013.

7.2. National Initiatives

7.2.1. GIS Success

Participants: Vincent Perrier, Pascal Bruel.

We are presently participating in the CNRS GIS (Groupement d'Intérêt Scientifique) which is provisionally called "Super-calcul en Combustion et en Mécanique des Fluides dans les Géométries Complexes" and is led by CORIA. A license agreement has been signed with CORIA to permit the installation of the code Yales 2. This installation has been completed on the LMA cluster by the end of december 2012 and the first test will begin in january 2013 in the framework of our benchmarking activity.

7.2.2. CEMRACS 2012

Participants: Dragan Amenga-Mbengoué [Bacchus], Damien Genet [Bacchus], Emeric Martin [ONERA], Maxime Mogé, Vincent Perrier, Floren Renac [ONERA], Francois Rué [Bacchus], Mario Ricchiuto [Bacchus].

Jointly with the team Bacchus and with ONERA, we participated to the project *Colargol*, which aimed at comparing implementations and performances of high order finite elements methods implemented in our library Aerosol, and in the high order discontinuous Galerkin library AGHORA developed at ONERA. For making fair comparisons with this library, we had to extend our library to three dimensions, and to finish the first parallel version of the code. Our first conclusions is the necessity of stocking all geometrical terms of the finite elements methods (Jacobian, Jacobian matrices, etc...) for having good performances. We are still running the comparison tests on the Mésocentre de Calcul Intensif Aquitain.

7.3. European Initiatives

7.3.1. FP7 Projects

Participants: Vincent Perrier [responsible], Pascal Bruel [substitute].

Program: Propulsion

Project acronym: IMPACT-AE

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

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

Coordinator: Roll Royce Deutschland

Other partners:

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

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

7.4. International Research Visitors

7.4.1. Visits of International Scientists

- Dr. A. Naïmanova, Institute of Mathematics, Almaty, Kazakhstan came for a one-month stay in September 2012.

8. Dissemination

8.1. Scientific Animation

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

- Journal of Computational Physics [VP]
- International Journal for Numerical Methods in Fluids [PB, VP]
- Computers and FLuids [VP]
- SIAM Journal on Applied Mathematics [VP]
- Shock Waves [VP]
- ESAIM: Mathematical Modelling and Numerical Analysis [VP]
- Mathematics and Computers in Simulation [VP]
- Review for the Engineering Computations [VP]
- Combustion and Flame [PB]
- Journal of Aerospace Engineering [PB]
- Computational Thermal Science [PB]

8.2. Teaching - Supervision - Juries

8.2.1. Teaching

- Licence :

  TP Transferts thermiques, 8h, L1, IUT-GTE-UPPA, Pau, France. [PB]

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

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

- Master :

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

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

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

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

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

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

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

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

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

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

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

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

8.2.2. Juries

+ PhD (PB, Referee) :J. Primus, Détermination de l'impédance acoustique de matériaux absorbants en écoulement par méthode inverse et mesures LDV, Université de Toulouse, 6 December 2012. Thesis advisors : F. Simon and E. Piot.

+ PhD (PB, external examiner) :L. Cheng, Combined PIV/PLIF measurements in a high swirl-fuel injector flowfield, Loughborough University, 19 december 2012. Thesis advisor: A. Spencer.

9. Bibliography

Publications of the year

Articles in International Peer-Reviewed Journals

[1] E. FRANQUET, V. PERRIER. *Runge-Kutta discontinuous Galerkin method for interface flows with a maximum preserving limiter*, in "Computers and Fluids", March 2012, vol. 65, p. 2-7 [*DOI :* 10.1016/J.COMPFLUID.2012.02.021], http://hal.inria.fr/hal-00739446.

[2] 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, p. 4096-4141 [*DOI :* 10.1016/J.JCP.2012.02.002], http://hal.inria.fr/hal-00684427.

[3] Y. MOGUEN, E. DICK, J. VIERENDEELS, P. BRUEL. *Pressure-velocity coupling for unsteady low Mach number flow simulations: An improvement of the AUSM+ -up scheme*, in "Journal of Computational and Applied Mathematics", 2012 [*DOI :* 10.1016/J.CAM.2012.10.029], http://hal.inria.fr/hal-00764285.

[4] Y. MOGUEN, T. KOUSKSOU, P. BRUEL, J. VIERENDEELS, E. DICK. *Pressure-velocity coupling allowing acoustic calculation in low Mach number flow*, in "Journal of Computational Physics", 2012, vol. 231, p. 5522-5541, http://hal.inria.fr/hal-00764270.

[5] E. MOTHEAU, T. LEDERLIN, J.-L. FLORENCIANO, P. BRUEL. *LES investigation of the flow through an effusion-cooled aeronautical combustor model*, in "Flow, Turbulence and Combustion", 2012, vol. 88, p. 169-189, http://hal.inria.fr/hal-00764269.

International Conferences with Proceedings

[6] Y. MOGUEN, P. BRUEL, E. DICK. *Bundary conditions for semi-implicit low Mach number flow calculation*, in "ECCOMAS - 6th European Congress on Computational Methods in Applied Sciences and Engineering", Vienna, Austria, September 2012, http://hal.inria.fr/hal-00768523.

[7] V. PERRIER, E. FRANQUET. *A high order conservative method for the simulation of compressible multiphase flows*, in "ECCOMAS - 6th European Congress on Computational Methods in Applied Sciences and Engineering", Vienna, Austria, September 2012, http://hal.inria.fr/hal-00767336.

Conferences without Proceedings

[8] P. BRUEL, J.-L. FLORENCIANO, T. KOUSKSOU, T. LEDERLIN. *A test facility for assessing simulations of jets in cross flow configurations*, in "9th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements", Thessaloniki, Greece, ERCOFTAC, June 2012, http://hal.inria.fr/hal-00768340.

References in notes

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

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

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

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

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

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

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

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

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

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

[19] R. J. GOLDSTEIN, E. ECKERT, W. E. IBELE, S. V. PATANKAR, T. W. SIMON, T. H. KUEHN, P. J. STRYKOWSKI, K. K. TAMMA, A. BAR-COHEN, J. V. R. HEBERLEIN, J. H. DAVIDSON, J. BISCHOF, F. A. KULACKI, U. KORTSHAGEN, S. GARRICK. *Heat transfer - A review of 2000 literature*, in "International Journal of Heat and Mass Transfer", 2002, vol. 45, n^{o} 14, p. 2853-2957 [*DOI :* DOI: 10.1016/S0017-9310(02)00027-3].

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

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

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

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

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

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

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

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

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

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

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

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

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