



Activity Report 2013

Team CAGIRE

Computational Approximation with
discontinuous Galerkin methods and
compaRison with Experiments

RESEARCH CENTER
Bordeaux - Sud-Ouest

THEME
Numerical schemes and simulations

Table of contents

1. Members	1
2. Overall Objectives	1
2.1. Overall Objectives	1
2.2. Highlights of the Year	3
3. Research Program	3
3.1. Computational fluid mechanics: resolving versus modelling small scales of turbulence	3
3.2. Computational fluid mechanics: numerical methods	4
3.3. Experimental aspects	5
4. Application Domains	5
5. Software and Platforms	6
6. New Results	8
6.1. DNS of a jet in crossflow: generation of a synthetic turbulent signal and coupling with characteristics based boundary conditions	8
6.2. Low Mach number flows simulations issues	9
7. Partnerships and Cooperations	9
7.1. Regional Initiatives	9
7.2. National Initiatives	10
7.3. European Initiatives	10
7.4. International Research Visitors	10
7.4.1. Visits of International Scientists	10
7.4.2. Visits to International Teams	10
8. Dissemination	11
8.1. Scientific Animation	11
8.1.1. Review activity	11
8.1.2. Participation in Congress organising committees	11
8.2. Teaching - Supervision - Juries	11
8.2.1. Teaching	11
8.2.2. Supervision	11
8.2.3. Juries	12
8.3. Popularization	12
9. Bibliography	12

Team CAGIRE

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

Creation of the Team: 2011 June 01.

1. Members

Research Scientists

Pascal Bruel [Team leader, CNRS Researcher, HdR]
Rémi Manceau [from Feb 2013, CNRS Researcher, HdR]
Vincent Perrier [Inria Researcher]

Faculty Members

Erwin Franquet [Associate Professor, University of Pau]
Tarik Kousksou [team member until Sep 2013, Associate Professor, University of Pau, HdR]

Engineer

Maxime Mogé [until Nov 2013, Inria]

PhD Student

Simon Delmas [from Jan 2013, University of Pau]

Post-Doctoral Fellow

Yann Moguen [University of Pau]

Administrative Assistant

Josette Baron [Inria]

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, numerical studies of such jets.

From an experimental point of view, the test facility MAVERIC¹ available at LMAP and its metrology are used. An overview of this test rig is presented in Figure 1-bottom. This test facility is able to produce the kind of flow depicted in Figure 1-top. The configuration of an isolated jet in a turbulent crossflow is experimentally investigated to produce high quality data (mainly related to the velocity field properties). One-component laser Doppler velocimetry (LDV) as well as particle image velocimetry (PIV) are the two workhorses used in order to experimentally characterize the flowfield.

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

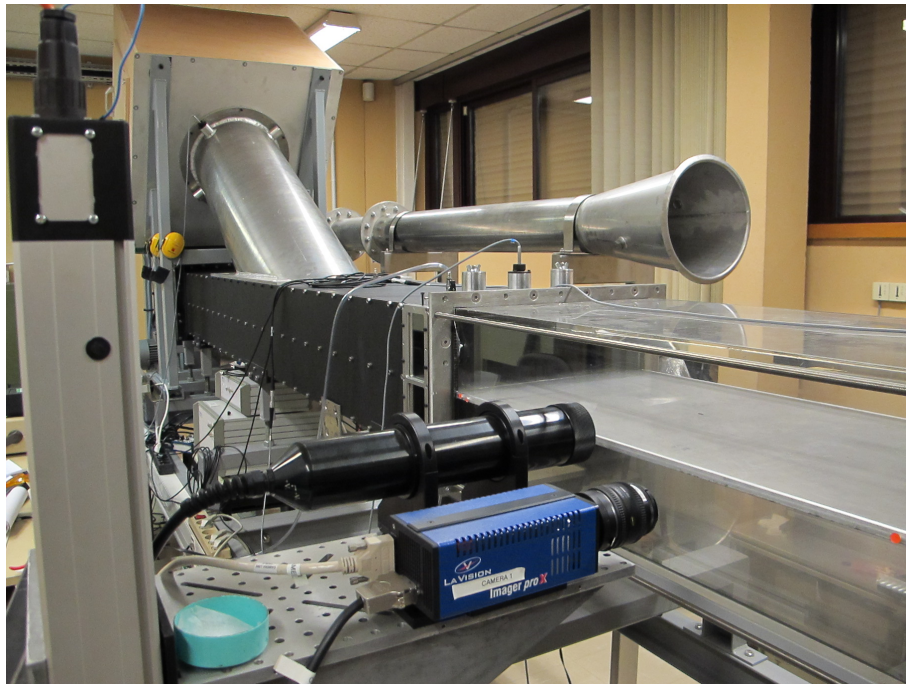
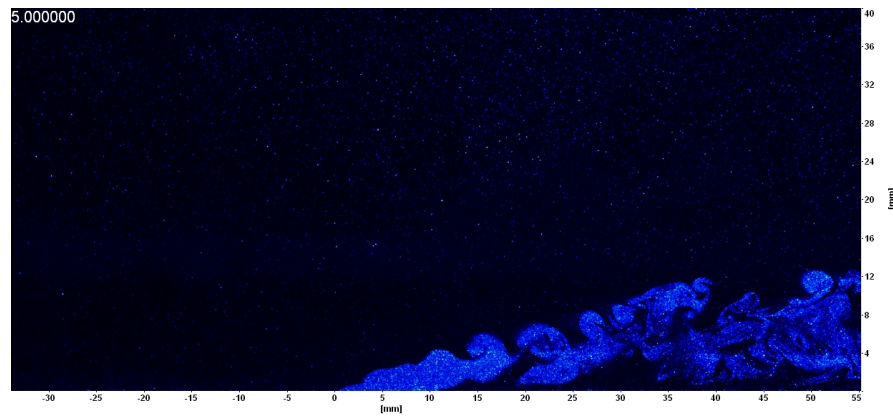


Figure 1. MAVERIC test facility: visualization of a single jet in crossflow (top) and overview of the test rig (bottom).

A close interaction during the course of the project between experiments, simulation and physical modelling represents the backbone of our methodology. From the simulation point of view, one of the short term aims is to perform a direct numerical simulation of an isothermal configuration of an inclined turbulent jet discharging into a turbulent crossflow. This is done in the framework of our participation in the IMPACT-AE EU funded program (<http://www.impact-ae.eu/>). The flows Mach number being small, preserving the accuracy of the specifically developed compressible solver for such flow regime represents quite a challenge. A collaboration has been established with the Bacchus team in order to avoid too many useless redundancies. The Cagire team shares with Bacchus a common framework of development in which both common and team specific tools are being elaborated. From a numerical point of view, the challenge stems from the recourse to hybrid unstructured meshes, which is quite mandatory for our flow configuration, and implicit time integration, which is induced by the low Mach number of the flow. From the point of view of the interaction between experiments and CFD, the challenge is mostly related to the capability of ensuring that the flow simulated and the flow experimentally investigated are as close as possible.

2.2. Highlights of the Year

AeroSol has been successfully tested on the Turing machine of the IDRIS computing center. This was a pre-requisite for the subsequent simulation of the targeted flow configuration.

3. Research Program

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

A typical continuous solution of the Navier Stokes equations is governed by a spectrum of time and space scales. The broadness of that spectrum is directly controlled by the Reynolds number defined as the ratio between the inertial forces and the viscous forces. This number is quite helpful to determine if the flow is turbulent or not. In the former case, it indicates the range of scales of fluctuations that are present in the flow under study. Typically, for instance for the velocity field, the ratio between the largest scale (the integral length scale) to the smallest one (Kolmogorov scale) scales as $Re^{3/4}$ per dimension. In addition, for internal flows, the viscous effects near the solid walls yield a scaling proportional to Re per dimension. The smallest scales may have a certain effect on the largest ones which implies that an accurate framework for the 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 [10]. 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 were introduced by Reed and Hill [32] and first studied by Lesaint and Raviart [25]. 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 [14], [13], one dimensional systems [12], multidimensional scalar conservation laws [11], and multidimensional systems [15]. For the same system, we can also cite the work of [17], [23], 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 [9], 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 [8]. 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 [21]. The symmetric nature of the discretization is particularly well suited with mesh adaptation by means of the adjoint equation resolution [22]. Last, we note that the discontinuous Galerkin method was already successfully tested in [16] at Direct Numerical Simulation scale for very moderate Reynolds, and also by Munz' team in Stuttgart [26], with local time stepping.

For concluding this section, there already exist numerical schemes based on the discontinuous Galerkin method which proved to be efficient for computing compressible viscous flows. Nevertheless, there remain things to be improved, which include: efficient shock capturing term methods for supersonic flows, high order discretization of curved boundaries, or low Mach behaviour of these schemes (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 [27]. For inclined cylindrical holes, the experimental database produced by Gustafsson and Johansson² represents a sound reference base and for normal injection, the work by [34] served as reference for LES simulations [31]. 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 [24]. In 2007, Most [28] developed at UPPA a test facility for studying jet in crossflow issued from shaped holes. The hole shape was chosen as a 12.5 scale of the holes (i.e. at scale 1) drilled by laser in a combustion chamber. His preliminary 2-component PIV results have been used to test RANS simulations [29] and LES [30]. This test facility is extensively used in the framework of the present project to investigate a 1-hole jet i.e. an isolated jet in crossflow. PIV and LDV metrology are 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.

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.

²Slanted jet

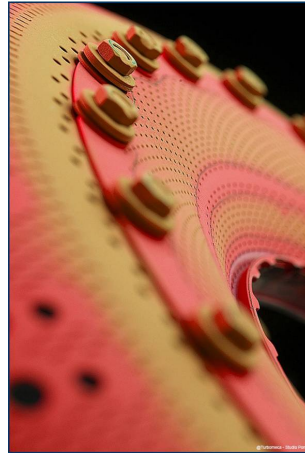


Figure 2. Effusion cooling of aeronautical combustion chambers: close view of a typical perforated chamber wall

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

An important aspect that we began to address in this project is the interaction between the flow and the wall. The aim is to understand the effect of coupling between the heat propagation in the wall and the flow near the wall. A careful study of this interaction can allow to determine the exchange coefficients, and so the efficiency of the cooling by the jet. Such determination may be particularly useful to develop one or multidimensional models of wall-fluid interaction [19]. The large eddy simulation performed by Florenciano [18] clearly put into evidence the strong effect of the presence of an acoustic wave in the crossflow on the dynamics of the heat transfer coefficient at the wall.

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

5. Software and Platforms

5.1. AeroSol

Participants: Dragan Amenga-Mbengoué [Bacchus], Simon Delmas [Cagire], Damien Genet [Bacchus], Maxime Mogé [Cagire], Yann Moguen [Cagire], Francois Pellegrini [Bacchus], Vincent Perrier [Cagire, 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 work of the team Bacchus is focused on continuous finite elements methods, while the team Cagire is focused on discontinuous Galerkin methods. However, everything is done for sharing the largest part of code we can. More precisely, classes concerning IO, finite elements, quadrature, geometry, time iteration, linear solver, models and interface with PaMPA are used by both of the teams. This modularity is achieved by means of template abstraction for keeping good performances.

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

This year, Simon Delmas and Yann Moguen were recruited within the team Cagire. Their respective development, low Mach solver for compressible flows and turbulence injection boundary conditions are performed in the library Aerosol. At the end of 2012, Aerosol had the following features

- **development environment** use of CMake for compilation, CTest for automatic tests and memory checking, lcov and gcov for code coverage reports. Development of a CDash server for collecting the unitary tests and the memory checking. Beginning of the development of an interface for functional tests.
- **In/Out** link with the XML library for handling with parameter files. Reader for GMSH, and writer on the VTK-ASCII legacy format (cell and point centered). Parallel GMSH reader, XML paraview files on unstructured meshes (vtu) and parallel XML based files (pvtu).
- **Quadrature formula** up to 11th order for Lines, Quadrangles, Hexaedra, Pyramids, Prisms, up to 14th order for tetrahedron, up to 21st order for triangles. Gauss-Lobatto type quadrature formula for lines, triangles, quadrangles and hexaedra.
- **Finite elements** up to fourth degree for Lagrange finite elements and hierarchical orthogonal finite element basis (with Dubiner transform on simplices) on lines, triangles, quadrangles, tetrahedra, prisms and hexaedra. Finite element basis that are interpolation basis on Gauss-Legendre points for lines, quadrangles, and hexaedra.
- **Geometry** elementary geometrical functions for first order lines, triangles, quadrangles, prisms, tetrahedra and hexaedra.
- **Time iteration** explicit Runge-Kutta up to fourth order, explicit Strong Stability Preserving schemes up to third order. Optimized CFL time schemes: SSP(2,3) and SSP(3,4). CFL time stepping.
- **Linear Solvers** link with the external linear solver UMFPack, PETSc and MUMPS. Internal solver for diagonal matrices.
- **Memory handling** discontinuous and continuous, sequential and parallel discretizations based on PaMPA for generic meshes.
- **Models** Perfect gas Euler system, real gas Euler system (template based abstraction for a generic equation of state), scalar advection, Waves equation in first order formulation, generic interface for defining space-time models from space models.
- **Numerical schemes** continuous Galerkin method for the Laplace problem (up to fifth order) with non consistent time iteration or with direct matrix inversion. Discontinuous Galerkin methods for hyperbolic systems. SUPG and Residual distribution schemes.
- **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++.

This year, the following features were added

- **development environment** Definition of CMake options for optimization and for using different compilers. Currently, the following compilers have been tested: GNU gcc, Intel icc, and IBM xlc. Aerosol can now be linked with HDF5, PAPI, and can use different BLAS implementations like eigen or MKL.
- **In/Out** Point centered visualization for discontinuous approximations. XML binary output for Paraview was added. The link with HDF5 was added for parallel IO for defining XDMF format. A geometrical pre-partitioning was developed for reducing the size of the parallel graph in the parallel mesh reading.
- **Pyramids** Mesh reader, Lagrange and hierarchical orthogonal finite element basis were added for pyramids. Geometrical functions for linear pyramids were also added.
- **Finite element** Gauss Lagrange finite element basis (order 1 and 2) for triangles.
- **Time iteration** the following implicit integration schemes were added: backward Euler, Crank-Nicolson, and BDF from 2nd to 6th order.
- **Linear Solvers** Interface with PETSc was tested on a parallel environment. An in house block diagonal solver was developed.
- **Memory handling** Aerosol can now work on hybrid meshes.
- **Models** the generic model interface supports now diffusive models. Anisotropic diffusion and (compressible) Navier-Stokes models were added.
- **Instrumentation** Aerosol can give some traces on memory consumption/problems with an interfacing with the PAPI library. Tests have also been performed with VTUNE and TAU.
- **Parallel computing** Tests were performed on the clusters Pyrene (Université de Pau), poincaré (Maison de la Simulation), and on the Tier-1 cluster Turing (IDRIS).
- **Numerical schemes** The DG discretization of advection problems was optimized by stocking most of the geometrical functions and finite elements computations, and by using BLAS implementations for linear computations. Implicit versions of the DG discretization of advection problems. Development of explicit and implicit version of the DG discretization of diffusion problems. Time dependent boundary conditions, periodic boundary conditions, non reflecting boundary conditions. Development of low Mach numerical fluxes, and development of stationary and unstationary tests for this kind of problem.

6. New Results

6.1. DNS of a jet in crossflow: generation of a synthetic turbulent signal and coupling with characteristics based boundary conditions

The implementation of the boundary conditions for DNS of the flow configuration that consists of a jet issuing from an inclined cylindrical hole and discharging into a turbulent crossflow is investigated in the framework of our current participation in the Impact-AE EU funded program. First, a method allowing the generation of turbulent inflow that matches targeted statistics (mean velocity and Reynolds stress tensor components measured on the MAVERIC test facility) has been chosen. On the basis of a study of the main classic methods identified in the literature, it has been considered that the Synthetic Eddy Method (SEM) represents the best compromise between effectiveness and cost, from both a computation and a storage point of view. With this approach, eddy structures are created and injected at the inlet plane of the computational domain. These analytically defined structures are chosen in order to reproduce the most relevant ones present in a turbulent channel flow. The SEM implementation has been considered for (1) a basic form of SEM that does not differentiate the vortices in function of their distance to the wall, and (2) a more elaborated version of the method, denoted SEM-WB, where the inlet plane is split into different zones that accommodate different types

of coherent structures according to what is observed in a turbulent boundary layer. In order to prescribe realistic turbulence statistics, the targeted mean velocity and Reynolds stress values of the SEM-WB method were obtained by performing dedicated PIV measurements on the MAVERIC test facility (UPPA). The basic form of the method gives quite satisfactory results. The values of some parameters of the SEM-WB method have still to be adjusted in order to achieve a better convergence rate towards the targeted statistics. In november 2013, the deliverable D2.211 (Confidential) documenting in details this methodology and the results obtained with the related module written in C++ has been issued by the team to the IMPACT-AE office. Assuming that the synthetic turbulent signal is generated in a satisfactory way, one is left with the set-up of the procedure necessary to incorporate this signal into a characteristics based method for handling the boundary conditions at the flow inlet(s). We have developed an approach that proved suitable, in a 1-D configuration so far, to accurately superimpose acoustics and turbulence while preserving the non reflective properties at the inlet boundary [5].

6.2. Low Mach number flows simulations issues

Our activity for developing schemes suitable for the simulation of low Mach number flows considers the two main techniques developed initially for dealing with either zero Mach number flows (pressure-velocity coupling) or compressible flows (density based approach). For the methodology addressing the pressure-velocity coupling, we concentrated on the issue of handling in a semi-implicit way the unsteady set of characteristics based equations at both the outlet and the inlet of a subsonic internal flow. The methodology employed to solve the boundary equations has been designed to mimic the pressure-velocity coupling employed in the interior of the computational domain. The numerical experiments carried out with an acoustic CFL number significantly larger than unity show that the expected reflective and non-reflective behavior is preserved at these boundaries [3].

For the density based approach [6], the Euler or Navier-Stokes equations semi-discretised with a Roe-like flux scheme are analysed using an asymptotic development in power of the Mach number. As expected, this development shows that the inaccuracy at low Mach is due to the bad scaling of the pressure gradient in the momentum equation [20]. In addition, the behaviour of any compressible solver based on that scheme proved to be highly dependent on the geometry of the mesh elements [33]. Several cures to this inaccuracy problem exist in the literature for steady flow calculations. But for unsteady low Mach flows simulations, our numerical experiments with high order discontinuous Galerkin discretisation put into evidence the bad stability properties of these modified schemes. In order to adress that second issue, a semi-discrete wave equation for the order one pressure in the system has been derived by including the acoustic time scale in the asymptotic development. An analysis of the dissipative terms of this wave equation has been started in order to determine the possible way of regaining good stability properties while ensuring a good accuracy at low Mach.

7. Partnerships and Cooperations

7.1. Regional Initiatives

7.1.1. *An experimental database for DNS assessment (6 months of post-doct funded by Communauté d'Agglomération Pau- Portes des Pyrénées)*

The quality of our unsteady simulations have to be compared with high quality experimental data. Since the targeted baseline 1-jet in crossflow configuration is isothermal, the relevant comparisons will be made mainly on the velocity field for which detailed PIV measurements have to be carried out. In order to assess in depth the quality of our numerical simulations, it is important to generate experimental data that must give access to both the global flowfield statistics (one-point mean values and probability density functions) as well as the velocity field dynamics (spectra) and the most relevant related turbulence scales. In that framework, the objective of this one-year post-doc (co-funded by CNRS and UPPA) is to built-up a stereo-PIV based database giving access simultaneously to the three velocity components in the planes of measurement.

7.2. National Initiatives

7.2.1. GIS Success

We are presently participating in the CNRS GIS Success (Groupement d'Intérêt Scientifique) organised around the two major codes employed by the Safran group, namely AVBP and Yales 2. In the framework of mastering the Yales2 code, one team member has participated in October 2013 in a training session organised by Coria. Then, the yales2 code has been implemented locally and the evaluation of the code has started.

7.3. European Initiatives

7.3.1. FP7 Projects

Participants: Vincent Perrier [responsible of the team contribution], Pascal Bruel [substitute], Simon Delmas [PhD], Yann Moguen [Post-doc].

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 of Science, Technology and Medicine, Loughborough University.

Abstract: The environmental benefits of low emissions lean burn technology in reducing NO_x emissions up to 80% only be effective when these are deployed to a large range of new aero-engine applications. While integrating methodologies for advanced engine architectures and thermodynamic cycles. It will support European engine manufacturers to pick up and keep pace with the US competitors, being already able to exploit their new low emission combustion technology to various engine applications with short turn-around times. Key element of the project will be the development and validation of design methods for low emissions combustors to reduce NO_x 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 NO_x models will increase the accuracy of the numerical prediction. The contribution of our team is to create with AeroSol a direct numerical simulations (DNS) database relevant to the configuration of film cooling for subsequent improvement of RANS based simulations of isothermal and non isothermal wall flows with discrete mass transfer.

7.4. International Research Visitors

7.4.1. Visits of International Scientists

June 2013 (4 days): Prof. E. Dick from Ghent University: improvement of pressure-velocity coupling for low Mach number flow simulation by introducing inertia terms in the flux scheme.

7.4.2. Visits to International Teams

P. Bruel spent a two-week stay at the Institute of Mathematics in Almaty (Kazakhstan) to set-up a joint project around the simulations of combustion of air and coal in a laboratory scale burner. A joint supervision of a Kazakh student was started at this occasion.

8. Dissemination

8.1. Scientific Animation

8.1.1. Review activity

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

- Advances in Mechanical Engineering (RM)
- Combustion and Flame [PB]
- Computational Thermal Science [PB]
- Computer and Fluids [RM][VP]
- Experiments in Fluids [RM]
- Fluid Dynamics Research [RM]
- Flow, Turbulence and Combustion [RM]
- Int J Heat and Fluid Flow [RM]
- International Journal of Computational Methods [YM]
- Journal of Computational and Applied Mathematics [YM]
- Journal of Computational Physics [VP]
- Mathematical Modelling and Numerical Analysis (M2AN) [VP]
- Physics of Fluids [RM]

8.1.2. Participation in Congress organising committees

- Turbulent shear flow phenomena (TSFP-8) held in Poitiers (France) [RM].
- European Workshop on High Order Nonlinear Numerical -Methods for Evolutionary PDE: Theory and Applications (HONOM 2013) held in Bordeaux (France) [VP]
- European Community on Computational Methods in Applied Sciences (ECCOMAS) for Young Investigators Conference (ECCOMAS YIC 2013) held in Bordeaux (France) [VP]

8.2. Teaching - Supervision - Juries

8.2.1. Teaching

- Master : [PB], An introduction to the numerical simulation of reacting flows, 15h, ISAE-Supaéro and University of Toulouse, France.
- Master : [RM], Turbulence Modelling, 40h, École centrale de Lille/ENSI Poitiers/ISAE-ENSMA, Poitiers, France.
- Master : [EF], Simulations industrielles, Fluides compressibles, Combustion industrielle, 100h, ENSGTI, Pau, France.
- Master: [TK], Condensation/Ebullition, 40h, ENSGTI, Pau, France.
- Master: [TK], Exergoéconomie, 20, ENSGTI, Pau, France.
- Master: [TK], Réseaux Fluides, 16h, ENSGTI, Pau, France.

8.2.2. Supervision

- PhD : Simon Delmas, Simulation d'écoulements pariétaux génériques à bas nombre de Mach pour l'amélioration du refroidissement des chambres de combustion : développement et mise en œuvre de schémas de type Galerkin discontinu adaptés, University of Pau, started January 2013, Dir.: [PB] and Co-dir.: [VP].

- PhD : Juan-Luis Florenciano Merino, Étude de la réponse d'un écoulement avec transfert pariétal de masse à un forçage acoustique, University of Pau, defended on July 12, 2013, Dir.: [PB] and Co-dir.: [TK]
- PhD : Jean-François Wald, Modélisation de la turbulence avec traitement adaptatif des parois prenant en compte la thermique active ou passive, started October 2013, Dir.: [RM]
- PhD : Nurtoleu Shakhan, Modelling and simulation of coal combustion, University of Almaty (Kazakhstan), started October 2013, Dir.: Altyn Naïmanova and Co-dir.: [PB]
- PhD : Tran Thanh Tinh, Développement d'une méthode hybride RANS-LES temporelle pour la simulation de sillages d'obstacles cylindriques, University of Poitiers, 28 March 2013, Dir.: [RM]

8.2.3. *Juries*

Several team members participated in the following thesis or HdR juries ("referee" in a French doctoral thesis jury is more or less equivalent to an external opponent in an Anglo-Saxon like PhD jury):

- + PhD : Julien Apeloig, Étude expérimentale de la phase liquide dans les instabilités thermo-acoustiques agissant au sein des turbomachines diphasiques, University of Toulouse, 13 September 2013, [PB, referee]
- + PhD : Guillaume Cottin, Contribution à la modélisation thermique d'une paroi multiperforée, University of Toulouse, 18 October 2013, [PB, referee]
- + PhD : Mario Falese, A study of the effects of bifurcations in swirling flows using large-eddy simulations and mesh adaptation, University of Toulouse, October 7, 2013, [PB]
- + PhD : David Vanpouille, Développement de modèles de turbulence adaptés à la simulation des écoulements de convection naturelle à haut nombre de Rayleigh, University of Toulouse, December 6, 2013 [RM, referee]
- + HdR : Mathieu Fénot, Refroidissement aérothermique, University of Poitiers, 29 November 2013 [PB]

8.3. Popularization

One presentation in Unithé ou Café and presence to the "Inria-Industrie" days [VP]. Participation in the "Visage des Sciences 2013" [PB].

9. Bibliography

Publications of the year

Articles in International Peer-Reviewed Journals

- [1] E. FRANQUET, V. PERRIER. *Runge-Kutta discontinuous Galerkin method for reactive multiphase flows*, in "Computers and Fluids", August 2013, vol. 83, pp. 157-163 [DOI : 10.1016/J.COMPFLUID.2012.07.011], <http://hal.inria.fr/hal-00788235>
- [2] D. MBENGOUE, D. GENET, C. LACHAT, E. MARTIN, M. MOGÉ, V. PERRIER, F. RENAC, M. RICCHIUTO, F. RUE. *Comparison of high order algorithms in Aerosol and Aghora for compressible flows*, in "ESAIM: Proceedings", December 2013, vol. 43, pp. 1-16, <http://hal.inria.fr/hal-00917411>
- [3] Y. MOGUEN, P. BRUEL, E. DICK. *Semi-implicit characteristic-based boundary treatment for acoustics in low Mach number flows*, in "Journal of Computational Physics", 2013, vol. 255, pp. 339-361 [DOI : 10.1016/J.JCP.2013.08.019], <http://hal.inria.fr/hal-00929713>

- [4] Y. MOGUEN, E. DICK, J. VIRENDEELS, 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", 2013, vol. 246, pp. 136-143, This article belongs to a special issue Fifth International Conference on Advanced Computational Methods in ENgineering (ACOMEN 2011) [DOI : 10.1016/J.CAM.2012.10.029], <http://hal.inria.fr/hal-00929726>

National Conferences with Proceedings

- [5] Y. MOGUEN, P. BRUEL, V. PERRIER, E. DICK. *Conditions d'entrée non réfléchissantes pour le calcul des écoulements instationnaires turbulents compressibles à bas nombre de Mach*, in "21ème Congrès Français de Mécanique", Bordeaux, France, AFM, Maison de la Mécanique, 39/41 rue Louis Blanc, 92400 Courbevoie, France(FR), 2013, <http://hal.inria.fr/hal-00930043>

Conferences without Proceedings

- [6] S. DELMAS, V. PERRIER, P. BRUEL. *Behaviour of upwind schemes in low Mach number flow*, in "2nd ECCOMAS Young Investigators Conference (YIC 2013)", Bordeaux, France, September 2013, <http://hal.inria.fr/hal-00855904>

Research Reports

- [7] D. AMENGA MBENGOUE, D. GENET, C. LACHAT, E. MARTIN, M. MOGÉ, V. PERRIER, F. RENAC, F. RUE, M. RICCHIUTO. , *Comparison of algorithm in Aerosol and Aghora for compressible flows*, Inria, January 2013, n° RR-8200, <http://hal.inria.fr/hal-00773531>

References in notes

- [8] F. BASSI, A. CRIVELLINI, S. REBAY, M. SAVINI. *Discontinuous Galerkin solution of the Reynolds-averaged Navier-Stokes and k-omega turbulence model equations*, in "Computers & Fluids", 2005, vol. 34, n° 4-5, pp. 507-540
- [9] F. BASSI, S. REBAY. *A high-order accurate discontinuous finite element method for the numerical solution of the compressible Navier-Stokes equations*, in "J. Comput. Phys.", 1997, vol. 131, n° 2, pp. 267-279, <http://dx.doi.org/10.1006/jcph.1996.5572>
- [10] V. BILLEY, J. PERIAUX, B. STOUFFLET, A. DERVIEUX, L. FEZOU, V. SELMIN. *Recent improvements in Galerkin and upwind Euler solvers and application to 3-D transonic flow in aircraft design*, in "Computer Methods in Applied Mechanics and Engineering", 1989, vol. 75, n° 1-3, pp. 409-414
- [11] B. COCKBURN, S. HOU, C.-W. SHU. *The Runge-Kutta local projection discontinuous Galerkin finite element method for conservation laws. IV. The multidimensional case*, in "Math. Comp.", 1990, vol. 54, n° 190, pp. 545-581, <http://dx.doi.org/10.2307/2008501>
- [12] 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° 1, pp. 90-113
- [13] 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° 186, pp. 411-435, <http://dx.doi.org/10.2307/2008474>

- [14] 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
- [15] 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>
- [16] S. S. COLIS. *Discontinuous Galerkin methods for turbulence simulation*, in "Proceedings of the Summer Program", Center for Turbulence Research, 2002
- [17] 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>
- [18] J.-L. FLORENCIANO. , *Étude de la réponse d'un écoulement avec transfert pariétal de masse à un forçage acoustique*, Pau University, 2013
- [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, pp. 2853-2957 [DOI : DOI: 10.1016/S0017-9310(02)00027-3]
- [20] H. GUILLARD, C. VIOZAT. *On the behaviour of upwind schemes in the low Mach number limit*, in "Computers & Fluids", January 1999, vol. 28, n^o 1, pp. 63–86 [DOI : 10.1016/S0045-7930(98)00017-6], <http://linkinghub.elsevier.com/retrieve/pii/S0045793098000176>
- [21] 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
- [22] R. HARTMANN, P. HOUSTON. *Symmetric interior penalty DG methods for the compressible Navier-Stokes equations. II. Goal-oriented a posteriori error estimation*, in "Int. J. Numer. Anal. Model.", 2006, vol. 3, n^o 2, pp. 141–162
- [23] 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>
- [24] H. LEE, J. PARK, J. LEE. *Flow visualization and film cooling effectiveness measurements around shaped holes with compound angle orientations*, in "Int. J. Heat Mass Transfer", 2002, vol. 45, pp. 145-156
- [25] 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, n^o 33, pp. 89–123
- [26] 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>

- [27] R. MARGASON. *Fifty Years of Jet in Cross Flow Research*, in "NATO AGARD Conference", Winchester, UK, 1993, vol. CP-534, pp. 1.1-1.41
- [28] A. MOST. , *Étude numérique et expérimentale des écoulements pariétaux avec transfert de masse à travers une paroi multi-perforée*, Pau University, 2007
- [29] 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
- [30] E. MOTHEAU, T. LEDERLIN, P. BRUEL. *LES investigation of the flow through an effusion-cooled aeronautical combustor model*, in "8th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements", Marseille, France, June 2010, pp. 872-877
- [31] C. PRIÈRE. , *Simulation aux grandes échelles: application au jet transverse*, INP Toulouse, 2005
- [32] W. REED, T. HILL. , *Triangular mesh methods for the neutron transport equation*, Los Alamos Scientific Laboratory, 1973, n^o LA-UR-73-479
- [33] F. RIEPER, G. BADER. *The influence of cell geometry on the accuracy of upwind schemes in the low mach number regime*, in "Journal of Computational Physics", May 2009, vol. 228, n^o 8, pp. 2918–2933 [DOI : 10.1016/J.JCP.2009.01.002], <http://linkinghub.elsevier.com/retrieve/pii/S0021999109000096>
- [34] S. SMITH, M. MUNGAL. *Mixing, structure and scaling of the jet in crossflow*, in "Journal of Fluid Mechanics", 1998, vol. 357, pp. 83-122