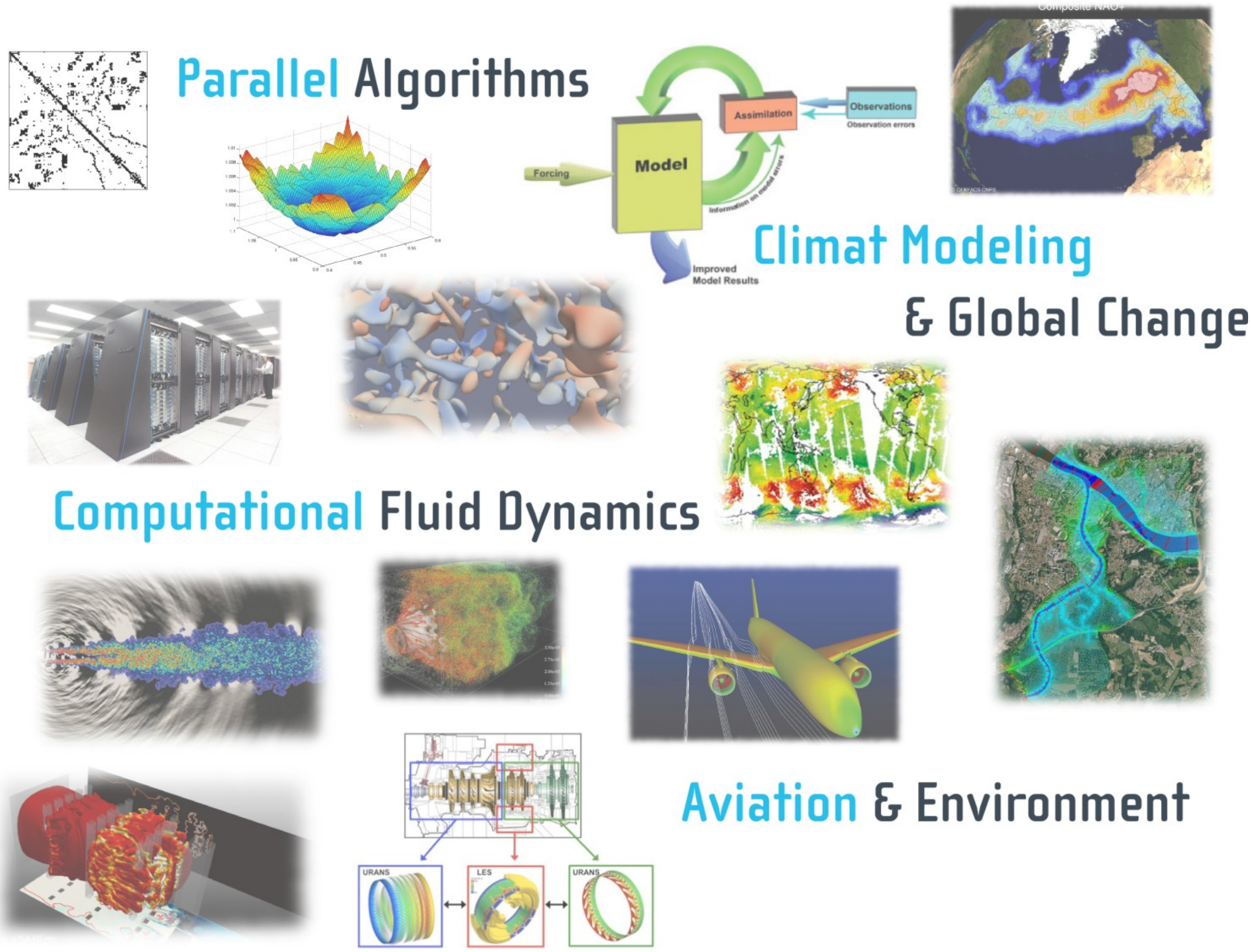
CERFACS

EUROPEAN CENTRE FOR RESEARCH AND ADVANCED TRAINING IN SCIENTIFIC COMPUTING

Journée des Doctorants (JDC)

Mercredi 15 mars 2017

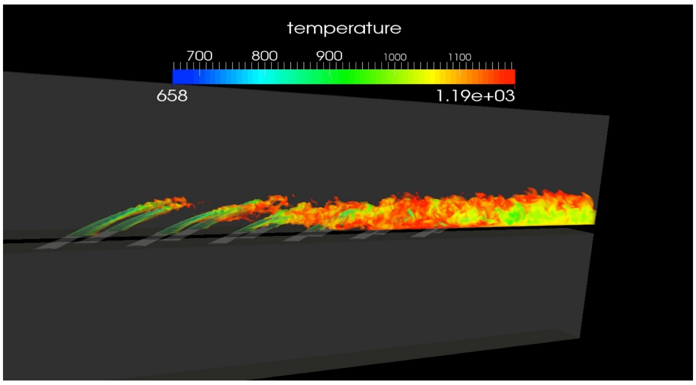
Programme et Recueil des Résumés



9h10 : BIZZARI Romain (CFD) : **Dynamical and thermal modelization of Multi-perforated plates in LES**

In aeronautical gas turbines, the liner of the combustion chamber is submitted to high temperature and risk to meld. To avoid this, Multiperforation are used. This cooling system, commonly uses is based on a simple principle. Fresh air coming from the casing goes through thousands of angled perforations and enters in the combustion chamber. The cooling film that protects the liner from the hot gases results from the coalescence of the micro-jets emanating from the perforations.

When computing the 3-D turbulent reacting flow with the burner, the number of sub-millimetric holes are far too large to allow a complete description of the generation and coalescence of the jets. However, effusion cooling cannot be neglected: it is known to have a drastic effect on the whole flow structure field. Appropriates models are then needed to reproduce the effect of effusion cooling on the main flow. One of the most efficient model is the homogeneous one proposed by S. Mendez and F. Nicoud. Using this model as a root, D. Lahbib and A. Dauptain proposed a more accurate model suitable for currents mesh, the Heterogeneous model.

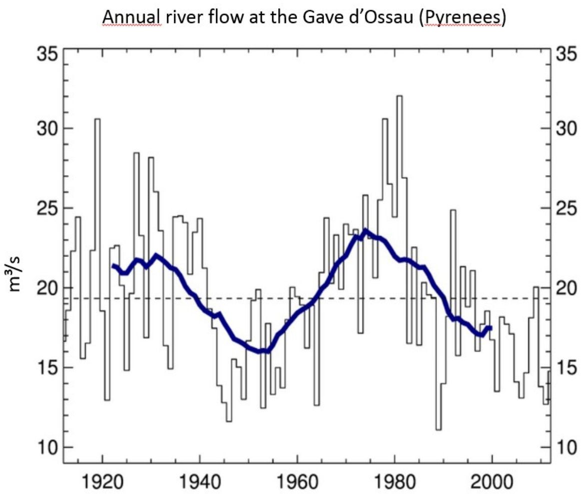


The goal of my thesis is first to modify this model in order to make it more robust and to validate it. The assessment of this work and validations of other methods currently used at CERFACS and by Safran Helicopter Engine is an important part of my thesis.

The second goal of this PhD is to propose an heterogeneous model which link the two sides of the liners, combustion chamber and casing, thank to a pressure term. At the end, this model will be extended in order to take into account the thermal coupling between solid liner and the fluid in both sides. Taking into account the dynamical and the thermal coupling with the casing will allow to care about the pre-heating of the cooling air coming in this part of the engine.

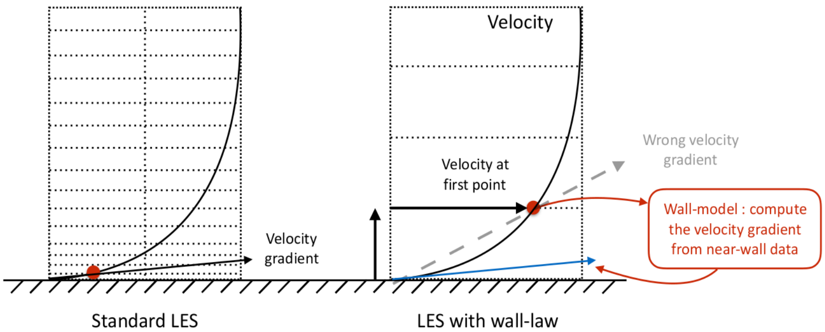
9h18 : BONNET Rémy (GLOBC) : **Evolution of the hydrological cycle in France over the last century**

Recent studies based on the few available long-term observations show a large multidecadal variability in French river flows. Characterizing and understanding this variability is a challenging issue given the limitation of observed data-sets. An approach based on hydrological modelling, with a forced soil-vegetation-atmosphere transfer scheme coupled to a routing scheme is used to reconstruct the French water cycle through the 20th century. Thanks to these reconstructions, we confirm that the multi-decadal variations previously noted in French river flows have mainly a climatic origin. Moreover, we show that multi-decadal variations exist in other hydrological variables (evapotranspiration, snow cover and soil moisture).



9h26 : CATCHIRAYER Mathieu (CFD) : **Wall modeling in CFD : application to a compressor**

Computational Fluid Dynamics is widely used as an engineering tool, but the currently used models for turbomachinery design have several limitations. They compute the time-averaged flow and are therefore unable to predict unsteady phenomena like separation. Large-Eddy Simulation is a way to overcome this limitation and the next step toward achieving more accurate simulations, but due to its high computational cost it is at present restricted to academic cases. This cost is due to the near-wall eddies which are very small and simulating those would require prohibitively fine meshes for industrial configurations. Advanced wall-models will enable us to avoid simulating the near-wall layer and simulate a compressor stage.

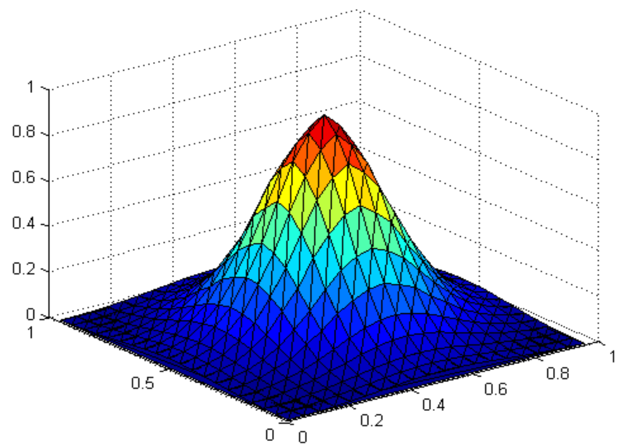


9h34 : PACAUD Frédéric (CFD) : **Understanding and modelisation of the deflagration to detonation transition phenomenon in the oil and gas industry**

Reynolds Averaged Navier-Stokes (RANS) CFD has long been used and developed in the oil and gas industry to assess explosion risk and design facilities against deflagrations. However, recent disasters suggest that detonations - far more destructive - can occur through Deflagration to Detonation Transition (DDT). DDT is by nature a local phenomenon, which still remains poorly understood. Being able to detect or predict DDT in the large scale simulations needed by the industry is a challenging ambition, that needs a serious upheaval in the way the industry models explosion scenarios. What if we tried downgrading accurate physical models instead of upgrading 30-year tweaked correlations ? Bridging the gap between local phenomenon-oriented Direct Numerical Simulation (DNS) and large-scale RANS, Large Eddy Simulation (LES) could prove a flexible method to find this DDT needle in the facility hay stack.

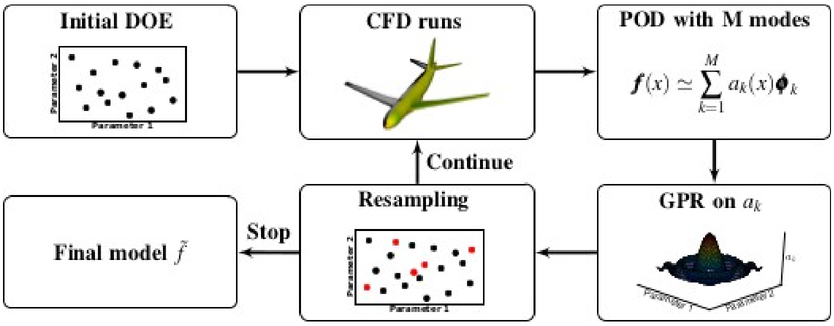
9h42 : GUILLET Oliver (Algo) : **Modelling correlation operators on unstructured grids**

In atmospheric and ocean models, observation errors are taken to be uncorrelated. Yet we know that this hypothesis is incorrect, notably when one considers satellite measurements, the errors of which may exhibit strong spatial, temporal and interchannel correlations. In this study, we adress the issue of accounting for spatially correlated observation errors in a variationnal data assimilation framework. A model that involves solving a partial differential equation is presented. Then, focus is put on dealing with scattered, heterogeneously distributed data.



_9h50 :_ DUPUIS Romain (CFD) : **Physical-Based Surrogate Modeling for Aerodynamics Simulations**

Accurate prediction of steady vector-valued functions, for example pressure or friction field, over a complete aircraft mission requires an important number of evaluations of computationally expensive high-fidelity models. The aim of this presentation is to introduce a process of an adaptive global surrogate model capable of substituting the high-fidelity part. It has been developed for problems with different physical regimes, such as subsonic transonic transition.



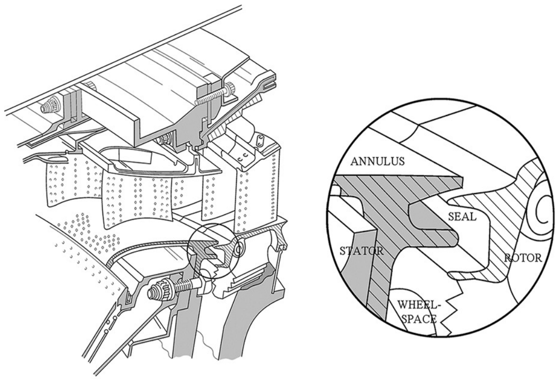
Local subsets of a limited number of simulations are computed with an unsupervised algorithm based on physical criterion and a supervised learning algorithm is used to map these subsets with the input parameter space. A non-intrusive Reduced Order Modelling approach using Proper Orthogonal Decomposition coupled with a data fits method is applied on each subset. Finally, an adaptive process adding information in the region of highly non-linear behavior has been developed. This method is assessed with the one-dimensional Burger equation and a two-dimensional transonic flow around an RAE 2822 transonic airfoil.

_9h58 :_ FIORE Maxime (CFD) : **Impact of the inter-disk cavity flow in aeronautical turbine**

A recent way to improve gas turbine efficiency is the study of secondary air losses.

It corresponds to air bleeded generally at the compressor and used for on-board air system, turbine cooling, sealing or more generally engine sustainability that doesn't produce work since it doesn't travel through the mainstream of gas turbine. In turbine, spaces are required between Nozzle Guide Vanes (NGV) and rotor rows to enable the rotation of the shaft holding rotor rows.

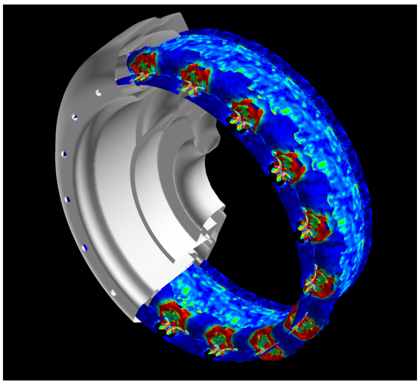
However, hot air coming from the mainstream could enter these spaces also called cavities and impinge rotor rows. Since rotor rows are thermodynamically loaded, they are generally not able to handle such high-temperature flows. That's why part of the secondary air is devoted to prevent air entering these cavities by feeding these cavities with this colder air. The amount of air used is generally higher than what is needed to seal the cavity and some secondary air blows into the mainstream. The aim of this PhD is a better understanding of the interaction between this secondary air and the mainstream flow that lead to additionnal losses for the turbine. This study is leaded through the use of the LES codes AVBP and elsA and the Lattice-Boltzmann solver LaBS.



10h05-Session Posters 1: **BIZZARI / BONNET / CATCHIRAYER / GUILLET / DUPUIS / FIORE**

_11h10 :_ DUPUY Fabien (CFD) : **Reduced models and LES for thermo-acoustic instability in aeronautical gas turbine**

Using lean mixtures has become the standard in aircraft engines in order to tackle NOx production. However, systems operating with lean mixtures are prone to combustion instabilities that can damage the combustor. LES has been proven to be useful to predict instabilities but its computational cost remains too high for the number of configurations and regimes encountered in an engine validation process. This PhD focuses on reduced order models(ROM) to quickly predict unstable frequencies of annular geometries. The 3D acoustic solver AVSP and 1.5D ROM will be used and extended to determine eigenmodes of annular configurations. A continuation method will be applied to track the evolution of modes while going from no flame to the nominal value of a Flame Transfer Function (FTF), and compared to experimental data.



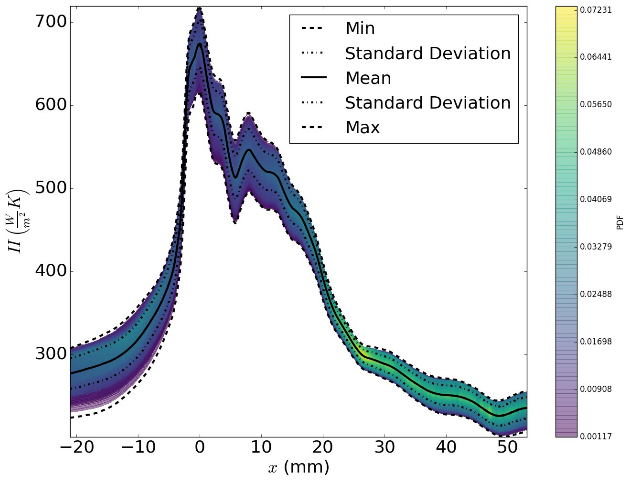
In a next step, a particular attention will be drawn to determining complex impedances at boundaries using surrogate models.

_11h18 :_ LUNET Thibaut (Algo) : **Time parallel strategies for CFD simulation of turbulent flows**

Nowadays, simulation of turbulent flows in Computational Fluid Dynamics (CFD) requires bigger and bigger problem sizes, hence an important increase of computational demand. To face these computation needs, exaFLOPS supercomputers supercomputers able to perform 1e18 floating point operations per seconds (FLOPS) will be built in the next decade. Most of the existing CFD solvers for LES or DNS favorably exploit current existing petaFLOPS architectures by using massively space domain decomposition. However, increase of size and complexity of CFD problems involves longer computational times on next generation computers, despite being more powerful. Time parallelization is an attractive feature that could further improve CFD solvers scalability. More precisely, a successful spatiotemporal decomposition may bring the starting point of their efficiency decay below a few thousands mesh points per computational corewhich is the current state of the art.

For a little more than fifty years, a community has developed an interest on time-parallelization for the solution of unsteady Partial Differential Equations (PDE). Several algorithms have been developed, and sometimes applied with success on PDE problems. However, applications of such algorithms to CFD turbulent flow has not been much investigated.

11h26 : ROY Pamphile (CFD) : **Optimization of surrogate model for Uncertainty Quantification in LES**



Deterministic CFD simulations only provide limited knowledge on a system as uncertainties in the numerical model and its inputs translate into uncertainties in the outputs. Hence the needs of a surrogate model which enables UQ studies at a low CPU cost. The accuracy of an UQ being directly correlated to the surrogate's quality, this work aims at improving its construction by using novel methods such as resampling the space of parameters, using multifidelity surrogate, etc. These methods will be assessed on industrial LES cases.

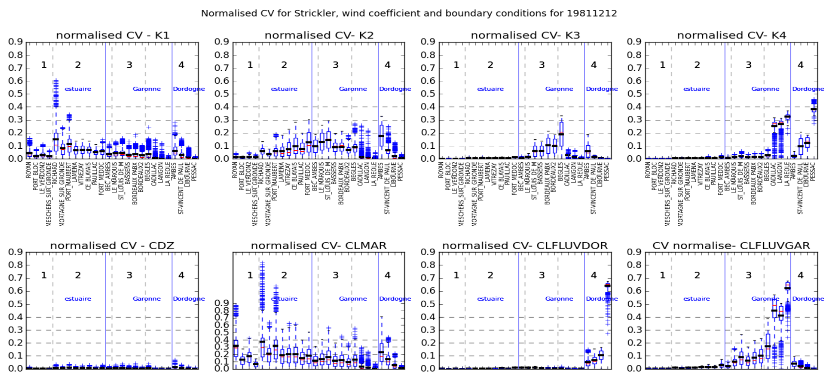
11h34 : GALLEN Lucien (CFD) : **Large-Eddy Simulation to predict soots in aeronautical engines**

Expected more stringent emission legislation for kerosene combustion in aero-engines has driven considerable effort to better understand, model and predict soot formation in gas turbine combustors. To design the next generation of combustion chamber, numerical simulation has become an essential tool, especially to improve their performances in terms of pollutant emisions. This thesis is funded by the European project SOPRANO, which aims at making a breakthrough in the field of soot emission prediction in aeronautical combustors including the soot particle size distribution and their radiative effect on the flame. Contrary to Eulerian approaches largely used to describe soot particles, the present thesis will use a semi-deterministic Lagrangian approach.

11h42 : LABORIE Vanessya (GLOBC) : **Uncertainty quantification in hydrodynamics bidimensional models : the case of Gironde estuary forecasting numerical model**

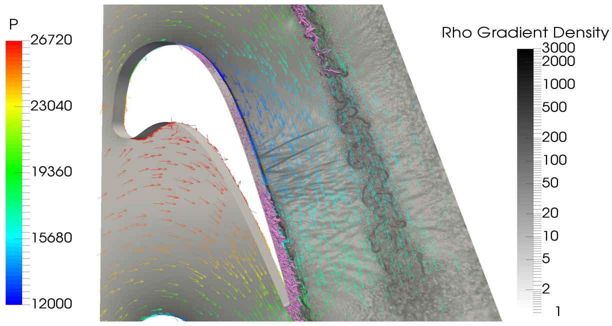
The scope of the PhD thesis is the development and the implementation of data assimilation techniques in Gironde estuary for flood forecasting and the study site is Gironde Estuary.

To meet the scope, a 2D numerical model is being used, based on bidimensional hydrodynamical software TELEMAC2D. This model was firstly calibrated on 10 major storms and then a mesh convergence study was realised to determine the best refinement to use.



Before undertaking a global sensitivity analysis which main challenge is the way to study the unstationnary and spatially distributed forcing fields such as river discharges and maritime water depths boundary conditions or meteorological pressure and wind velocities forcings, a study to propagate and quantify uncertainties due to main parameters and forcings was achieved.

11h50 : HARNIEH Maël (CFD) : **Simulation of a Nozzle Guide Vane (NGV) with fully resolved cooling system**

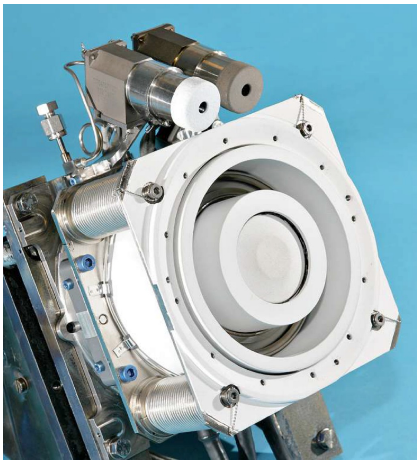


The exit temperature of the combustion chamber is a key point to design efficient aeronautic engines. The Nozzle Guide Vane (NGV) of the turbine is subjected to high thermal stress (temperature can reach up to 2000 K). Cooling air is injected through jets to limit the temperature of the blade. Prediction of the pressure distribution and the cooling efficiency of the blade is critical to improve the design of the NGV.

In this work, Large Eddy Simulations of the academic T120 blade are carried out to assess the effect of the boundary conditions on the flow predictions with the code AVBP. Simulations of an industrial geometry are also performed by imposing a 2D map at the inlet of the domain with realistic data computed from a combustion chamber simulation. Results show a very sensitive reaction of the flow prediction depending on the boundary conditions and numerical handling.

11h58 : JONCQUIERES Valentin (CFD) : **Numerical modelisation and simulation of Hall-effect thruster**

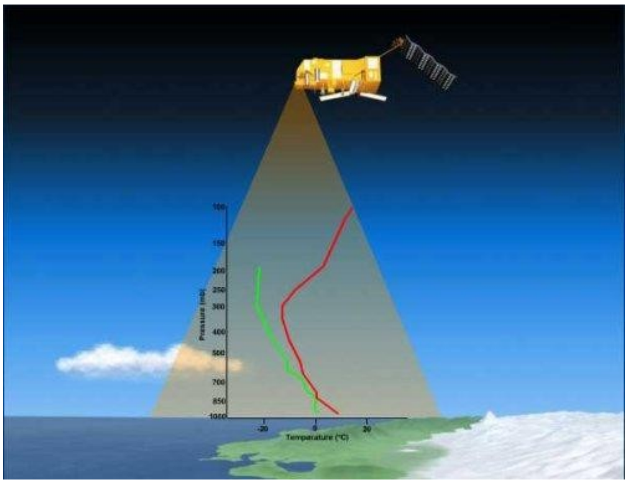
Hall effect thrusters have been used for spatial propulsion since the 1970's. However, complex physical phenomena such erosion or instabilities which may lower thruster efficiency and lifetime, are not yet fully understood. Unfortunately experiments for the design of Hall thrusters are long and expensive, and numerical simulations are now considered to understand and control plasma behavior and predict system efficiency. With the renewed interest for such electric propulsion to supply light satellites, the industrial need for accurate numerical solvers has become crucial.



To answer this demand, the AVIP code was developed which solves plasma physics in real industrial geometries using an unstructured parallel-efficient 3D PIC-fluid methodology. While full 3D PIC simulations of a Hall thruster still require unaffordable CPU time, fluid models can provide in a reasonable computational time 3D results of the plasma behavior inside the discharge channel. One of my thesis objective is to implement the fluid part of the AVIP solver. A particular attention is paid on specific numerical schemes implemented to deal with such equations and its source terms. After the presentation of models and numerics, results obtained with AVIP will be shown. First, solutions of classical benchmarks are compared to PIC reference simulations for validation purposes. The final objective is to perform a 3D sector of a Hall thruster to point out the capability of the solver to handle plasma flows in real geometry.

Poster only : PEIRO Hélène (AE) : **The importance of using dynamical a-priori profiles for infrared O3 retrievals : the case of IASI**

TTropospheric ozone (O3) is a trace gas involved in the global greenhouse effect. To quantify its contribution to global warming, an accurate determination of O3 profiles is necessary. The instrument IASI (Infrared Atmospheric Sounding Interferometer), on board satellite MetOP-A, is the more sensitive sensor to tropospheric O3 with a high spatio-temporal coverage. A possible source of retrievals errors is caused by the a priori profile. Until now, a constant a priori profile was based on a combination of ozonesondes and Aura/MLS (Microwave Limb Sounder) data. This study aims to i) build a dynamical a priori profile O3 with a Chemistry Transport Model MOCAGE (Modèle de Chimie Atmosphérique à Grande Echelle), which has been used with a linear O3 chemistry scheme to assimilate MLS data, ii) integrate and to demonstrate the interest of this a priori profile in IASI retrievals.

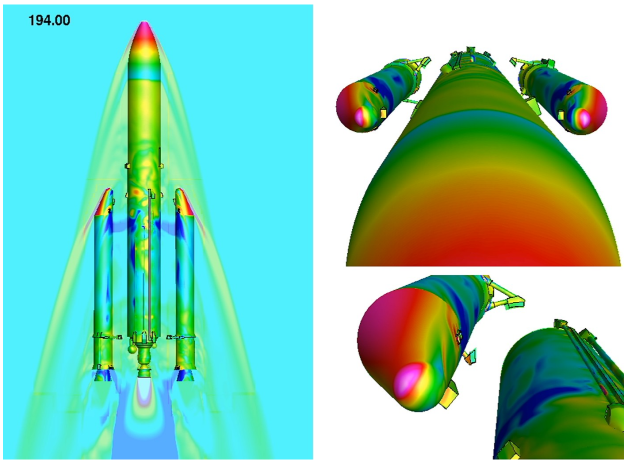


12h15-Session Posters 2 : **LUNET / ROY / LABORIE / HARNIEH / JONCQUIERES / PEIRO**

14h10 : VANDAMME Thibaud (Algo) : **Petrophysical inversion of well logs**

The geological formation evaluation process is based on a workflow combining series of dedicated studies ranging from purely volumetric to dynamic techniques. Classically, the so-called logs, that are the physical measurements taken vertically around a well, are used in a purely static approach. But, the developments presented here prove that some dynamic parameters of the formation can be recovered via the characterization, from the logs, of the mud-filtrate invasion. We will present the modelisation of the physical processes that are involved in the invasion process, when we suppose equilibrium between mud-filtrate and native fluids. The invasion is then solved jointly with the vertical capillary equilibrium to estimate dynamical and petrophysical parameters for each facies in all fluids configurations of the well. Finally, the estimated parameter values are compared with core data results.

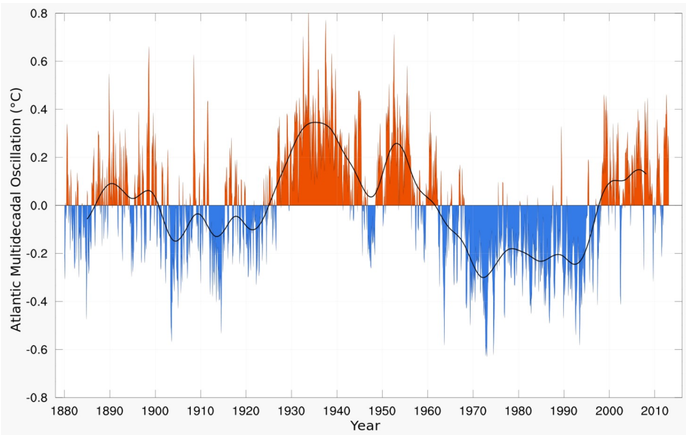
14h18 : MUSCAT Laurent (CFD) : **Zonal implicit/explicit time integrator for compressible unsteady flows in FLUSEPA®**



Airbus Safran Launchers developed for over 25 years the FLUSEPA® code to model all of its aero-propulsive applications. The code handle compressible unsteady flow simulations specific to launcher applications (body in relative movement, high enthalpy reactive flows ...). For this kind of simulations, at present, FLUSEPA® use an adaptive second order explicit time integration algorithm (which is both accurate and efficient). For steady flow simulations, a first order implicit scheme is used (fast convergence but low accuracy). The purpose of this thesis is to couple the two solvers (implicit / explicit) to take advantage of their specificities in terms of local flow characteristics (quasi-steady or very stiff evolution).

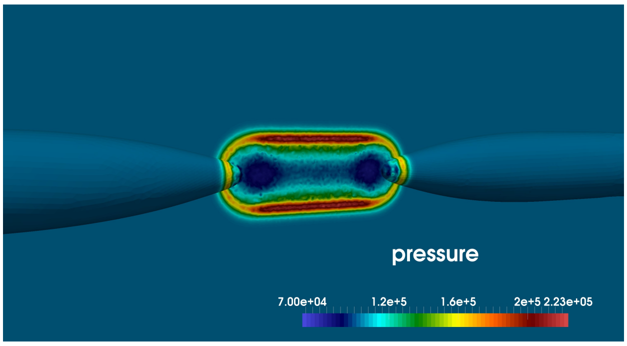
14h26 : QASMI Saïd (GLOBC) : **The climate variability at decadal timescales**

The 2000s are characterized by a slow-down of the global warming while emissions of anthropogenic greenhouse gases still grow. This modulation of this trend is mostly explained by the internal variability of the climate system which can amplify or weaken the climate system response to the anthropogenic (greenhouse gases, aerosols) and natural (volcanism, solar activity) forcings. The internal variability is dominated by two main modes of variability at decadal timescales: the Atlantic Multidecadal Variability (AMV) and the Interdecadal Pacific Variability (IPV). Several observed impacts over continents have been linked to the AMV: summer temperature over Europe, river streamflows in France, Sahelian droughts... The physical processes between the AMV and its impacts over continents are still in discussion. Studying these mechanisms needs the use of global climate models which do not always converge to the same conclusion. Quantifiying their ability to correctly simulate the internal variability of the climate system is crucial in a context of detecting and attributing climate change.



14h34 : COLLIN-BASTIANI Félix (CFD) : **Modelization and Detailed Large-Eddy Simulation (LES) of the ignition process in diphasic environment and real conditions**

Ignition of combustors in the aeronautical industry is a key challenge for clear reasons: in case of engine shut-off, total re-ignition should be guarantee. This relight should thus be well understood. The preferentially used ignition devices for aircraft engines are spark plugs. With this system, a very short and powerful electrical discharge is delivered to the mixture inducing first the creation of a plasma and potentially combustion. However, the transition from the electric discharge to the formation of a flame kernel is not well known today. In this work, an Analytically Reduced Chemistry (ARC) coupling combustion and plasma kinetics is developed to study the very first moments of ignition sequences by electric discharges on a simplified pin-pin ignition system.

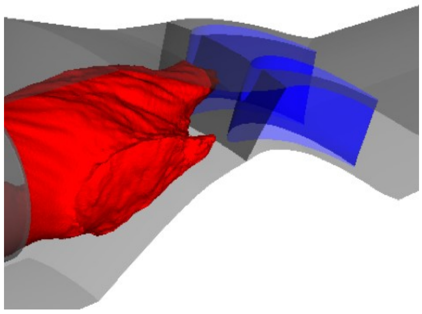


14h42 : LATRE Jean-Baptiste (Algo) : **Alternative Algebraic Structures for Modelling and Computation**

Various fields of application can benefit from using operations in non standard algebras rather than classical frames for computation (linear algebra, real and complex numbers). In fact, these alternative algebraic structures allow to really fit the physical and computational aspects as already demonstrated in robotics, dynamic rotation control, signal and image processing or continuous mechanics. Many other physical problems could take advantage from these alternative algebras such as quantum physic, electromagnetism, waves, ... Largely based on the specific possible properties of multiplication (non commutativity, ring structure with or without division), these structures introduce a path to tackle non linearity. A preliminary exploration of this novel path will be presented.

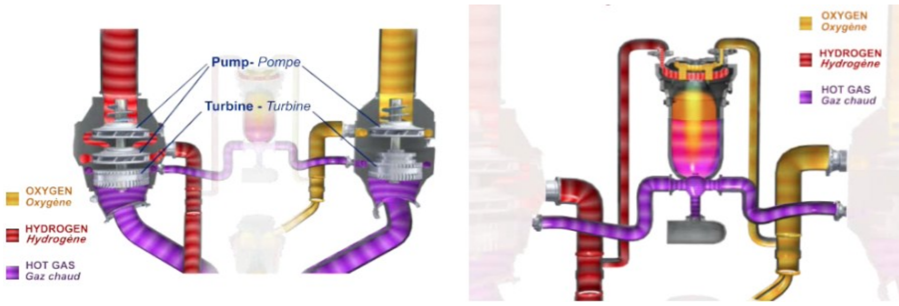
14h50 : THOMAS Martin (CFD) : **Combustion / Turbine interactions : hot spot migration and thermal environment prediction for a better understanding and design of helicopter engines**

Accurate prediction of combustor-turbine interactions using high fidelity LES is of major importance to increase efficiency and durability of next generation lean combustion chambers. As coupled LES of entire combustor-turbine geometries are prohibitively expensive such investigations have rarely been performed during engine design studies. A better understanding of fundamental interaction phenomena between both components allows for an improved combustor-turbine interface design, alleviating design teams from the necessity to apply huge safety margins, while at the same time increasing component durability and efficiency. The cooling system, in this work investigated by homogeneous and heterogeneous coolant injection modeling, has a major impact on the thermal environment in the engine. Thermally critical areas in the context of adiabatic simulations are highlighted using a conjoint analysis of higher order temperature moments and turbulence data.



15h00-Session Posters 3 : **VANDAMME / MUSCAT / QASMI / COLLIN-BASTIANI / LATRE / THOMAS**

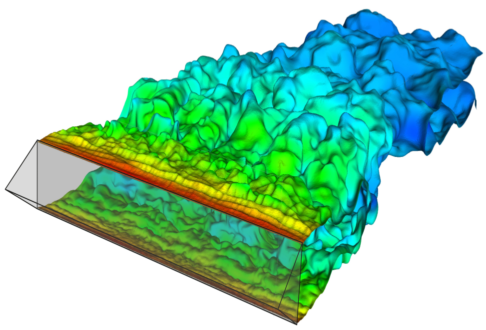
16h12 : QUEGUINEUR Matthieu (CFD) : **Investigation of unsteady phenomena in rotor/stator cavities using LES**



Vibrations are known to have caused problems during the development of space engine turbopumps. Even today, despite the numerous palliative measures taken during the design phases, test programmes often reveal unidentified unsteady pressure phenomena in rotor/stator cavities of turbines. Such features, named 'pressure bands', have proven highly detrimental to the proper operation of the turbopump. The work of T. Bridel has permitted to understand and identify the source of the 'pressure bands' on academic problems and an industrial case. The next step, motivated by this thesis , is the refinement of the industrial case's model. As such, the blades of the rotor will be taken in consideration. Furthermore, in order to validate the studies conducted by T. Bridel, a numerical control of the instabilities sources will be set up.

16h20 : ROCHETTE Bastien (CFD) : **LES modelization of cut-off's processes taking into account diphasic aspects and chamber's thermal conditions**

Facing many challenges such as globalisation, climate change and an increasing scarcity of resources, the aeronautic industry needs to move on to cleaner and more fuel-efficient aircraft engines.

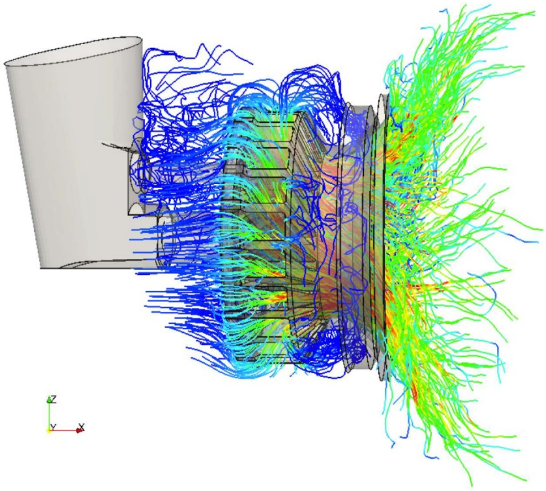


One way to achieve this goal, is to build engines which operate on more severe thermodynamic cycles, while using lean combustion injection technologies. These technological breakthroughs involve a more complex combustion chamber design, which could operate on critical operating regimes. Sometimes, these critical operating regimes lead to an extinction of the combustion chamber during the flight. A better understanding of these extinction phenomena is a key issue for, on the one hand, the safety of passengers during the flight, and on the other hand, the reliability and efficiency of aircraft engines. Measurements on a full-stage engine being difficult to realize, LES has been proven to be useful to predict extinction.

For a better understanding and modelling of these extinction phenomena, it is proposed during this PhD to improve turbulent two-phase flow combustion models.

16h28 : LAURENT Charlélie (CFD) : **Theoretical and numerical study of the combustion instabilities**

Thermoacoustic instabilities are known to occur in a large variety of combustion devices, such as gas turbines or rocket engines, where they can trigger potentially dangerous vibrations that can ultimately damage the combustor structure. Therefore, in order to prevent these oscillations, mechanisms leading to combustion instabilities need to be thoroughly studied, and in recent years LES has proven to be a primordial tool in this matter. However, due to its prohibitive cost, LES is mostly unusable at the design stage or to introduce Uncertainty Quantification. For this reason, it is of great interest to develop alternative, less costly methods such as Reduced Order Models (ROM). The work accomplished during the first months of this PhD aims at developing a ROM based on a time-domain state-space approach, which is expected to be able to handle complex geometries, as well as capturing nonlinear behaviors like limit-cycles, hysteresis phenomenon, or parametric instabilities. The method is also expected to naturally include active control solutions.



16h36 : WISSOCQ Gauthier (CFD) : **Application of the Lattice Boltzmann Method to the rotating cavity flows in turbomachine secondary air systems including high Mach number regions**

In the rotating disk cavities of secondary air systems of aero-engine compressors, natural convection can be induced by the buoyancy force due to inertia and Coriolis effect. This phenomenon results in complex non-axisymmetric unsteady and unstable flows. 2D-axisymmetric and 3D RANS methods have proved their inability to reproduce this behaviour, while correct temperature and heat tranfer predictions are of paramount importance in the design and choice of materials. In this context, the Lattice-Boltzmann Method (LBM) emerges as a promising technique for the simulation of complex unsteady flows. The aim of the present work is to demonstrate the capabilities of the LBM to simulate the flow in a rotating cavity representative of the secondary air system of an aero-engine compressor. The commercial LBM code PowerFLOW is used together with the thermal analysis code PowerTHERM thanks to a Conjugate Heat Transfer Coupling in order to take into account conductive and radiative heat transfer between solid surfaces and the fluid flow.