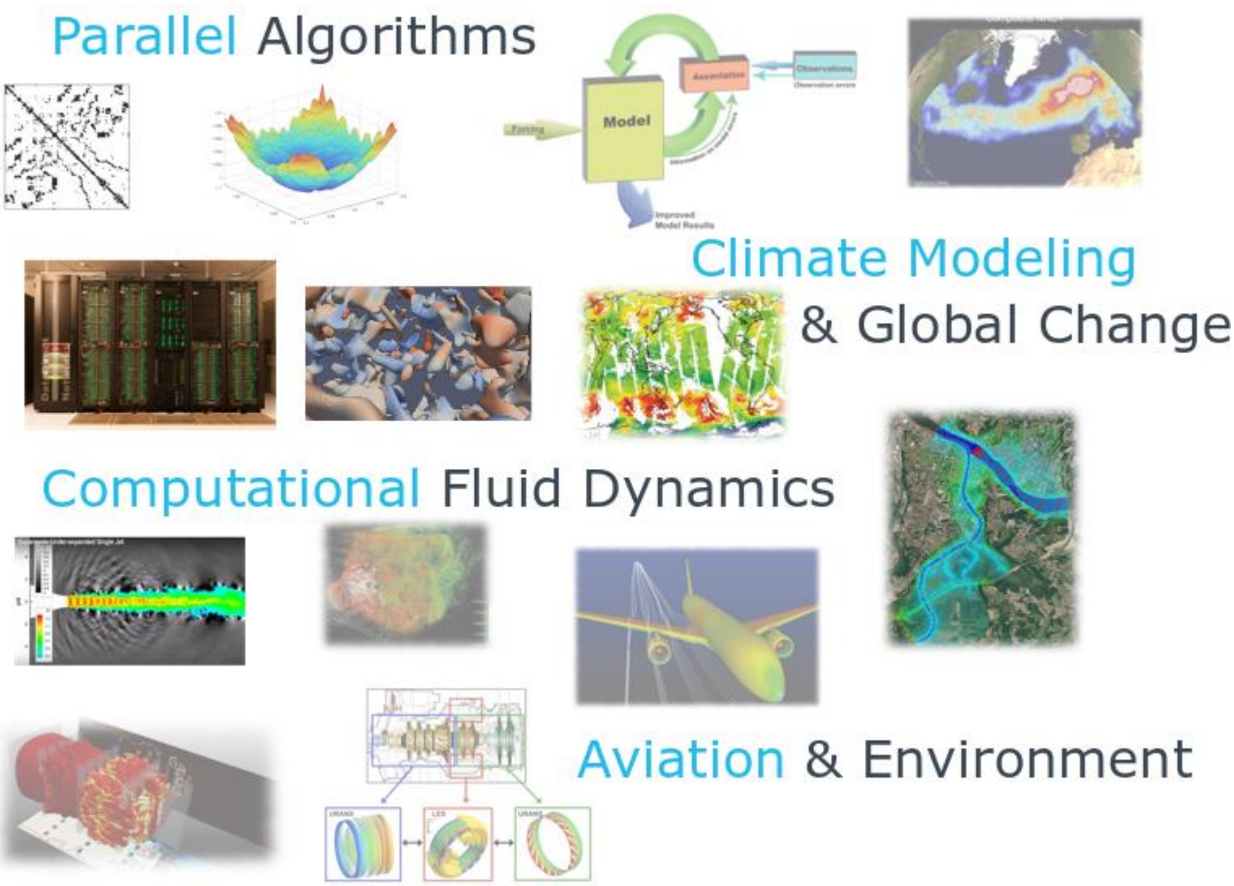
CERFACS

EUROPEAN CENTRE FOR RESEARCH AND ADVANCED TRAINING IN SCIENTIFIC COMPUTING

Ph.D. Students' Day
(JDD 2018)
Thursday 15 march 2018

Program and Abstracts



http://cerfacs.fr

www.cerfacs.fr

March 15, 2018

JCA Room

PhD Students' Day

CERFACS

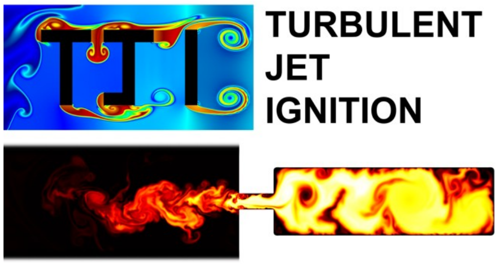
EUROPEAN CENTRE FOR RESEARCH AND ADVANCED TRAINING IN SCIENTIFIC COMPUTING

Time	Name	PhD title	Team
Session 1		**Chair : Thierry POINSOT**	
9:00		**Welcome speech : Catherine LAMBERT**	
9:10	MALÉ Quentin	*Advanced numerical simulations for Turbulent Jet Ignition technology conception in internal combustion engines*	CFD
9:18	CARMONA Julien	*Modelling of two-phase flow phenomena in an aeronautic injector*	CFD
9:26	CAZERES Quentin	*Analysis and reduction of bio-jet fuels chemical kinetics*	CFD
9:34	CHRIPKO Svenya	*Role of Arctic climate change on the mid-latitude atmospheric and oceanic circulation*	GLOBC
9:42	FIORE Maxime	*Influence of cavity purge flow on turbine aerodynamics*	CFD
9:50	DUPUIS Romain	*Surrogate models for aerodynamic and aerothermal applications*	CFD
9:58	DUPUY Fabien	*Reduced order models and LES for thermo-acoustic instabilities in gas turbines*	CFD
10:05		**Poster session n°1 :** MALÉ / CARMONA / FIORE / DUPUY	
11:00		Break	
Session 2		**Chair : MBarek FARES**	
11:10	ESNAULT Soizic	*Large Eddy Simulations of heat transfer in aeronautical honeycomb acoustic liners*	CFD
11:18	COSTES Aurélien	*Bi-directional fire-atmosphere coupling for on-demand simulations in case of forest fires*	GLOBC
11:26	GALLEN Lucien	*Soot predictions in aeronautical combustors using Large Eddy Simulations*	CFD
11:34	LAROCHE Thomas	*Liquid-gas modeling for two-phase flow large eddy simulations*	CFD
11:42	LAURENT Charlélie	*Flame-wall interaction effects on the flame root stabilization mechanisms of a doubly-transcritical LO2/LCH4 cryogenic flame*	CFD
11:50	LELEUX Philippe	*Hybrid direct and iterative solvers for sparse indefinite and overdetermined systems on future exascale architectures*	ALGO
11:58	LO SCHIAVO Ermanno	*ANNULIGhT Marie-Curie project: Thermoacoustic instabilities in annular combustion chambers*	CFD
12:05		**Poster session n°2 :** ESNAULT / GALLEN / LAURENT / LELEUX / MOURADI	
13:10		**Buffet**	
Session 3		**Chair : Bénédicte CUENOT**	
14:10	PACAUD Frédéric	*Understanding and modeling the deflagration-to-detonation transition phenomenon in the oil and gas industry*	CFD
14:18	NADAKKAL Sreeijith	*Simulation and optimization of the process Vapocraguage in the refinery furnaces*	CFD
14:26	VEILLEUX Adèle	*A multi-element-shape extension for the spectral difference method*	CFD
14:34	ROUSSEAU Victor	*Air-sea coupling over western boundary oceanic currents: benefits of high-resolution models in the representation of physical mechanisms and in present and future climate impacts*	GLOBC
14:42	QUEGUINEUR Matthieu	*Investigation of unsteady phenomena in rotor/stator cavities on a spatial turbopump turbine stage*	CFD
14:50	ROY Pamphile	*Uncertainty Quantification in a Costly Numerical Environment*	CFD
14:58	WISSOCQ Gauthier	*Application of the Lattice Boltzmann Method to the rotating cavity flows of turbomachine secondary air systems*	CFD
15:05		**Poster session n°3 :** PACAUD / VEILLEUX / QUEGUINEUR / ROY / WISSOCQ	
16:00		Break	
Session 4		**Chair : Olivier THUAL**	
16:10	RENARD Florian	*Extension of the lattice Boltzmann method to compressible aeronautical flows*	CFD
16:18	BLANCHARD Simon	*Multi-physics Large-Eddy Simulation of cryogenic combustion in rocket engines*	CFD
16:26	SCHUSTER Dominik	*A new approach for numerical flow simulations in the domain of biomedicine, petroleum and aeronautics*	CFD
16:34	SHASTRY Varun	*Numerical study of thermo-acoustic instabilities in spray flames*	CFD
16:42	TIBERI-WADIER Anne-Laure	*Ensemble flood forecasting with an integrated rainfall-hydrology-hydraulic approach*	GLOBC
16:50	POUECH Paul	*Large Eddy Simulation of isochoric combustion chamber and turbine interactions*	CFD
16:58	VILLAFANA Willca	*Modeling of physical phenomena in a Hall-effect thruster and assessment of their impacts on performances and wall erosion*	CFD
17:05		**Best poster award**	
17:15		Acknowledgments	

SESSION 1

Quentin MALÉ (Renault/CERFACS, CFD)

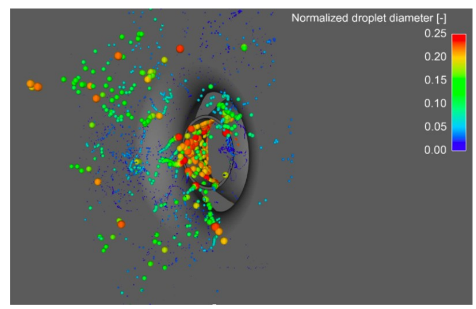
Advanced numerical simulations for Turbulent Jet Ignition technology conception in internal combustion engines



Engine designers face rigorous constraints regarding consumption, pollutant and noise emissions due to environmental issues. Consequently, engine designers are looking for innovations in order to improve efficiency and pollutant emissions. This study focuses on a technological breakthrough where a main homogeneous lean mixture is ignited with turbulent jets produced by an upstream prechamber. The turbulent jets ignition technology allows to burn lean mixture with high efficiency and reliability while lean burning improves energy efficiency and furthermore reduces pollutant emissions. However, there are scientific and technical obstacles with this technology that are yet unresolved, motivating the present study. CFD is nowdays a useful tool in order to investigate and design aerothermochemical processes. Advanced CFD simulations are therefore used to investigate flame propagation and combustion processes, turbulent jet penetration, flame/turbulence and flame/wall interaction as well as intake and exhaust prechamber processes.

Julien CARMONA (CERFACS/Safran Aircraft Engines, CFD)

Modelling of two-phase flow phenomena in an aeronautic injector



To fulfill future economic objectives and environmental regulations, the propulsion aeronautic sector must reduce engine pollutant emissions and fuel consumption. The new generation of engines is based on lean combustion which allows to decrease the maximum temperature in the combustion chamber. Such engines use airblast injectors which guarantee a good flame stability and a high combustion efficiency. These injectors are complex systems where multiple and coupled physical phenomena occur in the gas and liquid phases. On top of turbulence and boundary layer flows, spray/wall interactions, liquid film flow and atomization play a key role on the injection process, that must be characterized in detail to allow a good control of the spray. In the present PhD thesis, Direct Numerical Simulation of film flows and of atomization (with the code YALES2ae) will be used in combination with experiments from the literature to improve the injection models implemented in the AVBP Lagrangian solver. The influence of pressure, liquid fuel composition and film flow on the spray formation and the subsequent two-phase flame will be investigated. As a final objective, the new developed models will be applied to the simulation of an industrial configuration of Safran Aircraft Engines. (figure credits : Chaussonnet 2014)

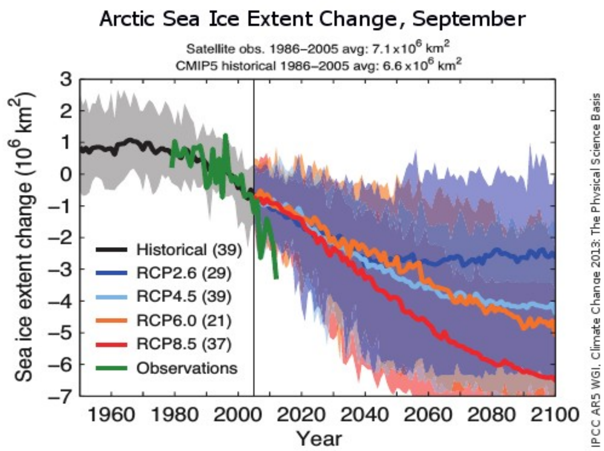
Quentin CAZERES (CERFACS, CFD)

Analysis and reduction of bio-jet fuels chemical kinetics

The imminent shortage of petroleum derived fuels incites the aeronautics community to invest in renewable alternative jet fuels. Large Eddy Simulation (LES) is a key tool to understand combustion in aircraft combustors however current technologies do not allow computations with detailed chemistries, especially not for biofuels with reaction mechanisms containing thousands of species and ten times more reactions. The objective of this PhD, funded by the JETSCREEN European project, is to use analytical techniques to reduce the chemical kinetics of a bio-jet fuel surrogate to a computationally affordable Analytically Reduced Chemistry (ARC) with less than 40 species and 100 reactions. The reduction will be performed with the YARC2 and CANTERA software. Then, the ARC will be used in the AVBP code in order to assess the characteristics of bio-jet fuel combustion in a real engine.

Svenya CHRIPKO (CERFACS, GLOBC)

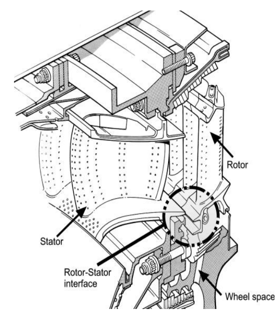
Role of Arctic climate change on the mid-latitude atmospheric and oceanic circulation



The Arctic region has experienced the largest temperature increases over the past several decades accompanied by rapid sea ice melting and thinning. Arctic sea ice extent has declined by more than 10% per decade during summer and climate projections indicate a high probability of having ice-free summers by the middle to end of the century. This is partly due to the polar amplification phenomenon in which global warming is amplified at high latitudes, through physical mechanisms that remain to be fully explained. Climate model studies have shown that changes in Arctic sea ice cover can have substantial impacts on weather and climate both locally and at mid-latitudes. For instance, Arctic sea ice decrease could affect large-scale atmospheric variability in the Northern hemisphere through change of the North Atlantic Oscillation; the dominant mode of atmospheric sea level pressure variability in the North Atlantic, which drives winter climate in Europe, North America and parts of Asia. However, partly because of the diversity in the models' response both in the atmosphere and the ocean, the mechanisms through which Arctic sea ice melt could affect climate remain unclear. The goal of this PhD work is to investigate, through climate model experiments, the physical processes at play in the mid-latitude climate response to projected Arctic sea ice decline.

Maxime FIORE (CERFACS, CFD)

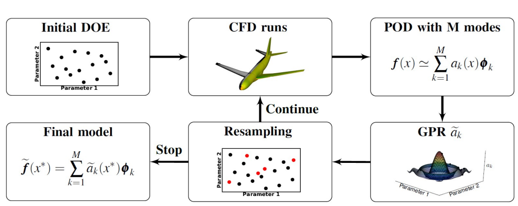
Influence of cavity purge flow on turbine aerodynamics



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 additional losses for the turbine. This study is leaded through the use of the LES codes AVBP and elsA and the Lattice-Boltzmann solver LaBS.

Romain DUPUIS (IRT Saint-Exupéry/CERFACS, CFD)

Surrogate models for aerodynamic and aerothermal applications



Aerodynamic simulations are intensively used in the industry to design complex systems. However, the computation cost of such simulations can be important and several parameters of the system can vary extensively, requiring thousands of simulations and becoming intractable. The aim of this PhD. thesis is to develop non-intrusive reduced order models capable of substituting the simulations and keeping a sufficient accuracy. This replacement is achieved at the expense of model precision. Indeed, aerodynamic simulations involve hyperbolic equations and discontinuous features of solutions. Classical methods using global Proper Orthogonal Decomposition and interpolation encounter issues to predict aerodynamic fields. Therefore, a specific method has been developed in this thesis, consisting in building a local reduced order model for each physical regime. The main challenges are the identification of such subdomains using Machine Learning and the intelligent development of the database used to build the reduced order model.

Fabien DUPUY (CERFACS, CFD)

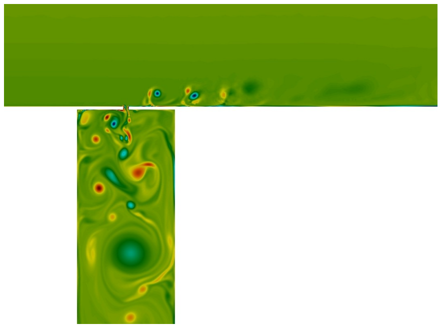
Reduced order models and LES for thermo-acoustic instabilities in gas turbines

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. This PhD focuses on reduced order models (ROM) to quickly predict unstable frequencies of annular geometries. Flame Transfer Functions (FTF) are extracted from LES in an effort to parametrize the flame response to acoustic fluctuations as a function of a small set of dimensionless numbers. The modeled FTF will then be introduced back in ROM codes and obtained frequencies will be compared to experimental data of the NoiseDyn burner. A continuation method will be applied to track the evolution of modes while going from no flame to the nominal value of the FTF.

SESSION 2

Soizic ESNAULT (CERFACS, CFD)

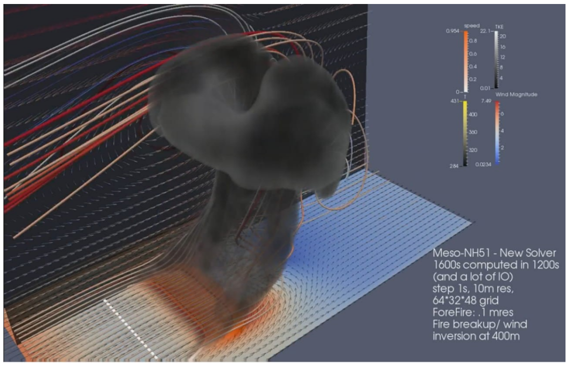
Large Eddy Simulations of heat transfer in aeronautical honeycomb acoustic liners



The OPTIMA project, funded by DGA for Airbus, focuses on two major issues for propulsion systems: thermal protection and thermal optimization. This thesis is aiming at setting high fidelity numerical modelling for acoustic treatment devices combined with heat exchangers, that are studied to enhance the available energy use within engines nacelles. Using Large Eddy Simulations with the AVBP code, we are first targeting at increasing knowledge about flows near and within acoustic treatments for nacelles walls. We want then to understand the influence of heat exchangers on acoustic treatments performance, while understanding and controlling the effect of acoustics on heat transfer. Following this, the performance regarding different configurations will be compared. This will enable to identify the key factors that affect the devices performance, and propose a methodology to evaluate an industrial set-up.

Aurélien COSTES (CNRS, GLOBC)

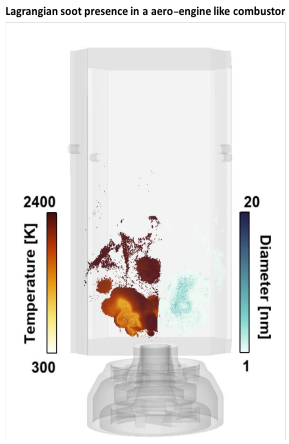
Bi-directional fire-atmosphere coupling for on-demand simulations in case of forest fires



Wildfires are part of devastating natural phenomenons and lead, each year, to significant human, material and environmental damages. The purpose of the FireCaster project, in which my PhD falls within, is , on one hand, to develop forecasting tools to predict the fire spread after its ignition and, on the other hand, to constitute risk maps from ensemble forecasts systems. In order to build the most reliable model possible, it is important to correctly adjust the system governing equations. Within this framework, the atmospheric model Meso-NH is operated in its compressible version which implies the use of time-splitting methods to provide a time step comparable as the one used in the anelastic version of Meso-NH, despite the substantial CFL constraint. Moreover, the implementation of non-reflecting upper boundary conditions would allow the evacuation of acoustic waves, uninteresting for the weather forecasting field. This section is currently investigated and results will be soon presented. Furthermore, the use of data assimilation in the coupling between the atmospheric model, Meso-NH, and the fire spread model, ForeFire, would provide a higher precision in the forecasts.

Lucien GALLEN (CERFACS, CFD)

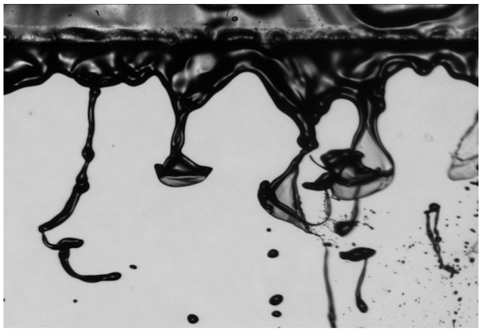
Soot predictions in aeronautical combustors using Large Eddy Simulations



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.

Thomas LAROCHE (CERFACS, CFD)

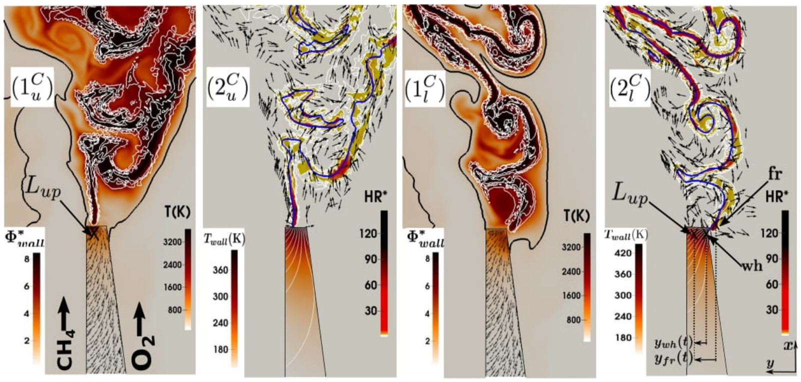
Liquid-gas modeling for two-phase flow large eddy simulations



Nowadays, the aeronautic field – as well as other ways of transportation – is submitted to many restrictions and regulations linked to the growing awareness of pollution and its impact on climate change. This includes the emission of soot which results from many complex physical and chemical processes. Among them, the injection and atomization of liquid fuel, possibly forming liquid films on the walls, are known to play a key role in the formation of soot. In the context of Direct Numerical Simulation, the Level-Set and VoF (Volume of Fluid) methods have proven their efficiency and accuracy in describing the liquid-gas interface. However, in the context of LES and real configurations, modelling liquid atomization still represents a challenge and is often replaced by phenomenological models. In the present work, multi-fluid approaches will be investigated for an approximate description of atomization. Particular effort will be put on the evaluation of the gain in accuracy of such methods in comparison to their computing cost.

Charlélie LAURENT (CERFACS, CFD)

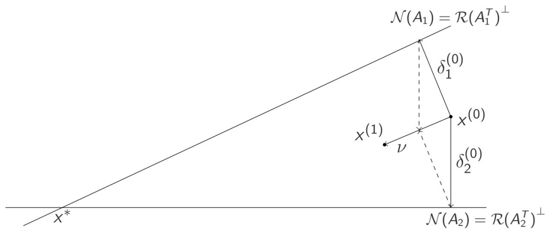
Flame-wall interaction effects on the flame root stabilization mechanisms of a doubly-transcritical LO2/LCH4 cryogenic flame



High-fidelity numerical simulations are used to study flame root stabilization mechanisms of cryogenic flames, where both reactants (O2 and CH4) are injected in transcritical conditions in the laboratory scale test rig Mascotte operated by ONERA (France). Simulations provide a detailed insight into flame root stabilization mechanisms for these diffusion flames: they show that the large wall heat losses at the lips of the coaxial injector are of primary importance, and require to solve for the fully coupled conjugate heat transfer problem. In order to account for flame-wall interaction (FWI) at the injector lip, detailed chemistry effects are also prevalent and a detailed kinetic mechanism for CH4 oxycombustion at high pressure is derived and validated. This kinetic scheme is used in a real-gas fluid solver, coupled with a solid thermal solver in the splitter plate to calculate the unsteady temperature field in the lip. A simulation with adiabatic boundary conditions, an hypothesis that is often used in real-gas combustion, is also performed for comparison. It is found that adiabatic walls simulations lead to enhanced cryogenic reactants vaporization and mixing, and to a quasisteady flame, which anchors within the oxidizer stream. On the other hand, FWI simulations produce self-sustained oscillations of both lip temperature and flame root location at similar frequencies: the flame root moves from the CH4 to the O2 streams at approximately 450 Hz, affecting the whole flame structure.

Philippe LELEUX (CERFACS, ALGO)

Hybrid direct and iterative solvers for sparse indefinite and overdetermined systems on future exascale architectures



Many large-scale numerical computations involve the solution of large, sparse linear systems of the form Ax=b, coming from the discretization of continuous Partial Differential Equations. This was the case in the context of the Energy oriented Center of Excellence (EoCoE) which involved simulation codes arising from several applications from sustainable energy fields. Our approach to these problems has been developed at CERFACS in the ABCD solver. This solver follows the well-known block-Cimmino idea of row projection for solving linear systems. This is an hybrid direct-iterative method designed for sparse, square, unsymmetric systems of equations. A new pseudo-direct version of this approach relies on an augmentation of the matrix which leads to the direct solving of a new condensed system.

Ermanno LO SCHIAVO (CERFACS, CFD)

ANNULIGhT Marie-Curie project: Thermoacoustic instabilities in annular combustion chambers

The objective of the PhD is to use high-performance Large Eddy Simulation (LES) on massively parallel computers to analyse and understand how such instabilities can appear in annular combustion chambers: such chambers are found in multiple systems such as gas turbines used for power production or for aerospace engines. The physics of these instabilities remains sufficiently mysterious to be very difficult to predict at the design stage so that they are often discovered at late stages during the project (usually when the engine is fired for the first times), creating significant risks for many industrial programs. Instabilities in combustion chambers are often due to a coupling phenomenon between acoustics waves and unsteady combustion. The numerical simulation of this highly unsteady phenomena in realistic configurations is one of the main goal of the thesis an the influence of boundary conditions will be a critical part. Designing chambers which are ’stable by design’ will be one of the main targets.

SESSION 3

Frédéric PACAUD (CERFACS/Total, CFD)

Understanding and modeling the deflagration-to-detonation transition phenomenon in the oil and gas industry

The oil and gas industry have considered deflagration the only explosion regime likely to occur in accidental vapor cloud explosions. However, recent disasters suggest that detonations - far more destructive - may occur through the process called Deflagration-to-Detonation Transition (DDT). The problems arising are serious: theoretically, DDT remains poorly understood despite intense research efforts; technically, protection against detonations would be unaffordable; and numerically, DDT is a small-scale local phenomenon, keeping us from simulating it easily at large (industrial) scales. Bridging the gap between local phenomenon-oriented Direct Numerical Simulation and large-scale 30-year adjusted URANS simulation, Large Eddy Simulation could prove a flexible method to find the needle in the hay stack and help prevent DDT in accidental gas explosions.

Sreejith NADAKKAL (CERFACS, CFD)

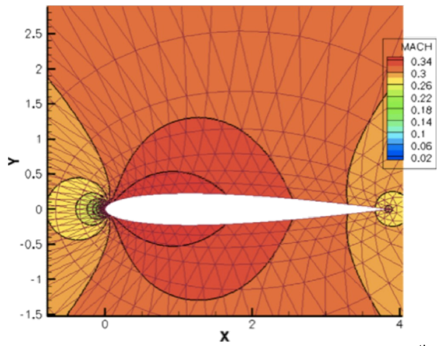
Simulation and optimization of the process Vapocraguage in the refinery furnaces

This PhD is part of the IMPROOF European project, aiming at drastically improving the energy efficiency of steam cracking furnaces in a cost-effective way, while simultaneously reducing emissions of greenhouse gases and NOX. The performance of industrial reactors strongly depends on the flame to which it is exposed, and on the reactor geometry which decides the heat transfer through conduction and radiation. Large Eddy Simulation (LES) of the furnace-reactor configuration will be performed using efficient numerical tools developed in CERFACS in a multiscale-multiphysics framework to include effects of conduction, radiation and analytically reduced chemistry.

Adèle VEILLEUX (CERFACS/ONERA, CFD)

A multi-element-shape extension for the spectral difference method

Large Eddy Simulation is able to capture turbulence large scales but requires refined meshes, high-order discretization method and HPC capability. Among available methods, the Spectral Difference approach is a recent high order way to discretize LES equations on unstructured grids which seems to be promising. The SD method is well defined for segments, quadrangles and hexahedra. The aim of the PhD is to define a stable and robust discretization for other standard elements (tetrahedron, prisms, pyramids) and to validate it on aerodynamic and combustion applications. This extension will bring mesh flexibility within a reactive LES solver in order to handle more complex geometries.

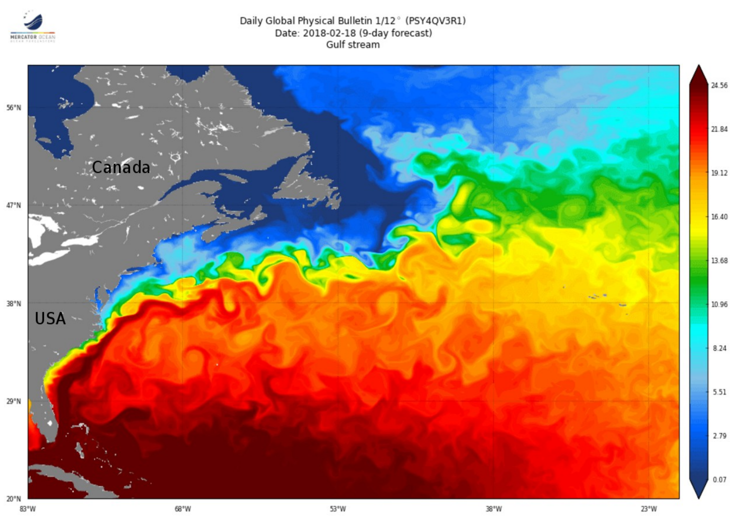


Mach number contours generated using 4th order SD scheme for $M^{\infty} = 0.3$ and $\alpha = 0°$ [1]

[1] A Balan, G May, and J Schöberl. A stable high-order Spectral Difference method for hyperbolic conservation laws on triangular elements. *Journal of Computational Physics*, 231(5) :2359–2375, 2012

Victor ROUSSEAU (Université Paul Sabatier, GLOBC)

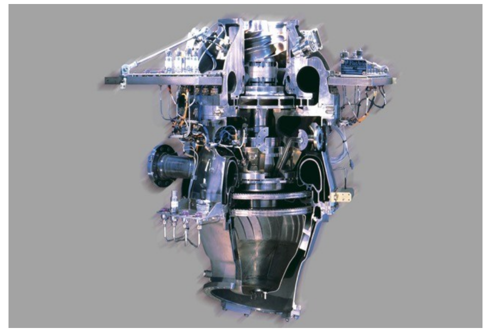
Air-sea coupling over western boundary oceanic currents: benefits of high-resolution models in the representation of physical mechanisms and in present and future climate impacts



Western boundary current (WBC) regions such as the Gulf Stream and the Kuroshio in the Northern Hemisphere and the Agulhas in the Southern Hemisphere, contribute to the transport of warm and salty waters from the equator to higher latitudes and are characterized by sharp sea surface temperature (SST) gradients (from 4ºC to 10ºC in 100 km) as well as an important eddy activity. Thus, they have a relevant role in the global climate system. Over the past decades, a better understanding of air-sea coupling in WBC regions has become possible with the development of new satellite measures. Model simulations suggest that stronger SST gradients as represented by higher resolution climate models, can affect the position and strength of the atmospheric storm tracks, and hence they can impact mid latitude weather. However the mechanisms through which oceanic fronts can influence the large scale atmospheric circulation remain unclear and model dependent. Up to now only few studies have investigated this problem using fully coupled climate models, which is essential to fully understand how air-sea coupling is affected by stronger oceanic fronts. The goal of this PhD work is to improve our understanding of physical processes occurring over WBC regions, in order to better understand the role of oceanic fronts in present day climate variability, and how their role might change in a warming planet.

Matthieu QUEGUINEUR (CERFACS, CFD)

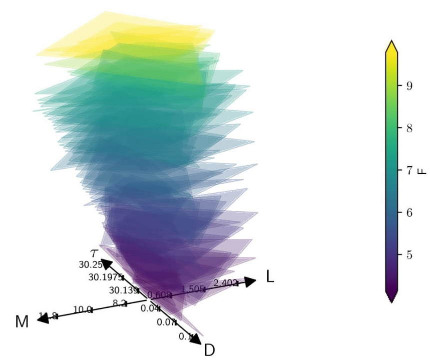
Investigation of unsteady phenomena in rotor/stator cavities on a spatial turbopump turbine stage



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 programs often reveal unidentified unsteady pressure phenomena in rotor/stator cavities of turbines. Such features, named pressure bands, have proven to be 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 problemsand an industrial case. The next step, motivated by this thesis , is the refinement of the industrial case□smodel. As such, the blades of the rotor will be taken in consideration. Furthermore, in order to validatethe studies conducted by T. Bridel, a numerical control of the instabilities sources will be set up.

Pamphile ROY (CERFACS, CFD)

Uncertainty Quantification in a Costly Numerical Environment

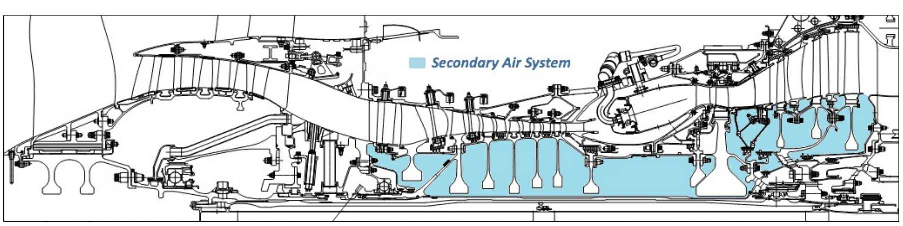


Computational Fluid Dynamics (CFD) simulations are standard tools for reducing the number of iterations between design phase and industrialization. The maturity and performance of these solvers allow to reliably consider parametric studies, robust optimization, robust design assessment and Uncertainty Quantification (UQ) analysis. UQ methods aims at providing tools for: (I) discovering the importance factor contributing to the Quantity of Interest (QoI) ; (ii) ranking them ; (iii) assessing risk ; and (iv) propagating uncertainties on the simulation parameters. These experiments are often computationally expensive and surrogate models can be constructed to address this issue. My work focuses on handling high-dimensional parameter space. We proposed new methods to refine the design of experiments as to retrieve as much information as possible. These methods have been applied to both analytical and industrial configurations successfully. Another concern about a high-dimensional space is its visualization. Besides all the work and interest that has been devoted to this field, the community has yet to propose a systematic way to observe uncertainties. Hence, we propose some tools and methods to assess this question.

Gauthier WISSOCQ (Safran Aircraft Engines, CFD)

Application of the lattice Boltzmann method to the rotating cavity flows of turbomachine secondary air systems

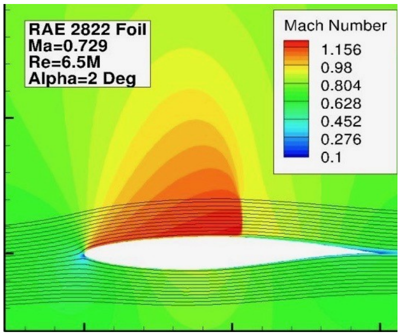
In the rotating cavities of secondary air systems of turbomachines, natural convection can be induced by the buoyancy force due to inertial forces. This phenomenon results in complex non-axisymmetric unsteady and unstable flows. 2D-axisymmetric and 3D RANS methods have proved their unability to reproduce this behaviour, while correct temperature and heat tranfer predictions are of paramount importance in the design of turbomachine components. In this context, the Lattice-Boltzmann Method (LBM) emerges as a promising technique for the simulation of unsteady flows. The simple numerical scheme of the LBM makes it well suited for parallel computing, and the use of cartesian mesh allows to easily deal with complex geometries. 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 turbomachines. The commercial LBM code PowerFLOW is used together with the thermal analysis code PowerTHERM in order to take into account conductive and radiative heat transfer between solid surfaces and the fluid.



SESSION 4

Florian RENARD (CERFACS, CFD)

Extension of the lattice Boltzmann method to compressible aeronautical flows



The lattice Boltzmann Method (LBM) turned out to be a serious competitor face to standard Navier-Stokes solvers those last two decades. It is an interesting alternative which allows to accurately model a wide range of complex physical phenomena from blood flows in vessels to aeronautic flows with a relatively low computational cost. Indeed, one of the strengths of LBM is its ability to easily handle very complex geometries of realistic configurations. This is possible through the octree based refinement technique which perfectly fits with the Cartesian nature of the LBM. Unfortunately the standard construction of the LBM only permits the simulation of isothermal weakly compressible flows. Nevertheless, others constructions allowing the simulation of thermal compressible flows have been performed, leading either to higher computational cost, substantial loss in accuracy or critical instabilities. The main objective of this thesis is to develop a fully compressible LBM model, based on the Hybrid method, able to model standard compressible test cases followed of applications to external transonic flows and internal flows in turbomachine.

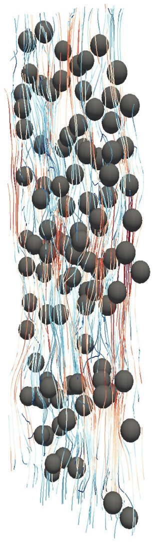
Simon BLANCHARD (CERFACS/CNES, CFD)

Multi-physics Large-Eddy Simulation of cryogenic combustion in rocket engines

Combustion in liquid rocket engines happens in extreme conditions which imply several multi-physics phenomena. For this reason, numerical simulation is used to predict and thus to optimize the engine performances and lifetime. In particular this thesis focuses on: modelling and chemistry for non-premixed turbulent oxy-combustion of methane; prediction of heat transfers at the walls including the influence of gas composition; description of the transition between the sub-/ and supercritical states of the fluid. Finally with all these elements, a multi-physics coupled simulation will be carried out in a real rocket engine configuration.

Dominik SCHUSTER (CERFACS, CFD)

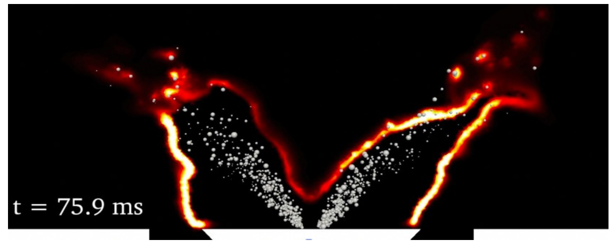
A new approach for numerical flow simulations in the domain of biomedicine, petroleum and aeronautics



CFD (computational fluid dynamics) is at the heart of developments in many fields such as aeronautics, oil industry or the biomedical field. In recent years, a new method has appeared in CFD: LBM (Lattice Boltzmann method) codes. These codes change the paradigm of the CFD by solving the Boltzmann equation instead of Navier Stokes and thus considerably simplifying the mesh phases, making calculations in complex and mobile geometries possible. Long seen as a scientific curiosity, LBM methods have proven their potential in aeroacoustics for example. RENAULT and AIRBUS have already implemented LBM codes for some of their production. Their low calculation costs make them ideal candidates for real-time applications or deployment in small and medium sized enterprises. However, this evolution must be done by mastering the price / performance ratio and precisely assessing the areas where LBM techniques are competitive.

Varun SHASTRY (CERFACS, CFD)

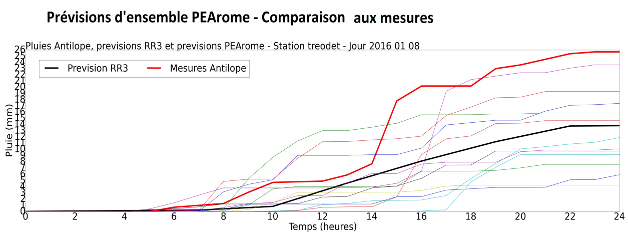
Numerical study of thermo-acoustic instabilities in spray flames

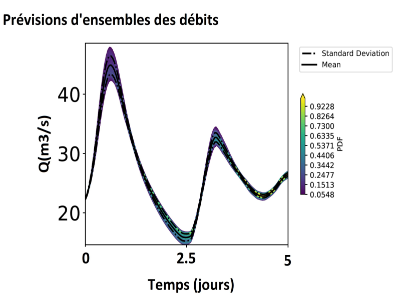


Lean fuel air mixtures used in aircraft engines to achieve low emission levels may induce strong thermoacoustic oscillations. The MAGISTER project aims to study and predict these thermoacoustic problems using adaptive machine learning and complex computational tools. The interaction between liquid spray flames and thermoacoustic oscillations in complex geometries is still unclear. The goal of this PhD is to perform numerical simulations of acoustically forced or self-exciting liquid fuel in lean aero engine combustors at high pressure and reduced wall cooling air. Improved spray models to represent multicomponent fuel droplets subjected to acoustic fields as well as models for acoustic liners will be developed and used to perform LES of pulsated spray flames. Flame Transfer Functions extracted from the LES simulations will be used to study the acoustic instabilities in pulsating spray flames. The results will be compared with experimental studies done by other project partners of MAGISTER.

Anne-Laure TIBERI-WADIER (Université Paris-Est, GLOBC)

Ensemble flood forecasting with an integrated rainfall-hydrology-hydraulic approach

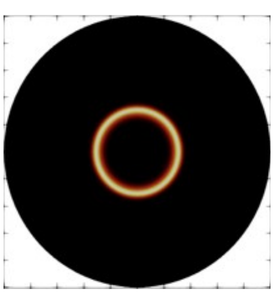




SCHAPI and SPC (i.e. flood forecasting services) use day-to-day deterministic models, while taking into account rain forecasts and runing hydrological and hydraulic forecast models. Each modelling step is affeceted by uncertainties. These uncertainties impact the reliability of a deterministic forecast. A ensemble approach is an efficient way to undersand forecast uncertainties. This PhD aims at implemeting an ensemble prediction chain from rain forecasting to discharge forecasting. Severals studies deal with hydrological ensemble forecasting. In this study, the ensemble prediction chain will be extended until the hydraulic model, the last element of the whole prediction chain. Data assimilation will also be introduced to reduce uncertainties in the forecast chain. Meteo France Arome ensemble prediction of rain is used as input of the two hydrological models GRP and MORDOR. Then, hydraulic simulations are performed with the numerical 1D model MASCARET.

Paul POUECH (CERFACS, CFD)

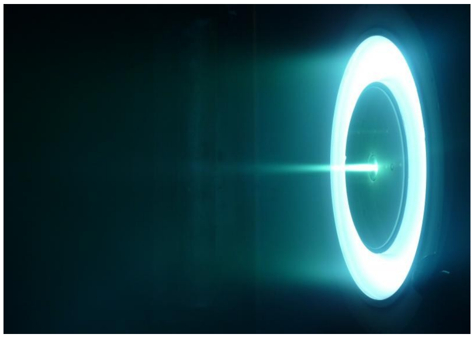
Large eddy simulation of isochoric combustion chamber and turbine interactions



Aeronautical engines are in constant improvement, with the objective of increasing efficiency and reducing pollutant emissions. Isochoric combustion is one of the new approaches considered to reach this goal, as it significantly improves the combustion process. Using an Humphrey thermodynamical cycle instead of a Brayton cycle, related to isobaric combustion, constant-volume combustion shows a 30% increase in terms of engine efficiency, meaning less fuel is needed to reach constant-pressure engines performances. This thesis will aim at numerically simulating a new industrial engine which uses isochoric combustion, focusing on injection of liquid fuel, evaporation and ignition in turbulent flows, the influence of cycle-to-cycle variations on engine efficiency and turbine coupling.

Willca VILLAFANA (CERFACS, CFD)

Modeling of physical phenomena in a Hall-effect thruster and assessment of their impacts on performances and wall erosion



Today, satellites and space vehicles tend to operate more and more with electric-based propulsion systems such as Hall-effect thrusters, spurred by ambitious projects such as OneWeb and recent successes like the Smart-1 lunar mission. Initially developed by the USSR in the 1960's, the plasma dynamics within Hall-effect thrusters is still not well-understood and current modeling fails to capture or accurately predict physical phenomena such as electronic transport or wall erosion. As a result, space companies struggle to develop cutting-edge devices to meet the market demand. In this context, under the Poséidon research project, CERFACS, Safran Aircraft Engine and the Plasma Physics Laboratory of Ecole Polytechnique united their efforts and skills in 2016 to develop AVIP, an unstructured parallel-efficient 3D code suitable for real industrial geometries. Fluid and kinetic/PIC approaches have been implemented and validated for some classical benchmarks but additional modeling needs to be done to perform a realistic 3D simulation. Among objectives but not limited to, this PhD aims to model plasma-wall interactions and heat transfers and assess their impact on the thruster's performances and lifetime. Ultimately, validations with experiments and actual computational requirements will decide which modeling, fluid or PIC, should be applied for optimal results.