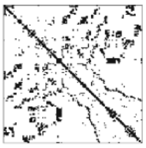
CERFACS

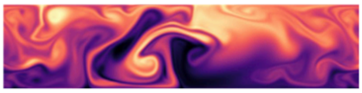
CENTRE EUROPÉEN DE RECHERCHE ET DE FORMATION AVANCÉE EN CALCUL SCIENTIFIQUE

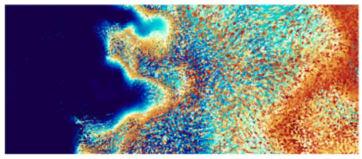
Ph.D. Students' Day (JDD 2024)

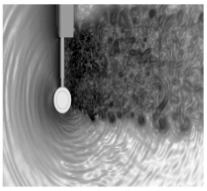
Thursday May 30th 2024

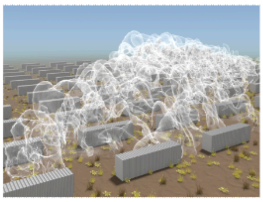


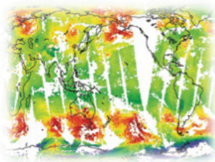
Algo-Coop



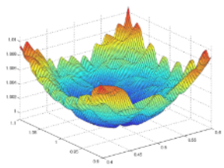


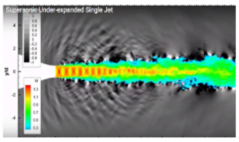


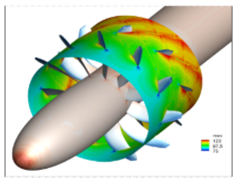


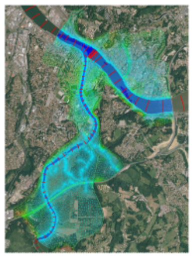


Climate Modeling & Global Change

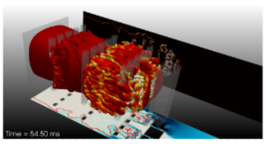


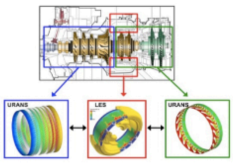


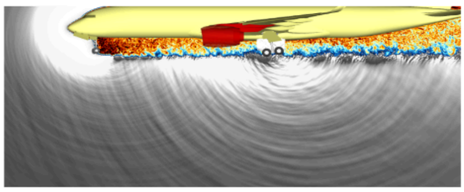




Computational Fluid Dynamics







AIRBUS · EDF · ONERA THE FRENCH AEROSPACE LAB · TotalEnergies

cnes CENTRE NATIONAL D'ÉTUDES SPATIALES · METEO FRANCE · SAFRAN

http://cerfacs.fr

09:15	**Welcome Coffee**			
09:30	**Welcome speech: Laurent Gicquel**			
Session 1		Chair: Paul Mycek		
09:45	Anthony Dupuy	*A low reflexion, fast convergence NSCBC methodology for transient flows*	p.5	E&S
09:50	Emile-Marie Berthoumieu	*LES model construction for H2/air combustion*	p.6	E&S
09:55	Alexandre Dutka	*Numerical methods for exascale performance portable simulations*	p.6	AAM
10:00	Alexis Boudin	*Numerical Methods for Large Eddy Simulations in Turbomachinery*	p.7	E&S
10:05	Éloïse D'Ayer	*Modeling Surface Roughness with Immersed Boundary Conditions and the Spectral Difference Method*	p.7	AAM
10:10	**Coffee break and Poster Session n°1 : A. Dupuy, E. Berthoumieu, A. Dukta, A. Boudin and É. D'Ayer**			

Session 2		Chair: Ludovic Cassan		
11:00	Robin Cazalbou	*Optimizing code coupling in a massively parallel hybrid CPU-GPU environment*	p.8	E&S
11:05	Quentin Bonassies	*Assimilation of remote sensing flood extent for flood and inundation forecasting*	p.9	GlobC
11:10	François Collet	*Evolution of winter compound low wind and cold events impacting the French electricity system over the 1950-2022 period.*	p.10	GlobC
11:15	Lakshmanan Gopalakrishnan	*Aeroacoustic simulation of high lift devices using the lattice Boltzmann method*	p.11	AAM
11:20	Raul Vazquez Casique	*Aeroacoustic simulation of an installed propeller via a lattice Boltzmann approach*	p.12	AAM
11:25	**Poster Session n°2: R. Cazalbou, Q. Bonassies, F. Collet, L. Gopalakrishnan and R. Vazquez Casique**			
12.00	**Lunch**			

Session 3		Chair: Laurent Gicquel		
14:00	Vincze Balazs	*Turbulent multiphase combustion modeling for the Large Eddy Simulation of post-combustors*	p.13	E&S
14:05	Loïc De Nardi	*Hydrogen deflagrations in confined spaces and flame-wall interaction*	p.13	E&S
14:10	Justin Bertsch	*Large Eddy Simulation of an aeronautical hydrogen-air combustion chamber*	p.14	E&S
14:15	Lukas Gaipl	*Numerical simulation of jet flames in cross flow with flame-wall interactions*	p.15	E&S
14:20	Eric Matas Mur	*High-Fidelity simulation of flame acceleration on stratified H2/air mixtures*	p.16	E&S
14:25	**Coffee Break**			

Ma thèse en 3 minutes		Chair: Matthieu Pouget		
14:40	Mehdi Ettaouchi	*Parallel Nonlinear Schwarz Domain Decomposition Solvers for Two-Phase Flow in Porous Media*	p.17	Algo-Coop
14:45	Hector Vargas	*High-Performance Large Eddy Simulations of Ammonia Hydrogen Industrial Gas Turbines*	p.18	E&S
14:50	Mickaël Theot	*Modeling and simulation of hydrogen-enriched gas turbines with exhaust gas recirculation for efficient integration of CO2 capture*	p.18	E&S
14:55	Imane Fadli	*Aerodynamic optimization with uncertain shape parameters*	p.19	AAM
15:00	Arthur Berthelot	*Development of methodologies and tools for non-polluting combustion*	p.19	E&S
15:05	Aldo Schioppa	*Modelling of Flame Transfer Function of Lean H2 Turbulent Flames in a Real Ground-Based Turbine*	p.20	E&S
15:10	Théo Briquet	*Learning for predictng the rank of hierarchical matrices*	p.21	Algo-Coop
15:15	Solène Hoflack	*Modelisation and Large Eddy Simulations of turbulent ammonia/air flames*	p.21	E&S
15:20	**Poster session n°3: V. Balazs, L. De Nardi, J, Bertsch, L. Gaipl, E. Matas Mur, A. Berthelot, M. Ettaouchi, M. Theot**			
16:00	**Voting for best Poster and best "Ma thèse en 3 minutes" presentation**			
16:15	**Closing speech: Catherine Lambert**			

Ma thèse en 3 minutes

Some students have chosen to do their presentation under the “Ma thèse en 3 minutes” format, which is a theatrical science communication exercise. They entered a specific training course given by Dr. Matthieu Pouget, who holds a PhD in Performing Arts from the Université de Toulouse and is a professor at the Conservatoire d’art dramatique de Toulouse. He guided them throughout the process of writing the script, creating their performance, and learning how to communicate their research subject to a wide audience in the format of the national competition “Ma thèse en 180 secondes”.

The presentations will be in French. The participants who have prepared a poster will move on to a regular poster session in English.

Anthony Dupuy

09:45 (poster 10:10)

A low reflexion, fast convergence NSCBC methodology for transient flows

A robust methodology for non-reflecting inlet and outlet boundary conditions for subsonic flows to both reach faster convergence and dampen acoustic modes is presented. Both objectives are achieved by allowing numerical waves to leave the domain, i.e. ensuring low reflexion at the inflow and/or outflow boundary conditions. Using the characteristic boundary condition (NSCBC) with a linear relaxation [Poinsot and Lele, JCP, 1992] as well as the estimation of the outgoing wave from NRI method [Daviler et al. Comput. Fluids, 2019], this more robust strategy use an exponential moving average (EMA) filtering of the outgoing wave. To avoid any numerical divergence from the targeted value like in NRI, a frequency-domain scale separation between acoustic and mean flow contributions of the outgoing wave is mandatory. In subsonic flows, the acoustic contributions are associated with high frequencies and mean flow contributions with low frequencies. The objective is to damp as much acoustic energy as possible and correct only the mean flow contributions with the linear relaxation. Unlike the classical NSCBC linear relaxation formalism, this Non-Drifting Non-Reflecting (NDNR) method allows a fast convergence with a low reflexion at the same time. The choice of the cutoff frequency of the EMA filter is analytically developped and can be adapted to each case depending on the operating conditions and geometry. The reflexion coefficient and the convergence time on a simple test case are in good agreement with the theory and show the robustness and the greater flexibility of NDNR: being the convolution of two first order low-pass filters, it enables a greater flexibility on K choice while keeping high acoustic fluxes due to the EMA filtering.

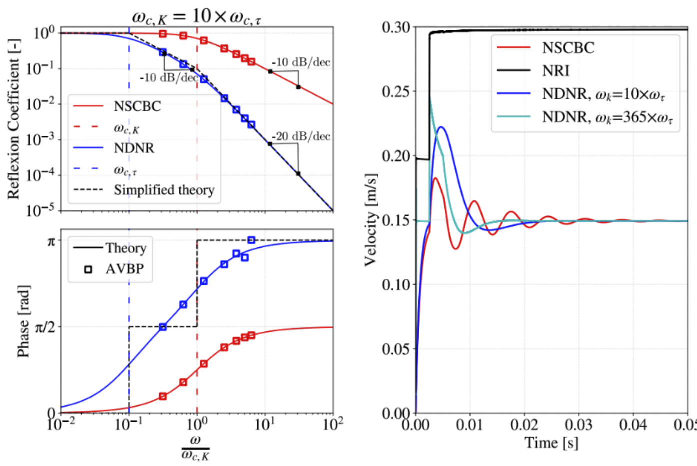


Figure 1: Left : Theoretical and numerical simulation gain and phase of inlet Boundary Condition for NSCBC linear relaxation and NDNR methodology when the cutoff frequency of the EMA filter, $\omega_{c,\tau}$, is ten times lower than the cutoff frequency of the relaxation term, $\omega_{c,K}$. Right: Convergence time for a Poisseuille flow set up with AVBP solver using NSCBC, NRI and NDNR methods (with $\omega_{c,K} = 10\times\omega_{c,\tau}$ and $\omega_{c,K} = 365\times\omega_{c,\tau}$) at the inlet and a fixed pressure at the outlet.

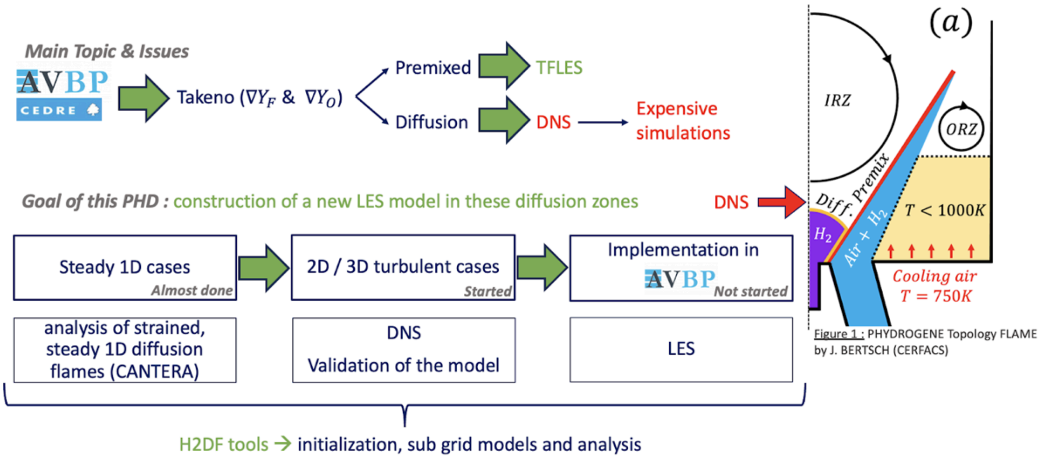
chambers where strong temperature gradients take place.

09:50 (poster 10:10)

Emile-Marie Berthoumieu

LES model construction for H2/air combustion

The aim of this thesis is to develop a new LES model specifically for diffusion zones. To this end, a three-part methodology has been developed. Firstly, the topological study of steady, one-dimensional diffusion flames via CANTERA in order to create a new one-dimensional model. Secondly, the study of 2D and 3D stable or turbulent flames in AVBP to verify the



veracity of the 1D model. Additionally, to define the reference index through a less and less refined flame study. Finally, the implementation of the models in AVBP built up over the course of the thesis.

09:55 (poster 10:10)

Alexandre Dutka

Numerical methods for exascale performance portable simulations

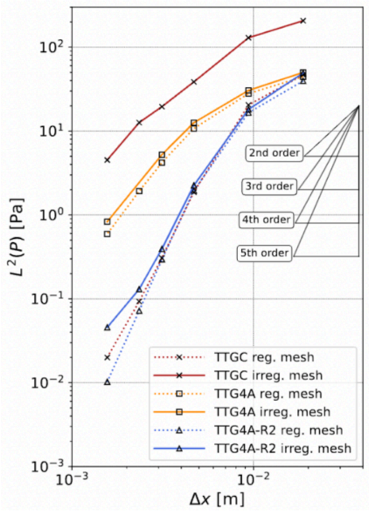
Computational fluid dynamics (CFD) has become a great tool for understanding fluid dynamics phenomena. Many mean flow problems can be computed on a laptop thanks to the Reynolds-Averaged Navier-Stokes (RANS) approach, but more subtle phenomena such as vortex dominated flows can't reliably be captured with these low-order methods. For such problems, large eddy simulation (LES) is better suited, but this unsteady approach is often orders of magnitude more expensive. This work investigates both high-performance computing (HPC) techniques and numerical methodology improvements in order to lower the computational cost of high-fidelity LES. A performance portable implementation of the spectral difference method is developed for adaptive hierarchical grids, incorporating new diffusive and temporal schemes which improve the efficiency of the method. Finally, performance and scalability comparisons on various HPC architectures are carried out.

10:00 (poster 10:10)

Alexis Boudin

Numerical Methods for Large Eddy Simulations in Turbomachinery

Large-Eddy Simulations (LES) brought new capacities in Computational Fluid Dynamics (CFD). Being less costly than Direct Numerical Simulations (DNS) from a computational point of view, it nevertheless solves the unsteadiness part of the Navier-Stokes equations. To achieve this, numerical methods are used involving temporal and spatial discretizations. In AVBP, Taylor-Galerkin methods based on hybrid finite elements / finite volumes formulations are used. These methods have a given accuracy and robustness which are generally linked to their computation cost. The goal of this thesis is to improve these two aspects of the methods while keeping the cost as low as possible. These new methods allow either to get better results on the meshes currently in use either to be able to reduce the quality of the latter while keeping the same accuracy.

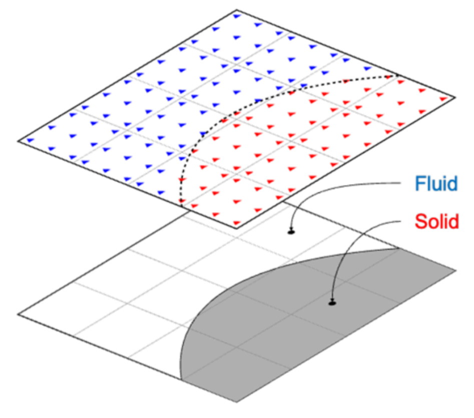


10:05 (poster 10:10)

Éloïse D'Ayer

Modeling Surface Roughness with Immersed Boundary Conditions and the Spectral Difference Method

The presence of roughness will affect performances of aeronautical surfaces. In order to develop new RANS models to take roughness effects into account, ONERA needs to generate a DNS database of flow over rough surfaces. Such databases are hard to generate, because of the high computational cost of the simulations, but also because the surfaces are hard to mesh, and the results needs to be accurate enough. Immersed Boundary Conditions will be used to solve the meshing problem, and the JAGUAR solver, developed by CERFACS and ONERA, will be used to provide accurate results with good computational efficiency.

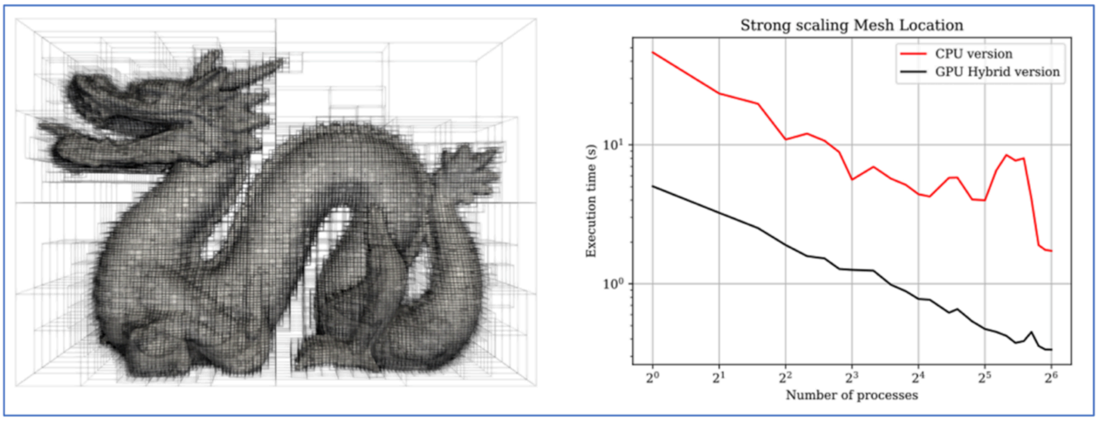


11:00 (poster 11:25)

Robin Cazalbou

Optimizing Code Coupling in a Massively Parallel Hybrid CPU-GPU environment

Nowadays, computing codes involve increasingly complex modelling techniques and numerical methods. The interaction between those specialized codes can be achieved using the CWIPI coupler, which performs interpolations at the coupling interface, while exploiting the massively parallel architectures of modern supercomputers through dynamic load balancing. A key step in CWIPI coupling involves locating the nodes of a target mesh in the elements of a source mesh. This operation is carried out internally by the ParaDiGM mesh manipulation library, which constructs a distributed octree on the point cloud and traverses it with the bounding boxes of the source mesh elements, in order to reduce to a logarithmic complexity. Even so, this step is time-consuming, especially when performed at each time steps in coupled simulations with large, moving meshes. In this context, this thesis work consists in particular of developing multi-GPU octree construction and traversal algorithms in order to speed up the location step. Using a bottom-up approach, distinct from the top-down one usually used in the literature, a speed-up of x5 to x10 has been measured on the



(Left) Octree built on the Stanford Dragon cloud (Right) Strong scaling study of the mesh location step involving the octree, on the Topaze supercomputer. New multi-GPU method scaling is as good as the CPU version, with a speedup.

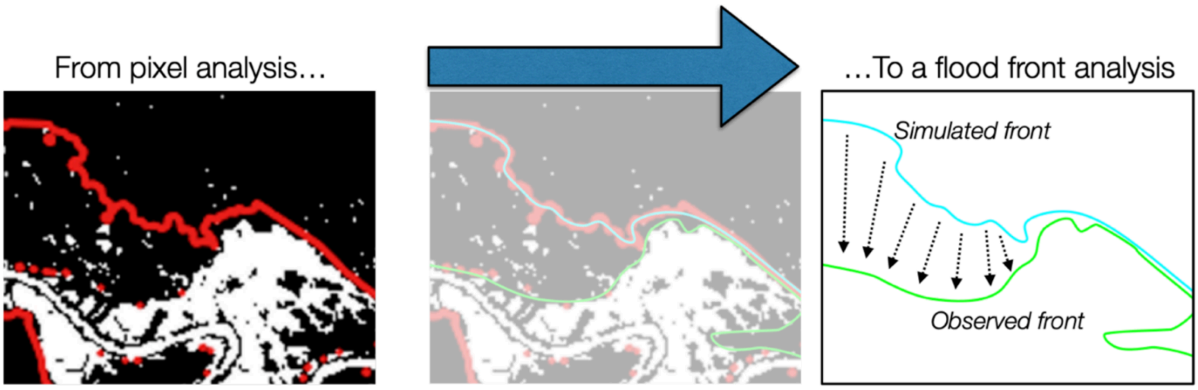
execution of the mesh location algorithm, and a speed-up of x25 to x120 for the construction of the octree on large point clouds. Further optimizations of other coupling stages will enable even higher acceleration factors.

11:05 (poster 11:25)

Quentin Bonassies

Assimilation of remote sensing flood extent for flood and inundation forecasting

Hydrodynamic models that provide water elevation 2D maps on a computational domain are essential tools for flood risk assessment and anticipation. Data assimilation (DA) allows to reduce the uncertainties in model inputs, parameters and outputs. This research work presents an innovative framework to assimilate flood extents derived from synthetic aperture radar (SAR) images by considering such observations as front-type wet-dry interface information. In order to deal with complex front topology, an object-oriented approach based on the Chan-Vese (CV) contour fitting functional, typically used in image processing, was proposed and implemented with an Ensemble Transform Kalman Filter (ETKF) algorithm. This strategy was demonstrated successful when applied on a simplified test case and is currently extended to a real case on the Garonne River.

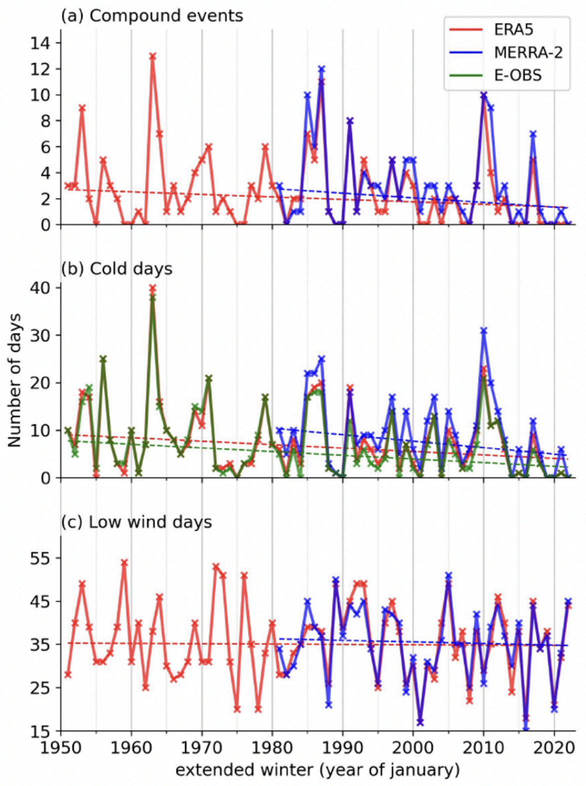


11:10 (poster 11:25)

François Collet

Evolution of winter compound low wind and cold events impacting the French electricity system over the 1950-2022 period.

In the context of the energy transition, it is anticipated that the share of wind power in the French power system will substantially increase. The power production is expected to rely more on wind power production in the future, therefore on wind speed conditions. Compound low wind power production and high electricity demand events, which are associated with compound low wind and cold events occurring in winter in France, are events that will pose a more important risk to the security of electricity supply in the future. Anthropogenic GHGs emissions have led to substantial changes in the French climate, including extreme events. In particular, the intensity, frequency, and duration of cold events have decreased. However, there remain high uncertainties on the evolution of low-wind events in the context of climate change, and therefore on the evolution of compound low wind and cold events. This study aims at characterizing the evolution of compound winter low wind and cold events over the 1950-2022 period, over which observation data are available. Compound events are identified at the daily scale using indices for wind power production and electricity demand in France that are derived from temperature and wind speed. We show that the frequency of compound events decreases over the 1950-2022 period and that atmospheric circulation changes do not play a strong role in this decrease. This study is part of my PhD thesis, which aims at characterizing the evolution of these events over the 21st century.



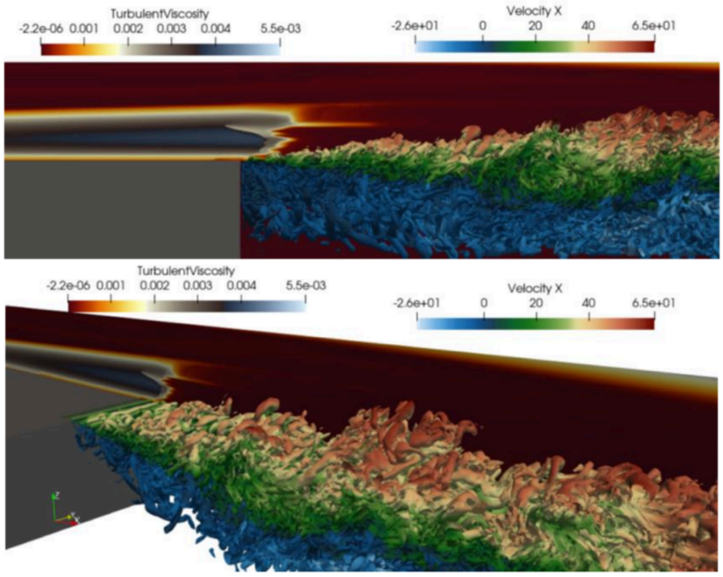
Evolution of the number of (a) compound low wind and cold events, (b) cold days, (c) low wind days per winter in ERA5 (in red; 1951-2022), MERRA2 (in blue; 1981-2022) and EOBS (in green; 1951-2022) datasets. Dashed lines show the linear trend (calculated with the Theil-Sen estimator).

11:15 (poster 11:25)

Lakshmanan Gopalakrishnan

Aeroacoustic simulation of high lift devices using the lattice Boltzmann method

The lattice Boltzmann method has proven to be an alternative and faster means of carrying out CFD simulations over the traditional Finite Volume or Finite Difference based Navier Stokes (NS) solvers and has lower dissipative properties compared to the latter. This makes LBM a suitable candidate for aeroacoustic simulations. In the recent years, Hybrid RANS/LES method which switches towards LES method only in regions where large velocity gradients are observed have been widely used for high Reynolds number flows. One such method is the ZDES mode 2 Enhanced Protection (EP) which has been observed to completely shield the boundary layer from prematurely switching to LES compared to the original DDES method on which it is based on, thus avoiding the drawbacks of DDES. The use of ZDES mode 2 EP with LBM is challenging and has only been recently validated. With the advantages LBM brings over NS, it is envisaged to carry out and validate aeroacoustic simulations of high lift devices using ZDES mode 2 EP on wings whose experimental data are available. Validation studies on benchmark test cases will be carried out initially before moving on to complex geometries. New wall treatment strategies will also be implemented and tested to study the advantages and disadvantages of using these strategies

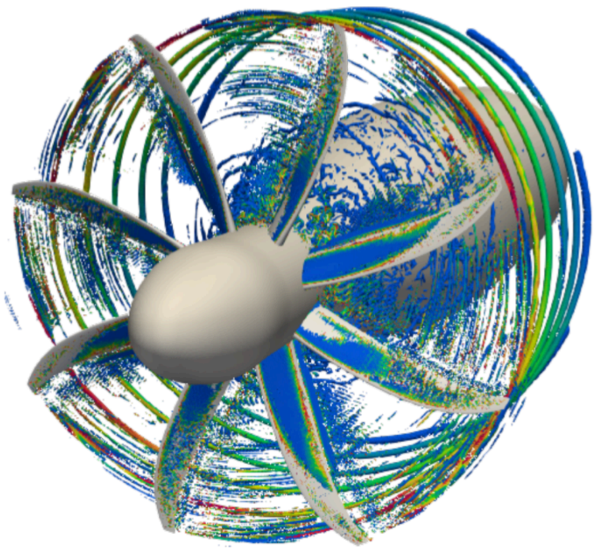


11:20 (poster 11:25)

Raul Vazquez Casique

Aeroacoustic simulation of an installed propeller via a lattice Boltzmann methods approach

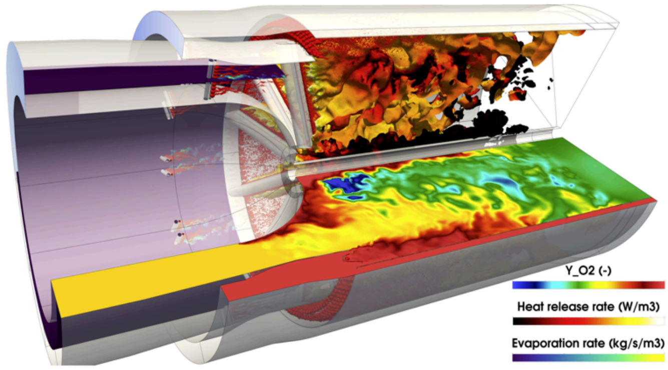
The lattice Boltzmann method has recently emerged as an advantageous method for the simulation of aeroacoustic phenomena thanks to its low numerical dissipation and its effortless integration with complex geometries. The simulation of rotating geometries is of great industrial interest for a wide range of applications, such as for turbomachinery. In an LBM context, the simulation of such flow configurations has been made possible by the use of overset grids (also called Chimera grids), where rotating and fixed meshes coexist over a common overlap region. Communications between grids occur via the interpolation of relevant variables at their respective boundaries from nodes located in this region. Up to now, the overset grids approach requires nodes inside the overlap region to be uniform and have the same size on both grids. This is a major constraint on the simulation of turbomachinery applications, such as for installed propellers. For such configurations, the addition of nodes following the blade tip vortices is desirable to accurately capture their interaction with downstream surfaces such as wings and high-lift devices. Nonetheless, the introduction of mesh refinements close to blade tips would require refining over all the overlap regions, resulting in a major increase in the number of nodes and thus computational cost. To overcome this, this thesis focuses on the modelling of non-uniform interpolation on the overset grids approach in an LBM framework for the aeroacoustic simulation of an installed propeller.



14:00 (poster 15:20)

Balas Vincze

Turbulent multiphase combustion modeling for the Large Eddy Simulation of post-combustors



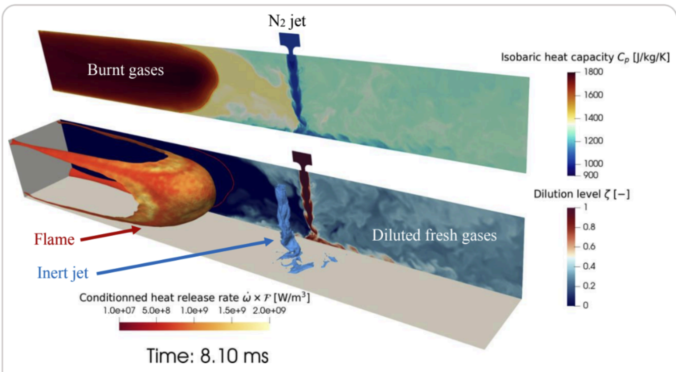
Post-combustors (or afterburners/reheat chambers) are a unique part of jet engines that power supersonic-capable aircraft. They are placed after the low-pressure turbine and are fed by the (diluted) combustion products from the primary chamber; their goal is to provide additional thrust for specific maneuvers (up to 50% in modern cases). Our objective is increasing the Technology Readiness Level (TRL) of Large Eddy Simulations (LES) to model combustion in a post combustor. The particular context of a post-combustor: i.e., large volume, high flow velocities, complex carburaJon systems... makes a multitude of modeling issues appear and several topics need to be revisited: kerosene chemistry with autoignition, non-premixed flame regimes, Lagrangian liquid phase modeling for fuel atomization, evaporation and interaction with flames.

14:05 (poster 15:20)

Loïc De Nardi

Hydrogen deflagrations in confined spaces and flame-wall interaction

Hydrogen has become one of the most promising energy carriers to face global warming and make aviation carbon-free. However, its enhanced reactivity raises safety issues that are critical for zero-emission aircrafts design. The present thesis is part of a cooperation between CERFACS and AIRBUS, in order to study hydrogen deflagrations within realistic flight conditions. Most state-of-the-art explosions are studied experimentally and numerically in quiet chambers. Using AVBP, a combustion code developed at CERFACS, the goal of my thesis is to address hydrogen deflagrations in turbulent flows and to develop numerical models that are able to capture the physics that drives flame propagation.



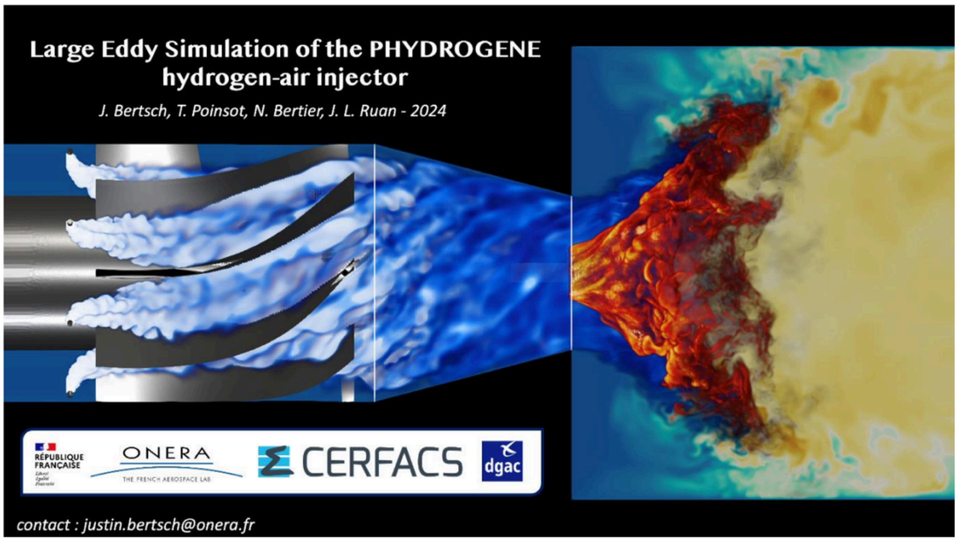
Hydrogen-Air deflagration interacting with a transverse jet of N_2 at high-pressure.
Operating conditions are: ϕ=1, T_0=300 K, P_0=0.2 bar, P_{jet}=4 bar.

14:10 (poster 15:20)

Justin Bertsch

Large Eddy Simulation of an aeronautical hydrogen-air combustion chamber

The goal of this thesis is to develop a new hydrogen injector for an aeronautical combustion chamber through computed fluid dynamics using two solvers, AVBP developed at CERFACS and CEDRE developed at ONERA. This research take place as part of the PHYDROGENE project. Simulation results are used to analyze and understand the flame topology, propose upgrades for the geometry and give directions for the optimization process of the injector shape. Phydrogene injector consist of a lean premixed swirled hydrogen-air mixture with a pure hydrogen injection at the tip of the injector (pilot injection). High fidelity numerical simulations of the Phydrogene injector are conducted at elevated pressure (12 bars) corresponding to an aeronautical working point. Numerically, high pressure premixed hydrogen flames leads to challenging computations since the flame front is very thin as it scales as the invert of pressure. Thickened Flame Model is used to solve this issue and



High pressure swirled aeronautical hydrogen air injector with a pure hydrogen pilot injection

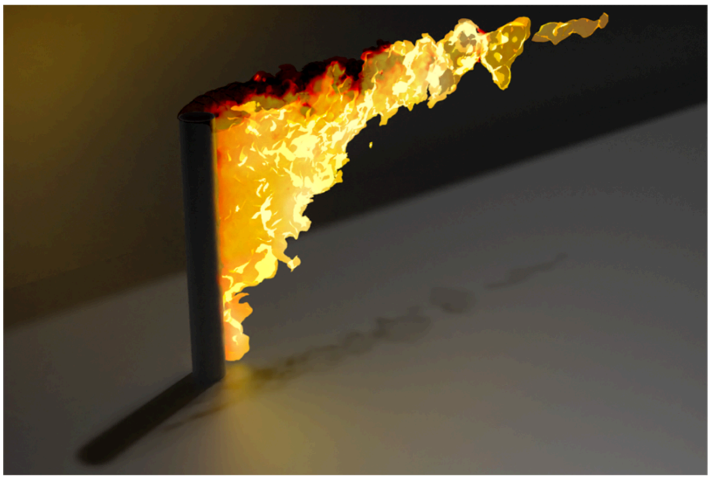
resolve the premixed flame front. The flame structure anchored at the injector's lips exhibits a unique structure consisting of a lean premixed flame close to a pure hydrogen diffusion flame due to the pilot injection. This structure has been studied in 1D and 2D simulations leading to a submitted paper.

14:15 (poster 15:20)

Lukas Gaipl

Numerical simulation of jet flames in cross flow with flame-wall interactions.

The objective of the thesis is the numerical simulation using LES of partially-premixed flames, which are created by a jet of combustible gas in a cross-flow of air. This configuration is used in many technical applications such as torches (gas industry), gas turbines and many safety scenarios, if a gas leak ignites in ambient air, for example in future hydrogen-propelled airplanes. The interaction of turbulence and combustion is the centre of research throughout



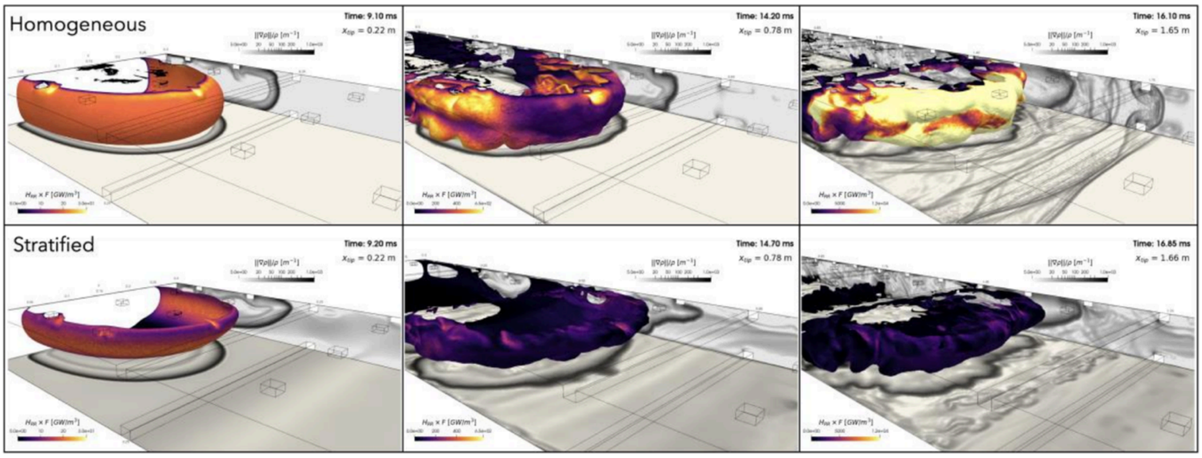
all studies. The first part of the thesis is dedicated to 'free' flames e.g., without any obstacle. The stabilisation of the flame in the vicinity of jet lips and the interaction of turbulence and chemical kinetics of two extreme cases is studied: methane with a slow proceeding and hydrogen with a very fast proceeding chemistry. A typical simulation is shown in Figure 1. In the second part, these flames are placed close to walls and the interaction of flames and walls is studied: the kinetics close to the wall are re-examined as well as turbulence models in the vicinity of walls. The presence of a flame generates extremely high fluxes and the coupling of thermal insulation of the wall is investigated.

14:20 (poster 15:20)

Eric Matas Mur

High-Fidelity simmixturesulation of flame acceleration on stratified H2/air .

Hydrogen is a current focal point in the industrial sector, emerging as a viable and sustainable fuel option for decarbonizing energy production through combustion. However, this fuel poses significant safety risks due to its high flammability, colorless and odorless nature, and propensity for leakage. Typically, an explosion scenario follows this sequence: (1) fuel release, (2) formation of a flammable fuel cloud, (3) ignition, (4) explosion. There are two explosion regimes: deflagration, with subsonic flame propagation and mild pressure waves, and detonation, characterized by flames propagating at supersonic speeds, large overpressures and shock waves, resulting in immense destructive power. Incidents usually end at the deflagration stage. However, under certain conditions, if the flame accelerates enough, the flame may undergo a deflagration to detonation transition (DDT). This PhD examines various explosion scenarios to analyze a critical aspect of explosions: the heterogeneity/stratification of the fuel mixture. This aspect is often overlooked due to its



complexity, yet it is prevalent in almost every explosion case. In reality, during leaks, fuel mixtures are hardly ever homogeneous. The aim of this thesis is to deepen the understanding of explosion scenarios by investigating stratification, thereby establishing more effective safety measures based on more realistic scenarios. To achieve this, the AVBP code developed at CERFACS will be employed for High-Fidelity calculations in canonical explosion cases.

3'

14:40 (poster 15:20)

El Mehdi Ettaouchi

Nonlinear Schwarz Domain Decomposition Solvers for Two-Phase Flow in Porous Media

In the context of the storage of radioactive waste, the need for advanced nonlinear solvers becomes evident in simulating the neighboring geological environment. These models for two-phase flow equations in porous media are highly nonlinear and are solved over timeframes that could reach millions of years, presenting a significant challenge. Addressing this requirement, innovative nonlinear solvers have been developed and applied, showcasing their effectiveness in handling complex environmental simulations These solvers stand out due to their use of nonlinear domain decomposition methods, which not only enable efficient parallelization but also significantly improve both nonlinear and linear convergence rates. This is achieved by partitioning the problem into subdomains and solving a smaller subproblem within each. Thanks to an implementation using petsc4py, the solvers capabilities were validated through their application to an international benchmark that models the injection of hydrogen into an initially saturated porous medium. This application highlighted the solver's robustness and rapid convergence, essential qualities for the simulation of geological environments. Moreover, the challenge of communication slowdown, which becomes apparent when dealing with a large number of subdomains and negatively affects the linear inner loops, is effectively mitigated. A second-level strategy employing a nonlinear multigrid approach, specifically the Full Approximation Scheme (FAS), addresses this issue. The FAS approach stabilizes the number of inner loops regardless of the number of subdomains, ensuring scalable efficiency suitable for extensive parallel computing applications. Furthermore, the sequence in which operations of the first and second level are applied plays a significant role in enhancing overall performance and convergence. Properly ordering these operations is critical, as it impacts both the effectiveness and efficiency of the method.

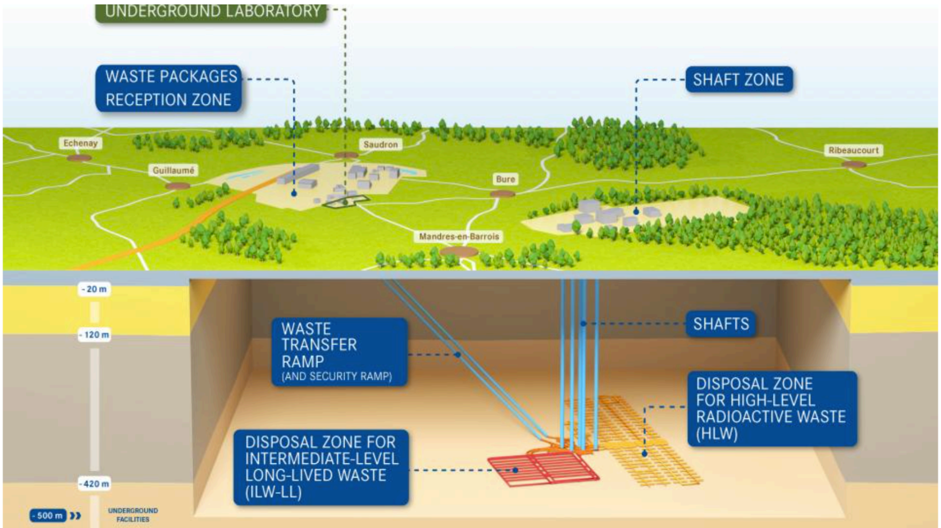


Figure 1: Radioactive waste storage repository

14:45

Héctor Vargas

High-Performance Large Eddy Simulations of Ammonia Hydrogen Industrial Gas Turbines.

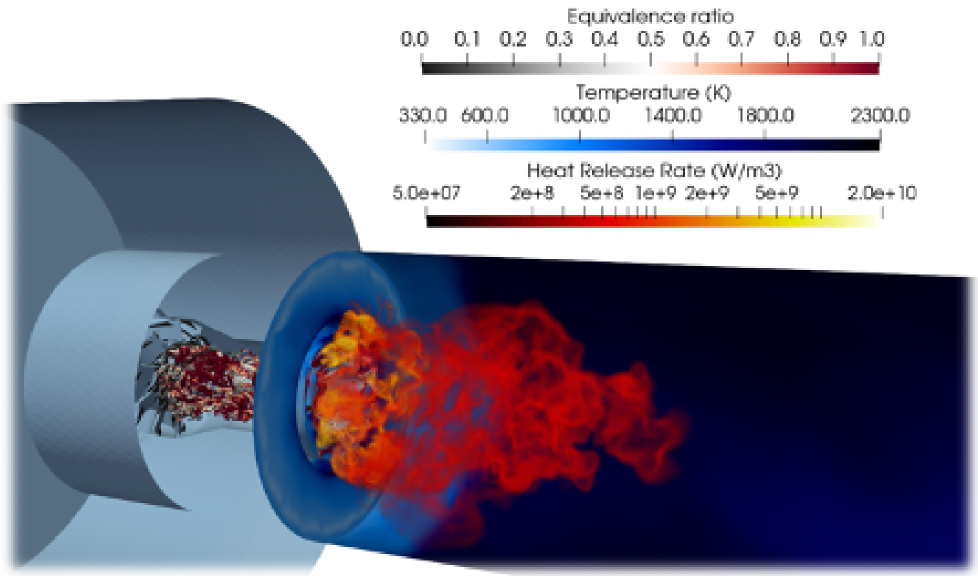
Decarbonization on off-grid off-shore oil platforms can be foreseen by adopting power generation fueled by NH3/H2 blends resulting from on-site ammonia cracking, thus benefiting from ammonia's logistical advantages (transportation and storage) and hydrogen's high reactivity. However, ammonia-based combustion is prone to high NOx production. High pollutant emissions are currently a showstopper for decarbonized fuels. Fortunately, recent studies have shown that novel staged-combustion burner technologies effectively mitigate pollutant emissions. The objective of the present work is thus to understand and accurately model flame dynamics and pollutant emissions of ammonia-hydrogen-fired gas turbines.

14:50

Mickaël Theot

Modeling and simulation of hydrogen-enriched gas turbines with exhaust gas recirculation for efficient integration of CO2 capture.

The main objective of TRANSITION is to pave the way for carbon-neutral energy production from natural gas-fired power plants using gas turbines (GT), by enabling a highly efficient CO2 capture and storage (CCS) process in the post-combustion phase. This will be achieved through the development of advanced hydrogen-assisted combustion technologies, capable of enabling stable burners operation with high rates of exhaust gas recirculation (EGR) leading to a high CO2 content in the exhaust gases sent to the CCS unit.



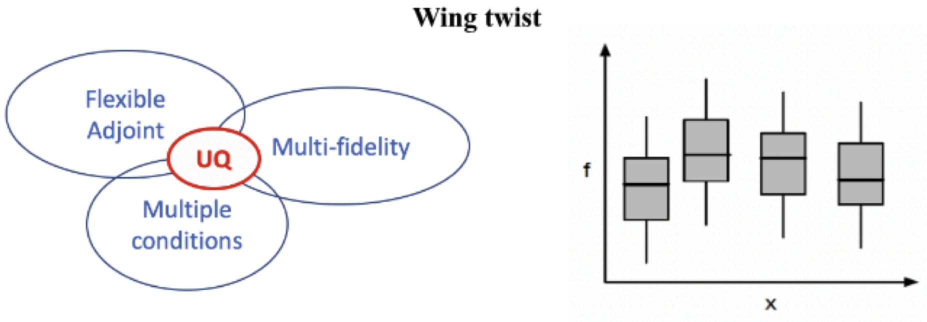
3'

14:55

Imane Fadli

Aerodynamic optimization with uncertain shape parameters

Accurately predicting wing twist poses a challenge in aircraft design, as the true value is revealed after initial flight test. This discrepancy is attributed to aero-structural coupling effects. The uncertainty surrounding mechanical and elastic properties increases the risk of aircraft performance issues, leading to costly rework and delays, especially in wings with innovative technologies like High Span or High Aspect Ratio Wings. The PhD aims to



showcase the feasibility of robust optimization methods with uncertain wing configurations.

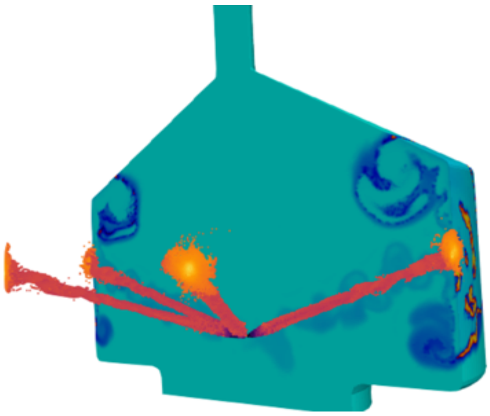
3'

15:00 (poster 15:20)

Arthur Berthelot

Development of methodologies and tools for non-polluting combustion.

The thesis project focuses on the modeling of Sustainable Alternative Fuels (SAFs) in order to highlight the behavior of fuel composition on combustion. The final step will be to predict breakthrough SAFs which are not used in actual combustion chambers yet, in order to determine operating conditions necessary to their use. In a first step, models used at CERFACS will be improved for turbulent combustion and two-phase injection. These models will then be used to run combustion chamber simulations and determine which SAF properties are important.



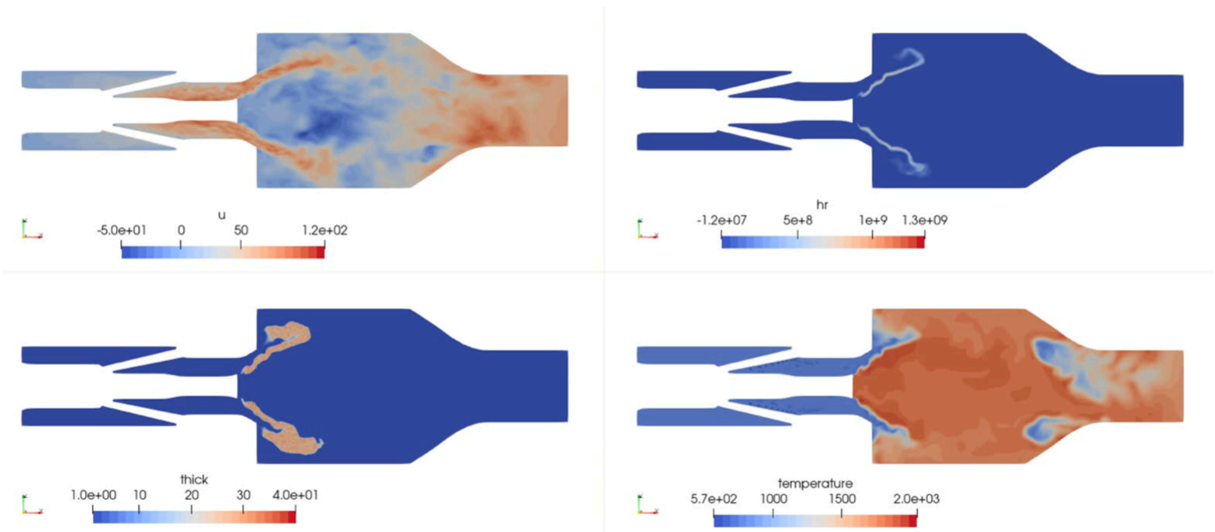
3'

15:05

Aldo Schioppa

Modelling of Flame Transfer Function of Lean H2 Turbulent Flames in a Real Ground-Based Turbine.

The phenomenon of unsteady flow oscillations, often denoted as combustion instability, has become a prevalent issue that impedes the advancement of lean-premixed combustors. These oscillations have the potential to attain significant amplitudes, thereby disrupting engine performance and, in severe instances, causing system failure due to excessive structural vibration and heat transfer within the chamber. The importance of predicting such instabilities, denominated thermo-acoustic instabilities, during the design phase, has been therefore risen to prominence. Advancements in Large Eddy Simulation (LES) techniques for reacting flows and Helmholtz solvers coupling revealed stunning potential in being able to duly predict acoustic modes in full annular combustors. Nonetheless, the modelling of the flame as an acoustic source, denoted Flame Transfer Function (FTF), is necessary to close the system of equations. In the framework of carbon-free fuels transition (EU project FLEX4H2), a modelling of the FTF of methane/hydrogen flames is hereby proposed in collaboration with ANSALDO Energia, envisioning operating conditions up to 100% H2, whereupon eminent thermo-diffusivity effects modulate the flame dynamics and heat release excursion. The numerical study shall be disclosed via the compressible reactive code AVBP coupled with AVSP or STORM, developed at CERFACS.



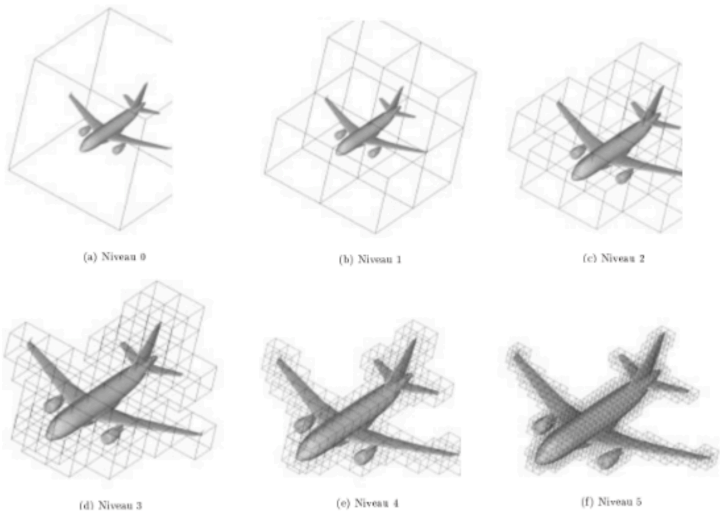
3'

15:10

Théo Briquet

Learning for predicting the rank of hierarchical matrices.

In the context of simulating the propagation of acoustic waves and aerodynamic calculations, the so-called Boundary Element Method (BEM) is commonly employed. This approach leads to the solution of dense complex linear systems of very large size, for which it may be impossible to form the complete matrix. Techniques of low-rank approximation of certain blocks can be used, leading to the definition of hierarchical matrix (H-matrix) based on a hierarchical partition of the computational domain as depicted below.



In collaboration with Airbus CR & T and INRIA, the objective is to predict the structure of the H-matrix and the ranks of the blocks corresponding to the interaction between row unknowns and column unknowns at any level of the hierarchy (tree).

3'

15:15

Solène Hoflack

Study of propagation of airborne viruses in closed environments using high fidelity simulations.

In the context of rising greenhouse gas emissions and the resulting climate change, the need to find renewable, low-carbon energy sources is growing ever more. A solution would be to use ammonia (NH3) as a hydrogen carrier and zero-carbon fuel. However, the lack of knowledge about ammonia combustion and associated NOx emissions limits its use. This issue is at the core of the ANR SIAC (Scientific Improvement on Ammonia Combustion) project coordinated by the University of Orléans, in which CERFACS is in charge of the "Large-scale modeling and simulation of turbulent ammonia/air flames" aspect. The aim of this research project is to provide high-fidelity numerical simulation tools for designing systems based on the use of ammonia as a fuel, either pure or weakly mixed with hydrogen (< 20% by volume).