Research Projects @ISAE-SUPAERO

Year 2025

Methods for data reduction of double cantilever beam (DCB) test

The use of laminated materials leading to laminated composite structures for aircraft structural parts aims at increasing the strength to mass ratio: more strength at lesser mass. Laminated composite structures, including adhesive bonded joints, induce the presence of interfaces, subjected to debonding or delamination. For the designer, the strength prediction of composite structures including their interface is then crucial. During the presizing stages, analytical or semi-analytical based methods are preferred since they allow for a quick decision process. The stake associated is then to control the fidelity of models wrt the physical reality. This Research Project (RP) is focused on the delamination in mode I through the use of the double cantilever test (DCB) specimen for composite materials and adhesively bonded joints. The DCB test allows for the measurement of the fracture toughness of the interface (interlaminar or adhesive) in mode I. The use of enriched data reduction method can be used to represent the interface as a cohesive zone model (CZM): J-integral based method. This method makes use of the measurement the local opening at crack tip, which is difficult to measure during experimental test.

In this context, the objective is to assess new data reduction scheme of DCB test to assess the CZM based on experimental measurement point easily to monitor and to simulate as the displacement at load point. The work is composed by: - literature review of data reduction scheme for DCB and analytical solutions for DCB tests (a detailed report will be provided) - adaptation of J-integral based method to replace the opening displacement at crack tip by the load point displacement as function of simplified hypotheses used in the various data reduction schemes in the literature - assessment of the sensitivity of the results of new data reduction schemes on the measurement capabilities - validation of relevant new data reduction schemes by numerical and eventually experimental tests

Performance and usage analysis of representative aircraft in the CS-23 category for identification of future aircraft configuration requirements

Description:
Electric and hybrid aircraft have been identified as a key lever for the reduction of aircraft emissions and costs. Due to the low technological readiness level, however, electric aircraft will likely be first adopted for applications in the General Aviation segment covered by the CS-23. In addition to that, aircraft with electrified propulsion are likely going to have smaller achievable ranges when compared to their thermal counterpart. Nonetheless, even if they are designed for great achievable ranges, thermal aircraft are rarely used on their maximum ranges. This makes an identification of the operational ranges of aircraft currently in the fleet a crucial point for defining future requirements for electric aircraft. Additionally, operational constraints and considerations will come into play when thinking about the design of new aircraft and evaluating the most optimal configurations. Metric such as the intensity of use or the lifetime will play a major role when comparing the life cycle impacts of future configurations.
The goal of this project is to identify, for each level of the CS-23, a representative mission as well as a representative aircraft. This will be achieved through an analysis and combination of several databases among which is the data from the OpenSky network. Particular attention to the range, frequency of use and lifetime of the aircraft will be required but additional relevant data might be identified to establish a concept of operation for future designs. An analysis of possible hybridization scheme based on the identified Top Level Aircraft Requirement will also be conducted. If time permits, a level of the CS-23 will be selected and a comparative analysis of the reference aircraft in that segment with a possible hybrid electric counterpart will be performed using the in-house OAD tool FAST-OAD-GA-HE.
The final deliverable will include:
- The identification of a reference representative aircraft for each level of the CS-23
- The identification, for those aircraft, of a commercially relevant range as well as a definition of its use during its lifetime
- For each of those aircraft: identify a promising use of electric propulsion
- If time permits assess potential gains of those configurations.

Keywords: Data Analysis, performance assessment, flight analysis, concept of operations, hybrid-electric aircraft, OAD

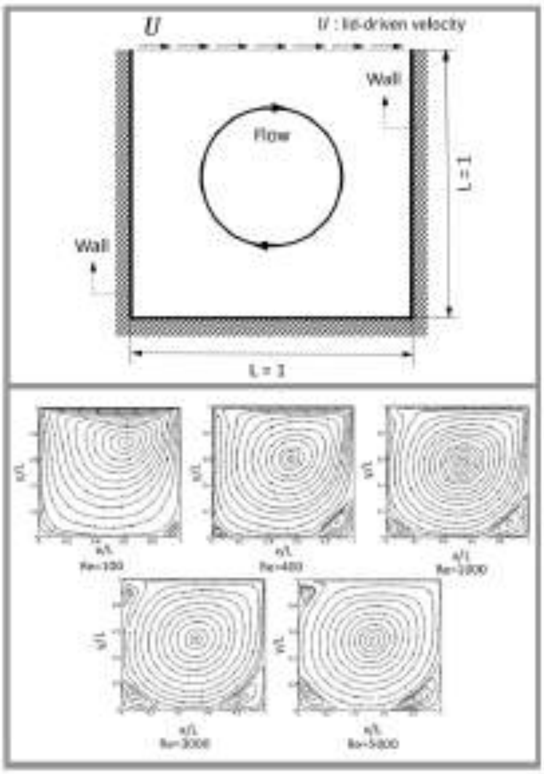
Integration of new aviation climate models into AeroMAPS - Exploration of simplified climate models

Description:
Aviation contributes to global warming through CO_2 emissions and non-CO_2 effects. The combustion of jet fuel at high altitudes directly impacts the climate by releasing CO_2 and H_2O, which are greenhouse gases, and sulphur dioxide and soot which have a short-term radiative effect. Aviation also has an indirect effect due to the short-lived formation of contrails cirrus, aerosol-cloud interactions, and alterations in O_3, CH_4 and stratospheric water vapour, all three greenhouse gases, caused by NO_x emissions. The various non-CO_2 effects are more complex to assess and therefore more uncertain (Lee et al., 2021).
In order to evaluate the current and future impacts of aviation on surface temperature change, three types of climate models can be used and applied to aviation.
- First, 4D climate models like Earth System Models (ESM) can be used for estimating the temperature increase. It allows accurate and local modelling, but at the cost of a complex usage and long calculation times. No 4D ESMs that include specific modelling of aviation non-CO_2 effects are available in open access. However, simplified frameworks including or dedicated to aviation, such as AirClim (Grewe et al., 2018), a linearisation of the E39/C climate-chemistry model (Hein et al., 2001), and APMT-IC (Mahashabde et al., 2011), are for instance used (Grewe et al., 2021, Dray et al., 2022).
- Second, the temperature can be obtained using 0D climate models, also called emulators, such as MAGICC (Meinshausen et al., 2011), CICERO-SCM (Sandstad et al., 2024), OSCAR (Gasser et al., 2017), or FaIR (Leach et al., 2021). These climate models integrate for instance carbon and methane cycle modelling and allow accurate estimations based on emissions or ERF. They are for instance used in (Terrenoire et al., 2019, Ivanovich et al., 2019, Boucher et al., 2021, Brazzola et al., 2022). However, they cannot model the geographical dependence of the radiative impact of some species, and do not always have specific modules for the non-CO_2 effects of aviation.
- Lastly, in a simplified approach, the temperature can be estimated by multiplying the Transient Climate Response to cumulative carbon Emissions (TCRE) coefficient by cumulative equivalent emissions. These equivalent emissions can be obtained from the ERF values using dedicated climate metrics, such as GWP* (Allen et al., 2018) or LWE (Allen et al., 2021). This method, for instance used in (Planès et al., 2021, Klöwer et al., 2021, Sacchi et al., 2023), is relatively simple to use and allows short calculation times. However, it may be limited in its ability to reproduce complex temperature profiles, particularly for sudden changes in emissions. The choice of the climate metric is crucial and conventional CO_2-equivalence metrics are for instance not appropriate in this context (Lynch et al., 2020).

For exploring aviation transition, the AeroMAPS open-source framework is developed at ISAE-SUPAERO (Planès et al., 2023). Using Python programming language, it allows performing simulation and multidisciplinary assessment of prospective scenarios for air transport. For instance, various strategies for mitigating aviation climate impacts can be compared on climate, environmental or economic point of view. Currently, AeroMAPS integrates two climate models: a simplified method based on GWP*, and another one based on the FaIR climate emulator.
The main objective of this project is to implement new climate models in AeroMAPS. This project targets the integration of other simple climate models (simplified LWE approach, other climate emulators such as MAGICC, OSCAR or CICERO-SCM in addition to FaIR), as well as the exploration of dedicated approaches developed for contrails in particular (CoCIP, aCCF). Several models will then be chosen for integration into AeroMAPS. The different models will be compared. Interaction with another similar research project, dedicated to the use of the OpenAirClim model (open-source version of AirClim), will be possible.

Splitting methods for incompressible Navier-Stokes equations.

Description :
In fluid mechanics, the Navier-Stokes equations play a central role and their resolution remains one of the greatest current scientific challenges. Many numerical methods exist to obtain approximate solutions. The best known of these consist of splitting the problem into several simpler sub-problems (splitting methods). Many splitting choices are thus possible. The objective of the work will be to study this type of approach to highlight the one that will be most suitable for coupling the Navier-Stokes equations modeling the fluid with equations modeling a structure (fluid structure interaction).
Problem statement and objectives :
We will consider the lid driven cavity in 2D [1] (see Figure) to validate the splitting numerical methods based on Finite Element Method (Getfem Library [2]).
The efficiency of the methods will be discussed for sequential and parallel implementations
Perequisites :
Scientific computing 1 (first semester) & Scientific computing 2 (second semester).
References :
[1] U. Ghia, K.N. Ghia & C.T.Shin, High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method, *Journal of Computational Physics*, 48 (1982), pp. 387-411.
[2] Getfem Library. https://getfem.org/



Bringing in human aspects in the design process

Description:

Context

In recent years, a transition from the automation-centric Industry 4.0 to a more human-centered Industry 5.0 emerged. Going beyond the efficiency and productivity achieved with Industry 4.0, respect for human values and contribution to society's vital needs came in the spotlight. Human-Cyber-Physical Systems (HCPSs) and their development are expected to provide features such as collaborative intelligence, adaptability, and resilience. Such features may be obtained by leveraging the consideration of the human (user/operator) early on in the design process and the use of capabilities such as integration, intelligence, and collaboration.

This project wants to contribute to the question on how to adapt current processes and tools to address the design, human in the loop integration, and multidisciplinary assessment of increasingly complex systems of systems with human and automation in the loop that characterise today's industrial context. Such a novel approach should facilitate and support the design of HCPSs with the objective of promoting a better integration of Systems of Systems (SoS), automation and human parts of such HCPSs from the earliest design phases.

Problem statement and objectives

This project proposes to build upon and contribute to a currently ongoing PhD project [Strobbe et al., 2024] through:

- refining the analysis of the state-of-the-art - many things are ongoing in this exciting field, and it would be good to have a complete comprehensive overview on what is happening
- propose solutions as to how to better incorporate human aspects in the design process
- work on the realisation of such improved processes

The exact topic on which this project will work on will depend on

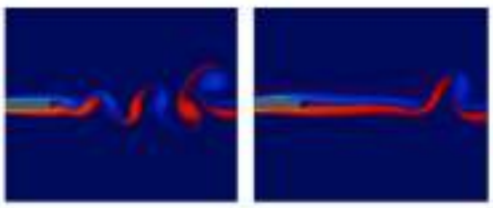
- where we are with the topics at the moment of the start of the research project
- your personal interest

It is therefore important to discuss in detail.

Optimal control of Navier-Stokes equations using sparse Polynomial Optimization

Description :
Fluid flow control is an important issue in a number of fields, including aeronautics, where the aim is to minimize aerodynamic drag and wake (vortices) around an aircraft wing.
The study of this type of model is a very active area of research, both theoretically and numerically, as well as experimentally (see Figures).
Problem statement and objectives :
One of the classic methods for introducing active control is to define an optimal control law from a discretized problem based on numerical methods involving fine spatial and temporal discretization (finite difference, finite element methods [2]...). In the end, we obtain a polynomial optimization problem of high dimension which cannot be solved by standard solvers.
The goal of this research project is to use sparse polynomial optimization [1] techniques which can tackle the dimensionality issue, in order to define new fluid flow control laws.
Perequisites :
Courses : Optimization, Scientific computing.
References :
[1] Magron, V., & Wang, J. (2023). Sparse polynomial optimization: theory and practice.

[2] Getfem Library. https://getfem.org.
Comments :
This project is proposed for 1 or 2 students motivated by the mathematical aspects of control.

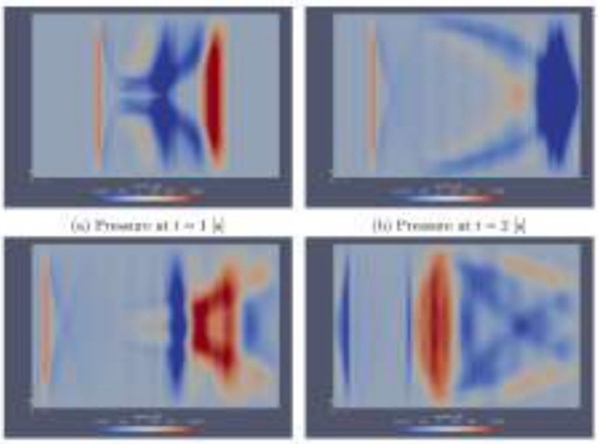


Highly parallelisable finite element models for wave propagation

Description:
In this research project we aim to explore new finite element discretization of wave propagation phenomena to speed up the computational efficiency while preserving the overall accuracy. These simulations may prove particularly useful in the context of architected material simulations, as those require overwhelming computational resources when modelled in fine details [1]. This topic lies in between mechanics (acoustic and elastic wave propagation) and applied mathematics. The idea is to develop a code to test a new discretisation scheme that is amenable to parallelisation.
Methodology
The idea is to reformulate the finite element problem so that each element is treated locally and then the information is transmitted through the boundary of each adjacent finite element via interconnections. This leads to a scheme where the finite element assembly is not at all standard. This strategy is a generalization of the ideas presented in [2, 3]. Figure 1 shows the pressure wave in a highly heterogeneous material using the method detailed in [2].
Challenges and learning objectives

- Understanding discontinuous Galerkin finite element methods;
- Use tools from system theory to develop a finite element scheme
- Implement the numerical scheme in open source finite element libraries like GetFEM FEniCS or Firedrake or implement the method from scratch;
- Develop a tailored linear algebra solver for the obtained system.

Candidate profile
This project can be performed by one or two students. The student is supposed to be familiar with finite element method, both its theoretical foundations and the implementation aspects. The interested candidates need to contact by email the supervisors. The curriculum vitae of the student should be attached to the email. Previous university transcript are also appreciated.
[1] T. Hirschler, R. Bouclier, P. Antolin, and A. Buffa. *Reduced order modeling based inexact FETI-DP solver for lattice structures*. International Journal for Numerical Methods in Engineering, 125(8):e7419, 2024. doi: 10.1002/nme.7419.
[2] Andrea Brugnoli, Ramy Rashad, Yi Zhang, and Stefano Stramigioli. *Finite element hybridization of port-Hamiltonian systems*. arXiv preprint arXiv:2302.06239, 2023.
[3] K. C. Park, J. A. González, Y. H. Park, S. J. Shin, J. G. Kim, K. K. Maute, C. Farhat, and C. A. Felippa. *Displacement-based partitioned equations of motion for structures: Formulation and proof-of-concept applications*. International Journal for Numerical Methods in Engineering, 124(22):5020–5046, 2023. doi: 10.1002/nme.7334



(a) Pressure at $t = 1$ [s]   (b) Pressure at $t = 2$ [s]
(c) Pressure at $t = 3$ [s]   (d) Pressure at $t = 4$ [s]

Figure 1: Wave propagation in an heterogeneous material

Predesign and proof of concept of a COntrolled MAGnetic INstallation for flow Experiments (COMAGINE)

Keywords: Predesign, magnetic devices, control, flow experiment

Context and objectives

Wind tunnel measurements are often compromised by spurious installation effects, which are induced by the very setup used to hold the object and measure forces, such as an aerodynamic balance. A common practice to account for these effects involves conducting two separate experiments: one with the setup in place but without the object, and another with the complete setup including the object (a in Fig. 1 in attached .pdf). The spurious effects are then estimated by subtracting the results of the first experiment from the second.

However, in practice, isolating these installation effects is much more complex due to strong interactions between the setup and the object. These interactions, such as blockage and vortex generation at connection points, can significantly distort the measurements. As a result, the total force in a typical wind tunnel experiment can be expressed as:

TOTAL FORCE = ISOLATED OBJECT + ISOLATED SETUP + INTERACTIONS

One way to eliminate these interactions is to physically remove the setup, but the object must still be securely held in place, and forces acting on it need to be measured. Achieving this without a physical mounting device poses a significant challenge. The goal of the COMAGINE project is to develop an innovative experimental setup where the object is suspended in the flow via magnetic levitation (b in Fig. 1 in attached .pdf), eliminating the need for a physical mounting structure and minimizing spurious installation effects. This can be achieved through the use of magnets installed on the wind tunnel walls, along with a controller to maintain the object's position regardless of flow disturbances or unsteady forces. Moreover, the controller itself can serve as a measurement tool, with its real-time adjustments providing data on the unsteady aerodynamic forces acting on the object. This approach has the potential to offer more accurate measurements by directly eliminating interference from mounting devices.

The COMAGINE research project will explore the feasibility of this novel setup, starting with the design phase and, if successful, moving toward a proof of concept.

Roadmap

This research project aims to investigate the COMAGINE concept for the first time. As such, the roadmap primarily focuses on an exploratory phase at a low Technology Readiness Level (TRL). The main goal of the project is to determine whether this idea is both feasible and promising for application in standard wind tunnel experiments, such as those available at ISAE-SUPAERO. The general roadmap is structured as follows:

1. **Literature Review**: Conduct a comprehensive review of existing research on similar ideas involving levitation in flow experiments. This includes an in-depth exploration of magnetic levitation systems, covering the physical principles, relevant equations, and the hardware/software technologies that could support the concept.

2. **Low-Order System Modeling**: Create a simplified mathematical model of the system to estimate real-time power consumption and sphere positioning. This stage includes testing the system under perturbations and developing an adaptive control mechanism, potentially utilizing a simple PID controller or a more advanced AI-based approach.

3. **Preliminary Experiment Design**: Develop a basic table experiment where a sphere is levitated using a controlled magnetic field to compensate its weight. This stage involves establishing key system parameters, such as mass, power requirements, and other relevant physical properties, based on the previously developed low-order model. Cost estimation is also critical, and if the design proves mature and cost-effective, a prototype could be built by students during the project.

4. **Simulations and Validation**: Depending on the progress and maturity of the solution, simulations could be conducted using *StarCCM+*, which supports external control inputs. The aim is to evaluate the proposed

solution by simulating the flow around an airfoil in a simplified numerical wind tunnel, with a real-time controller interacting with a modeled magnetic field.

PACIFIC - Passive Acoustic Control for Improved Fuselage Interaction with Propellers

Description:
The noise of propellers interacting with the fuselage/wing, from drones to light aviation, is an obstacle to the deployment of this type of aircraft. This is important as history showed us that not considering acoustic interaction effects can lead to unbearable noise in the cabin for example [1] and require a posteriori architecture changes that are extremely costly. Rapid analytical or semi-analytical methods permitting to estimate during the design process of the flying object its acoustic footprint on the ground are necessary.
In the low Reynolds number regime, studies have been conducted on rotor-strut interaction noise at ISAE-SUPAERO [1,2,3]. The presence of a strut nearby produces additional tonal noise at harmonics of the blade passing frequency. This noise depends on the geometry of the strut, its position relative to the rotor disc plane, and the distance from this plane. This extra noise source component can come from several noise source mechanisms, one of them being the unsteady loading on the strut. This noise mechanism is the dominant one in one case studied experimentally, numerically, and analytically at the laboratory [2], and whose far field acoustic spectrum is given in Figure 1.

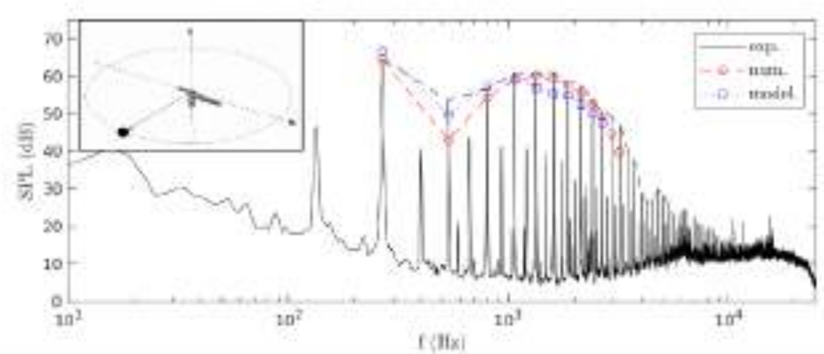


Figure 1: Experimental, numerical and analytical modelling of the interaction noise of a two-bladed rotor with a cylindrical strut [2]

It would thus seem interesting to mitigate this noise source by working on the strut. Preliminary tests have been performed with the use of a 3-D printed porous material (Figure 2, left) and small noise mitigation have been performed on the harmonics of the BPF. Moreover, for the students to have an easy access to experimental testing, a small test bench has been built (Figure 2, right) and will be used in order to test rapidly the mitigation technologies the student will build. Other geometries like airfoils could also be studied to go towards distributed electric propulsion architectures.

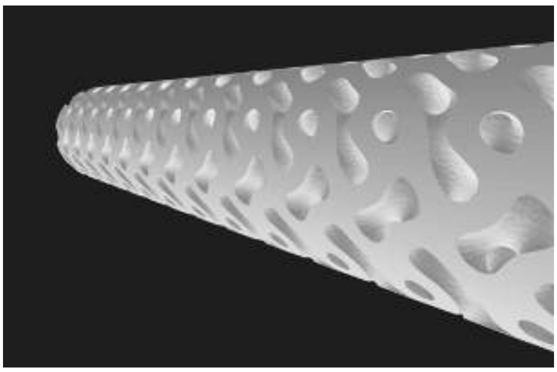


Figure 2: Left: Example of a porous material tested to try to mitigate the noise of a propeller interacting with a strut. Right: Test Bench that will be used during the project to test passive technologies

Program of internship

The aim of this internship is to do a bibliographic study to know what passive acoustic control technology would be the most beneficial to mitigate this noise source. A modelling part will be done to tune the material towards the acoustic source we are targeting. Then, manufacturing of this material (preferably using stereolithography 3-D printer but other technics could be deployed) will be conducted and exeperimental tests on the small test bench presented before will be carried out. The best candidates will be tested later in the anechoic room of ISAE-SUPAERO to have a precise quantification of the acoustic gains reached**.**

Bibliography

[1] **R. Gojon**, H. Parisot-Dupuis, B. Mellot, and **T. Jardin**. *Aeroacoustic radiation of low Reynolds number rotors in interaction with beams.* The Journal of the Acoustical Society of America, 154(2),2023.

[2] E. Vella, **R. Gojon**, H. Parisot-Dupuis, N. Doué, **T. Jardin**, and M. Roger. *Mutual Interaction Noise in Rotor-Beam Configuration*. 30th AIAA/CEAS Aeroacoustics Conference, 2024.

[3] **R. Gojon**, N. Doué, H. Parisot-Dupuis, B. Mellot, **T. Jardin**. *Aeroacoustic radiation of a low Reynolds number two-bladed rotor in interaction with a cylindrical beam.* 28th AIAA/CEAS Aeroacoustics Conference, 2022.

ALICE - Low Power and Wide Area Communication

Description:
Low Power and Wide Area Communication for Precision Agriculture in Life Support Systems
ALICE (Artificial Intelligence for Life In spaCE) project

Key words: Bioregenerative Life Support Systems, Precision Agriculture, ALICE project, Robotics, Data transmission, Radio Communication, Low Power Wide Area Communication Pro- tocol, LoRa, Data Collection, Robotics, Python, Git.

1 Context
Now that humanity is able to go to other planets, the next challenge is to send human there, and to be able to settle there permanently. Since supplying the future settlers from Earth would be far too expensive, if not impossible, one of the major problems in achiev- ing space exploration is the lack of resources and favorable local conditions. Moreover, according to the latest work of the Intergovernmental Panel on Climate Change (IPCC), sustaining life on our own planet also seems to be a challenge. Contexts that support life will be more difficult to access on Earth, mainly because of climate change itself, and maybe in the shorter term as a consequence of the policies to fight it, and the depletion of resources. Finally, on Earth or in space, it is necessary to find solutions to ensure the sustainability of life despite the lack of resources and unsuitable environments.
The research in *bioregenerative life support systems (BLSS)* [13, 1] aims at allowing a long term settlement of the human being in such environments. This is why ESA is interested in the implementation of BLSS: *"For more than 30 years, the European Space Agency (i.e. ESA) is active in the field of regenerative life support system. MELiSSA (Micro-Ecological Life Support System Alternative) is the European project of circular life support system. It was established to gain knowledge on regenerative system, aiming to the highest degree of autonomy and conse- quently to produce food, water and oxygen from mission wastes."*1. Within the framework of the MELiSSA project, design studies for greenhouses [11, 2, 14] and prototypes have been developed (e.g. for cultivation of tuberous plants [6] in the ESA Project "Precursor of Food Production Unit"), and many research works have been carried out (e.g. on hydroponic systems [4] or potatoes in controlled environments [5]).
The SpaceShip FR project of the CNES (*Centre National d'Etudes Spatiales*), that started in 2019 in Toulouse FRANCE, plans to build a lunar or martian type base in order to demonstrate the solutions implemented in various fields, such as health, energy, robotics, digital, plant cultivation and recycling. Regarding the issue of nutrition, the SpaceShip FR project considers hydroponic greenhouse, as well as circular aquaponic system whose fish can be used to diversify the astronauts' daily nutrition and create fertilizers from their waste.
Many experiments have already been carried out in space, one of the most recent being the production of chili peppers in the ISS by NASA three years ago, which of course also conducts a lot of research on systems growing plants for food production [3]. It was also the first fruit in space and the longest experimentation with plants. For his part, the ESA astronaut Thomas Pesquet has been named Food and Agriculture Organization (FAO) Goodwill Ambassador in April 2021, on the occasion of the International Human Space Flight Day. Finally, a french start-up has succeeded in growing lettuce in lunar regolith. Although this experiment was not in space, it represents a big step towards space agriculture.
2 ALICE Project
As explained and highlighted in [7, 12, 8], space crop production needs for advanced au- tomation, robotics, modeling, and machine learning. With the support of the Innovspace (Fablab of ISAE-SUPAERO), the ALICE (AI for Life In spaCE) project also contributes to the research domain of Precision Agriculture in Life Suport Systems (LSS) with the gen- eral goal of using Artificial Intelligence (AI) for maximizing production and minimizing ressource consumption. In this context, the ALICE project focuses on 4 main interrelated issues, namely the optimization of:

1. the *design* of plant cultivationf systems using Multi-Disciplinary Analysis and Opti- mization (MDAO) frameworks taking into account various models and criteria,
2. the plant and environment state *estimation* using Machine Learning and Computer Vision algorithms to derive informed decisions from data,
3. the sequential *decisions* using Planning and (Offline) Reinforcement Learning algo- rithms to compute efficient and economical autonomous cultivation strategies,
4. the *data transmission* between systems using low power, wide area communication protocol (e.g. LoRa).

The related studies paves the way towards autonomous plant growing systems capable of analyzing and even reacting to the growing process in order to quickly obtain healthy plants while optimally use space and resources (nutrient, water, energy, etc.). Twenty students are taking part or have taken part in this project since 2019.
The ALICE project also sets up test beds to collect plant cultivation system data useful for the 4 main issues, and to ensure that the tools developed work in practice. One of them is a robotic hydroponic system (see Figure 1) developped

and improved by previous students. This robot controls the intensity and the frequency of the light, as suggested by [10, 9], the flow of the water and the nutrients level, measures the temperature, the pH of the water, and with its robotic arm fixed on a rail, it is able to take pictures of each plant and to move them.
Another test bed of the ALICE project is the Farmbot Genesis XL2 (see Figure 2), an open source Cartesian coordinate robot farming machine that has been set up next to the ISAE restaurant. This robot can plant seeds, water plants, remove weeds, take pictures, and measure the soil moisture. The hydroponic system as well as the Farmbot may be used for this research project addressing the *fourth main issue*, namely *the optimization of the data transmission between systems using low power, wide area communication protocol (LoRa)*.

3 Research Project objectives

The objective of the presented research project is to use *low power, wide area communication protocol* (LoRa) for data transmission for plant cultivation systems.

- The first step in any research project is to begin a literature review of the subject. For this project, it will concern plant cultivation systems, precision agriculture, low power, wide area communication protocol, and LoRa.
- In parallel, it will be necessary to learn how to use the LoRa modules, as well as the test beds of the ALICE project (the robotic hydroponic system, the Farmbot, weather stations, etc.) listing accessible data and possible actions of the system on the envi- ronment, to review the data to be transmitted and the priorities in the communica- tions considered.
- Once the state of the art has been sufficiently analyzed, LoRa nextworks will be proposed, implemented and evaluated.

Low-cost Models for rotor noise in crossflow

Description:

Background

The noise of isolated rotors is strongly influenced by crossflow. Indeed, the presence of a uniform flow not aligned with the rotor axis generates unsteady loadings on the blades due to the change in effective velocity and effective angle of attack along rotation, hence additional aeroacoustic noise. Rapid analytical or semi-analytical methods allowing to quickly estimate extra noise sources from crossflow induced unsteady loadings are necessary.

A first internship has already been conducted where 3D URANS simulations were performed for an isolated rotor with and without crossflow. A first low-order, quasi-steady model relying on BEMT and simple geometrical considerations was developed and validated against URANS simulations. Figure 1 shows 3D URANS simulations of a rotor with and without lateral crossflow. The appearance of an asymmetry of the wake and of the flow pattern on the blades is visible when adding a crossflow.

Isosurfaces of Q-criterion for an isolated rotor without (left) and with (right) crossflow

However, this model is only capable of predicting the unsteady loading on the blades when the crossflow is quite low, i.e. notably without any leading-edge detachment on the blade. To go beyond this limitation, 2-D simulation replicating the flow on a cylindrical slice will be conducted. In this reference frame, the crossflow can be modelled either by a sinusoidal incoming velocity or by an oscillating motion of the profile (representing the blade cut at a certain radial location).

These simulations will rely on two distinct approaches. First, a modified vortex lattice method (VLM) accounting for leading edge separation will be used. Second, similar configurations will be computed using URANS simulations and results from both VLM and URANS will be compared. These approaches will help identify the unsteady mechanisms, with or without separation, that are not tractable with present quasi-steady approaches.

Program of internship

The aim of this internship is to do a bibliographic study to understand the physics behind codes VLM and URANS that will be used for the 2-D simulations during the internship. Then, 2-D simulations will be conducted in order to build a database. This database will then permit to construct a low-cost model reproducing the unsteady loading on the blades of a rotor with any crossflow intensity.

DEEP learning for video-based Reconstruction and Dynamic eQuations (DeepRDQ)

Description:
DEEP learning for video-based Reconstruction and Dynamic eQuations (DeepRDQ)
Supervisors: Michael Bauerheim (ISAE-Supaero), Sébastien Prothin (ISAE-Supaero), Yannick Bury (ISAE-SUPAERO)
Keywords: Deep Learning, surface reconstruction, dynamic equations, videos, experiments

Context and objectives
Many experimental setups require the reconstruction of deformed surfaces or quantities of interest (e.g., flow fields) in 3D space over time. In fluid mechanics, this task is particularly challenging due to the limited access to spatial and temporal data during experiments. Typically, complete fields are obtained using Particle Image Velocimetry (PIV) for velocity measurements, while surface deformations are often captured using Digital Image Correlation (DIC). However, traditional PIV and DIC algorithms have limitations, primarily due to the inherent noise from experimental setups, and constraints such as lighting and camera positioning among many others. Furthermore, certain quantities of interest, such as pressure fields, cannot be captured directly and are instead measured locally via probes.
This project aims to explore how modern deep learning frameworks can address these challenges, enabling the reconstruction of surface or flow quantities from time-resolved videos. Recent advancements, such as Generative Adversarial Networks (GANs) and Instant Neural Graphics Primitives (Instant NGP, see Fig. 1 bottom), have demonstrated the ability to reconstruct multi-resolution objects from simple video data (e.g., Instant NGP).
Additionally, deep learning techniques have shown promising results in extracting underlying dynamic governing equations from time-resolved data. For instance, the SINDy (Sparse Identification of Nonlinear Dynamics) method, combined with autoencoders, has been employed at ISAE-SUPAERO (see Fig. 1 in attached file) to reconstruct both the state variables and dynamic equations governing the oscillations of a simple pendulum, captured using a smartphone camera (e.g., SINDy Method).
As a consequence, the primary goals of this project are twofold: (1) to investigate the use of deep learning approaches for reconstructing surface deformations and flow quantities in space and time, and (2) to extract state variables and their corresponding dynamic equations for low-order modeling and system analysis. By leveraging these novel techniques, we aim to improve the accuracy and resolution of reconstructions in experimental fluid mechanics, while also gaining deeper insights into the underlying dynamics of the system.

Roadmap
This research project aims to explore the **DeepRDQ** concept for the first time, focusing on an initial exploratory phase at a low Technology Readiness Level (TRL). The primary objective is to assess whether the approach is both accurate and feasible for simple systems, such as a damped pendulum, and potentially scalable to more complex systems commonly encountered in fluid mechanics or fluid-structure interactions. The project's roadmap is structured as follows:

1. **Literature Review**: A thorough review of existing research on deep learning techniques for surface reconstruction and flow quantity estimation will be conducted. This includes exploring auto-encoder strategies, GANs, Instant-NGP, and other relevant methods. Additionally, a review of data-driven system identification approaches is necessary to understand the state-of-the-art and identify gaps that this project could address.
2. **Hands-on Benchmarking**: The most promising reconstruction techniques identified in step 1 will be benchmarked against known datasets or simple datasets produced by the students them-selves (e.g. a simple fixed object with multiple details). A potential example includes the reconstruction of CAD models from videos taken by one or more smartphone cameras (similar to fig. 1 bottom). The goal is to develop a Python notebook that allows users to input video data and outputs relevant for surface reconstructions or predictions of flow quantities.
3. **Proof of Concept on Simplified Synthetic Systems**: Next, the strategies evaluated in step 2 will be tested on simplified physical systems (similar to Fig. 1 top, attached file). For example, synthetic data from pendulum systems or time-varying structural problems (e.g., systems with damping or multi-mass-spring models) will be generated. These datasets will enable space-time reconstruction, which will then be used to extract the governing dynamic equations of the system.
4. **Proof of Concept on Real-World Systems**: Building on the insights from step 3, a simplified real-world test bench will be designed. The objective is to develop a complete setup where deep learning techniques can be

applied to reconstruct and extract governing equations from a real-world physical system. This test bench could serve as a future platform for demonstrating the application of AI to physical systems at ISAE-SUPAERO.

Student

We are looking for highly motivated students (maximum of two) for this research project. Due to its exploratory nature and low Technology Readiness Level (TRL), this project is uniquely challenging. It involves multi-physics, coding and AI skills. These can be learnt during the research project but requires a high motivation and will to learn. It also demands a high degree of independence, critical thinking, and decision-making. Students will need to be comfortable with performing preliminary analyses, including early-stage mathematical modeling, benchmarking existing solutions and should have the skills and enthusiasm to design and construct test benches to validate their models. Python skills would be a bonus, since the main language used in AI today.

Formula stUdent uSing artificial Intelligence for OptimizatioN (FUSION)

Description:
Formula stUdent uSing artificial Intelligence for OptimizatioN (FUSION)
Supervisors: Michael Bauerheim, Valérie Ferrand, Marc Montagnac
Keywords: Deep Learning, CFD, experiment, optimization, formula student
Context and objectives
This project takes its roots in the Formula Student club at ISAE-SUPAERO (https://ae-isae-supaero.fr/clubs/clubs-ae), which focuses on the design, manufacturing, and real-world competitive testing of an autonomous electric formula vehicle. While the club itself targets participation in international competitions, this project addresses the academic research topics associated with it, involving collaboration between researchers and PhD candidates from the DAEP department. Specifically, the project focuses on the aerodynamics of a simplified, downscaled version of the Formula Student vehicle.
The primary goal of FUSION (Formula stUdent uSing aI for OptimizatioN) is to develop and evaluate a new optimization framework for aerodynamic design by leveraging both Computational Fluid Dynamics (CFD) and experimental data, with the integration of Artificial Intelligence (AI). This year's focus is on optimizing the design of the front and rear wings of the vehicle. The rest of the race car is considered as fixed during the study.
Traditionally, the aerodynamic design pipeline follows a sequential process (illustrated in black in attached figure), where 2D parameterized CFD simulations explore the design space to achieve optimal results. A few 3D CFD simulations are then run for validation before the components are manufactured and experimentally tested in wind tunnels. However, FUSION proposes a new paradigm (illustrated in blue in attached figure), where AI, particularly deep learning techniques, enables the simultaneous integration of CFD and experimental data. This approach allows for the creation of fast and accurate surrogate models of the aerodynamic behavior, which can be efficiently deployed in gradient-based optimization algorithms.
The primary outcome of this research project is the development of this AI-driven optimization pipeline. This framework will not only enhance future designs for the Formula Student club at ISAE-SUPAERO, but also contribute to academic research at DAEP, providing valuable tools for aerodynamics and design optimization.

Roadmap
This research project focuses on the exploration and development of the **FUSION** concept, an AI-based optimization pipeline for aerodynamic design in Formula Student vehicles. Building on previous works conducted at the DAEP department, which utilized CFD data and genetic algorithms for optimization, as well as recent advancements in AI-driven optimization [1, 2, 3], this project aims to integrate these approaches into a unified framework. The roadmap for the project is outlined as follows:

1. **Literature Review**: A comprehensive review of current research on deep learning techniques for surrogate modeling and optimization will be undertaken. The review will cover various neural network architectures (such as MLP, Graph Networks, Implicit Neural Representations) as well as techniques for combining simulation and experimental data, known as data fusion). Uncertainty estimation and multi-fidelity methods, with a focus on transfer learning and active learning, will also be considered to enhance the robustness of the approach.
2. **2D Simulations**: Initial simulations will be performed on downscaled 2D front and rear wings with parameterized shapes, using Reynolds-Averaged Navier-Stokes (RANS) models. Particular attention will be given to mesh convergence and CFD quality. Existing setups from the Formula Student club can be leveraged to accelerate this process.
3. **Developing the FUSION Code**: In parallel with the simulation work, the **FUSION** code will be developed in Python. The framework will be designed to accommodate various meshing tools, CFD solvers, and optimization algorithms as backends. Several foundational components already exist, either in Python or Matlab, which will be adapted and integrated into the pipeline.
4. **Optimization Using FUSION**: Once the 2D simulations and FUSION code development are completed, the pipeline will be employed to automatically generate a dataset from the 2D simulations. This dataset will be used to train a deep neural network, which can then serve as a fast surrogate model for aerodynamic evaluations. The trained network will support optimization processes using either genetic algorithms (e.g., NSGA-II) or gradient-based optimization methods.
5. **3D Simulations and Experimental Validation**: Depending on the maturity of the solution, 3D simulations will be conducted on the full vehicle with the optimized front and rear wings. The final configurations can be 3D printed, allowing for experimental validation in a wind tunnel to confirm the performance of the downscaled

optimized vehicle/component. This step will provide a critical link between simulation-based optimization and real-world aerodynamic performance.

Student

We are looking for highly motivated students (maximum of two) for this research project. Due to its exploratory nature, this project is uniquely challenging. It involves multi-physics, coding (python) and AI skills. These can be learnt during the research project but requires a high motivation and will to learn. It also demands a high degree of independence, critical thinking, and decision-making. Students will need to be comfortable with performing preliminary analyses, including early stage predesign modeling, benchmarking existing solutions and should have the skills and enthusiasm to design and construct experimental mockups. Python skills would be a bonus, since the main language used in AI today.

Improving paragliding back impact protection

Description:
Back injuries are among the most common paragliding related injuries. Appropriate back protection is thus an essential safety item in paragliding harnesses. Traditionally this back protection has been either foam or airbag based, both technologies having their corresponding advantages and drawbacks. Recent years have brought new developments in paragliding back impact protection with the arrival of the Neo-Koroyd based protectors. These use polymer tubes to create cubes that crumple under the impact, thus dissipating energy. This behavior allows for significantly thinner back impact protection, making the harnesses less bulky and improving their aerodynamic efficiency. There has been however some controversy whether the Koroyd behavior, which has maximal energy absorption, but in this process leads to high acceleration jerk is actually able to reduce the injuries following a back impact. This project seeks to explore ways to further improve this type of back impact protection through multiple leads : selective stiffening of the microtubes, designing more complex architected microstructure that further improve injury prevention, etc.

To achieve this, the project can address (depending on time and interest) following:

1. Litterature review of back impact injury prevention, notably based on medical studies on fighter jets ejector seats
2. Definition of the most relevant design criteria for a paragliding back protector based on the findings of the literature review
3. Numerical exploration of the back protection behavior and possible improvements through selective stiffening of certain areas
4. Design and optimization of an architected material that best achieves the previously defined design criteria
5. 3D printing of the architected material
6. Impact testing of Koroyd and of alternative or 3D printed material to compare real and expected performance

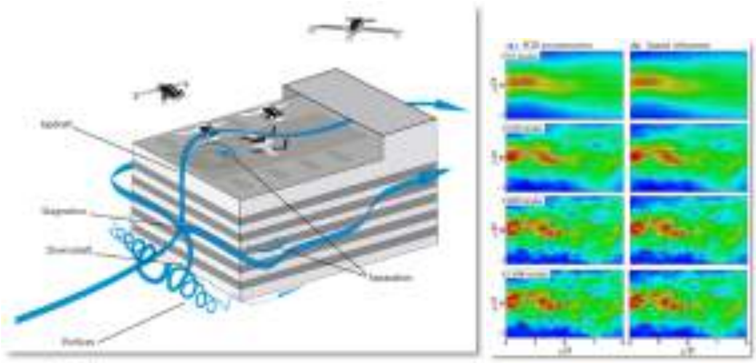
Exergy-based aerodynamic analysis methodology for low-fidelity aircraft design

Description:
The aerodynamic analysis based on the exergy method provides a drag breakdown assessment which can represent a particularly suitable design tool for innovative aircraft architectures. Until now, the existing exergy formulations are appropriate for mid-fidelity data such as RANS computation results as well as WTT data. However, the conceptual and preliminary aircraft design phases in the industry massively uses low-fidelity modeling such as panel method solver with viscous coupling since they require far less computing resources than mid-fidelity RANS simulations. Then, the objective of this work is to introduce a new exergy-based aerodynamic analysis formulation suited to low-fidelity data suited for the preliminary design process.
Emphasis was placed on profile drag determination through the link between the exergy and anergy terms and the boundary layer integral quantities. This low-fidelity exergy-based aerodynamic analysis approach was validated by comparison against existing higher-fidelity exergy formulations in the flat plate test case.

Simulation of disturbed areological environments

Description:
Key words: Disturbed environment, CFD, Urban flow, code development, numerical simulation
Description:
Simulating aerology in urban areas is a key strategic issue for understanding and controlling the environment in which drones and future flying taxis will operate. Cities, with their complex structures and airflow disturbances caused by buildings, present specific challenges that aerological modeling helps anticipate.
Urban environments are marked by dynamic interactions between winds and structures such as buildings, bridges, and roads. These elements create complex aerodynamic phenomena, such as vortices, gusts, and areas of weak or highly turbulent winds. Simulating these conditions allows for the creation of precise maps of airflows in cities, which are essential for determining the optimal and safest routes for drones and flying taxis.
One of the main benefits of aerological simulation is improving the safety of urban flights. By predicting high-risk areas where turbulence and strong winds could destabilize aerial vehicles, it becomes possible to develop more efficient and adapted flight algorithms. These simulations also help design safer flight paths, taking into account changing conditions and avoiding certain areas during extreme weather.
Urban aerological modeling helps better plan drone and flying taxi routes based on wind conditions. By anticipating areas where air currents are favorable, it becomes possible to reduce the energy consumption of these vehicles, thereby improving their autonomy and efficiency. This is especially important for flying taxis, which will need to cover longer distances in cities while minimizing their environmental impact.
Aerological simulations also help limit the noise and environmental impact generated by drones and flying taxis. By identifying flight paths where engine use can be reduced or predicting areas where noise will propagate less easily (such as between buildings or above certain infrastructures), it becomes possible to minimize the impact of flights on residents' daily lives.
With the rise of urban air mobility, managing air traffic in cities will become a crucial challenge. Aerological simulation allows for the design of airspace management systems tailored to dense urban environments. By better understanding airflows, it will be possible to create safe and efficient "air corridors" where drones and flying taxis can travel without interfering with one another or disturbing the environment on the ground.
Finally, simulating urban aerological conditions is essential for testing and developing the technologies that will equip the drones and flying taxis of the future. These simulations allow for the testing of propulsion, navigation, and sensor systems in realistic conditions before they are put into service, ensuring their robustness and reliability in a complex urban environment.
Simulating aerology in urban areas is a crucial step in preparing for the future of air mobility, whether for drones delivering packages or flying taxis transporting passengers. It helps better understand the aerodynamic challenges of urban environments, enhances flight safety, optimizes energy efficiency, and designs an urban air traffic management framework that meets future requirements for sustainable and intelligent mobility.
In this project, students will be asked to work on two aspects:

- Research the most suitable numerical methods for simulating urban environments.
- Consider a format, or an example of an "academic" city configuration for the simulation.
- Generate spatio-temporal wind data, representative of real flows (atmospheric boundary layer).
- Perform CFD simulations of an offshore wind turbine under StarCCM+ to extract wake data.
- Consider how to process this raw data (very heavy) to use it in a drone simulator.



Stealth bio-inspired, flapping wing propulsion

Bio-locomotion has fostered great attention these past decades as a potential solution for aerial and under-water propulsion. In addition to providing enhanced aero-/hydrodynamic performance, bio-inspired propulsion like flapping wings and fins could lead to stealth systems with low noise emission. A few studies have recently analyzed the noise from bio-locomotion under the prism of zoology and the evolution of natural species (Seo, Hedrick & Mittal, 2019). However, the role of design parameters on the aeroacoustic footprint of bio-inspired propulsion systems remains virtually unexplored, which is a crucial step towards the design of stealth systems.
Within this project, we thus aim to evaluate the role of design parameters on the noise generated by an airfoil undergoing both pitching and plunging (flapping) motions. The work relies on :

1. (1) Conducting direct numerical simulations of the flow past the pitching and plunging airfoil using StarCCM+ software
2. (2) Apply acoustic analogy to predict the farfield noise of different cases
3. (3) Identify parameters that promote lift, efficiency and low noise emission
4. (4) Analyze noise sources using force decomposition methods

HAICO: Human AI Collaboration in video game

Description:
Supported by the **CASAC chair**[1] and by the PANACHE (AID) project, **HAICO** aims to explore human-AI collaboration in cases where the artificial agent can learn to collaborate with the human agent. The idea is to exploit markers related to the human actions intention, and at the same time, to provide means to the artificial agent to express its own intention of action. The project focuses on using the *Overcooked* game environment to study this Human-AI Collaboration. More specifically, the student research project linked to HAICO aims to evaluate and compare different Deep Reinforcement Learning (DRL) techniques that could be applied to this collaboration.
A level of this game has been developed at ISAE-SUPAERO using the Unity game engine, and an artificial agent has already been trained on certain tasks. The student will built her/his approach based on that.

Main Objectives:

- Review of reinforcement learning techniques suited to human-AI collaboration.
- Integration of various methodologies into the *Overcooked* platform.
- Training of agents using different approaches (behavioral cloning, fictitious co-player, etc.).
- Participation in the recruitment of human participants for experiments.
- Analysis of post-experiment data.

Student's profile
MSc students with programming skills (Python, C++) and a background in AI (RL) is suitable. The candidate will study/learn machine learning techniques and theory related to Markov Decision Process and Reinforcement Learning.

Autonomous Exploration Rover Electronics

Description:
The "Autonomous Exploration Rover" project aims to design and create a functional prototype of a rover intended for exploring challenging terrains. This platform is intended to serve as a tool for teaching and research. Ultimately, this prototype should be fully autonomous in its ability to navigate and move in an unmarked environment and should be able to be enhanced with additional modules.

The rover we propose to build will be very similar to the open source rover from JPL. The mechanical conception of the rover has been proposed as a student project (FISA2) and should be available at the beginning of 2025. The objective of this MAE project is to first study the state-of-the-art of the electronic design of existing rover, then to propose and realize the electronic part and to make the rover controllable.

What will be done during this project?

- A review of Electronics architecture of space rovers
- The development and realization of the electronics of the rover
- The development of a dedicated controller to make the entire robot move

Thermodynamic/Statistical-Mechanical Approach to Evaluating Sustainability of Aircraft Propulsive Systems

The contemporary aeronautical industry, while demonstrating superb prowess in a variety of domains (gas-turbine propulsion, vehicle control, operational strategies, etc.) arguably lags behind in the domain of **sustainability**, the one systemic objective for which there is no explicit precedent. Historically, the individual actors constituting the industry have been optimising their respective objective variables (e.g. engine efficiency, airframe aerodynamics, fleet operations, manufacturing, etc.), but all without looking at the proverbial **whole** which ultimately decides on whether a system is ecologically sustainable or not. The hypothesis behind this project is that all such local improvements - including optimisation of these systems for ecological sustainability - are all simply different examples of what is deep down the same problem known in the domain of propulsion: keeping the system of interest **at constant entropy**.

The reasoning is the following (cf. the figure on the opposite side for a rough illustration):

- In general terms, **s**ustainability of a system can be seen as **maintaining the system at steady state** with respect to a certain metric of interest, e.g. *conceptually*, an engine can be considered to be sustainable if it can provide power output indefinitely.
- Keeping the system at steady state entails **keeping its entropy constant**, which entails importing low-entropy energy from the environment and exporting high-quality energy back; no matter what – the **entropy of the environment gets increased**.
- Furthermore, entropy can be interpreted (from statistical mechanics perspective) as **information/knowledge about micro-constituents of the system**: the more we know about the systems details, the better we can maintain it in order, thus keeping it sustainable. Naturally, knowing our system of interest does not mean that we know the environment, which is why local actions likely have negative impact on the entropy of the environment.
- **Planetary sustainability** is conceptually a matter of maintaining the global energy & matter flows at a steady rate, i.e. **keeping the "planetary entropy" constant**. This implies that all the aggregate entropy production of the constituent smaller-scale systems (such as e.g. aeroplane propulsive systems) needs to happen such a way that the aggregate local contributions cancel out at the planetary level.
- Suppose we concentrate on one such small-scale system, civil aeroplane propulsive system. A **sustainable propulsive system**, from the presented perspective, is a propulsive system which **carries out its objective without contributing to global entropy generation** – but it cannot hope to do so if it is isolated from its environment, as is the case in reality.
- It is therefore conjectured that all such systems, at different levels (individuals, companies, states), working in their own respective domains to satisfy their own individual goals, increase entropies of their respective environments, thus collectively increasing the global entropy by what might be characterised as **"internal entropy generation" from the planetary point of view**.
- Which leads to a *de facto* **contradiction** in terms of the enunciated sustainability goals for propulsive (or other) systems.

The goal of this project is to address this nested problem (i.e. the same problem seen at two different and mutually dependent scales) rigorously by employing concepts from thermodynamics/statistical mechanics, and to evaluate the conjecture about the apparent contradiction between isolated local systems optimisation and global sustainability goals.

The propulsive system case study is selected for several reasons. The most obvious one is that it is the system operating on thermodynamic principles and simply put – having a steady-state behaviour of an engine cycle literally means keeping its net cycle entropy at zero. Furthermore, the engine is the primordial system on which the prospect of aviation's planetary sustainability directly hinges, as it consumes the useful energy to yield the useful effects, but also because the new propulsive architectures dependent heavily on rare materials and/or energy sources whose synthesis can incur further atmospheric and local pollution and ecosystem degradation.

Taking a step back - propulsion is simply a matter of **collective movement in a desired direction**. (the bulk of the fluid particles moving in the desired direction) Yet, individual actors moving more and more in different directions (optimising in their own bubbles, while inevitably drawing on resources from the others) is what seems to bring about phenomena like global warming. the global entropy increases because the average (unavoidably self-centred) small-scale behaviour is not aligned with the higher-scale goals. Putting together a method to concurrently describe local and global movement could therefore be a promising avenue to - literally just as much as figuratively - reconcile the conundrum of globally sustainable local systems, as well as to shed some light on how to address the most important issues we collectively face today.

High-Level Language Implementation and Analysis of a 3D Navier-Stokes Solver: update of an existing Matlab code for fluid-structure interaction

Description:
PRIN-3D (PRoto-code for Internal flows modeled by Navier-Stokes equations in 3-Dimensions) is a code based on a high-level algebraic language (Matlab), which may be interesting for low accuracy modeling in Computational Fluid Dynamics (CFD) and FluidStructure Interaction (FSI) analyses.
Developed by Valerio Grazioso (see his thesis http://www.fedoa.unina.it/3189/1/Tesi_di_Dottorato_-_Valerio_Grazioso_-_Pdf-A.pdf) and downloadable at http://sourceforge.net/projects/prin-3d/, this open-source code can however be updated to the latest version of Matlab and its applications to 3D reactive flows revalidated.
This will be one of the purposes of the project, with possibly the development of some FSI applications (parachute, rotor, ...).

Semi-analytical mechanic modeling of fragmentation with cohesive zones

Description:
Fragmentation is a phenomenon of breaking a frangible part of a material system and at the end of which fragments of various sizes are detached at various speeds. This phenomenon is naturally involved in materials with a brittle behavior (glass, ceramics) or in materials subjected to severe stresses localized (impacts, shocks, explosions, etc.). The applications are numerous: energy impacts on composite structures, cutaways by propagation of cracks in metal structures, pressure vessel safety. Predicting the statistical distributions of fragment sizes and of their residual speeds is one of the current challenges of interest.
The numerical modeling of the fragmentation and generation of chips requires the development and implementation of methods dealing with breaking or cutting the discretization of space. To meet the size location and orientation of fractures, and the size and condition of the fragments after fracture, the spatial discretization must be consistent with the physical behavior of materials and structures that separate fragments by fracture surfaces.
Amongst the existing numerical approaches, those based on a grid (such as finite element method) can handle the break at the interface between two consecutive elements, using techniques of breaking the nodal connections (cohesive elements for example), while possible fracture zones exist a priori and are activated only when the criteria jump interfaces are achieved. This approach conveys however some disadvantages: one is the characteristic length that is related to the size of the space discretization and the presence or absence of nodal connections; furthermore, it is also necessary to introduce a contact between fragments if the trajectories of two fragments cross each other.
Knowing the limits of the foregoing methods including cohesive zones, this project proposes to numerically implement one or several semi-analytical models [1,2,3,4] in Matlab with the aim of performing a comparative analysis of the breakdown of elastic structures with fragile behavior to fragmentation.

<u>**References**</u>

[1] W. J. DRUGAN
Dynamic fragmentation of brittle materials: analytical mechanics-based models.
Journal of Mechanics and Physics of Solids, 49, pp. 1181-1208 (2001).
[2] J. R. Gladden, N. Z. Handzy, A. Belmonte, and E. Villermaux
Dynamic Buckling and Fragmentation in Brittle Rods.
Physical review Letters, PRL 94, 035503 (2005).
[3] B. AUDOLY, S. NEUKIRCH
Fragmentation of rods by cascading cracks: Why spaghetti does not break in half.
Phys. Rev. Lett. 95 (2005).
[4] P. ELEK ¤ S. JARAMAZ
Modeling of fragmentation of rapidly expanding cylinders.
Theoret. Appl. Mech., 32 (2), pp. 113-130, Belgrade 2005.

Multi-scale modeling of aircraft door elastomeric seal structures

Description:
Elastomer seals, which guarantee the tightness of aircraft doors, have a significant impact on the handle forces required to open and close aircraft doors. A compromise must be found between a macroscopically 'stiff' seal favoring sealing and a macroscopically 'soft' seal having little influence on internal forces.
ISAE SUPAERO and LATECOERE (world leader in aircraft doors) are jointly developing models and multi-scale tests to predict the behavior under quasi-static or dynamic stresses of these joints made of anisotropic and non-linear materials.

In this context, the objective is to achieve the characterization of the stiffness and flexibility of the joints around the doors by asking the question of what material laws and what modeling hypotheses to retain in order to achieve an equivalent representation of the behavior of the joint during of the movement of the structures which constrain it, and the representation of the contacts between the seal and the door structures, on the scale of the materials and the section of the seal.
Students will be able to contribute to:
• complete the bibliography on the modeling of reinforced elastomer joints and the identification of material properties and variabilities
• enrich the Python code in version 1.0 which allows the determination of material coefficients according to the behavioral laws identified in the bibliography
• develop a “user” material model in Fortran to integrate it into the E.F. solver.
• propose an analysis of uncertainties

Validation of a (Matlab) simulation code using discrete elements for modeling parachute fabrics and porous and anisotropic soft materials

Description:
This work aims to validate a Matlab code developed by the research team to simulate the behavior of the fabric of a parachute sail (Fig. 1). The fabric is modeled with masses, springs and dampers (Fig. 2).

The numericall validation will consist of carrying out various digital tests (traction, shearing, draping, see Fig. 3-4) by highlighting the appearance or not of wrinkles, etc.

Fig. 1. *Real parachute fabrics*

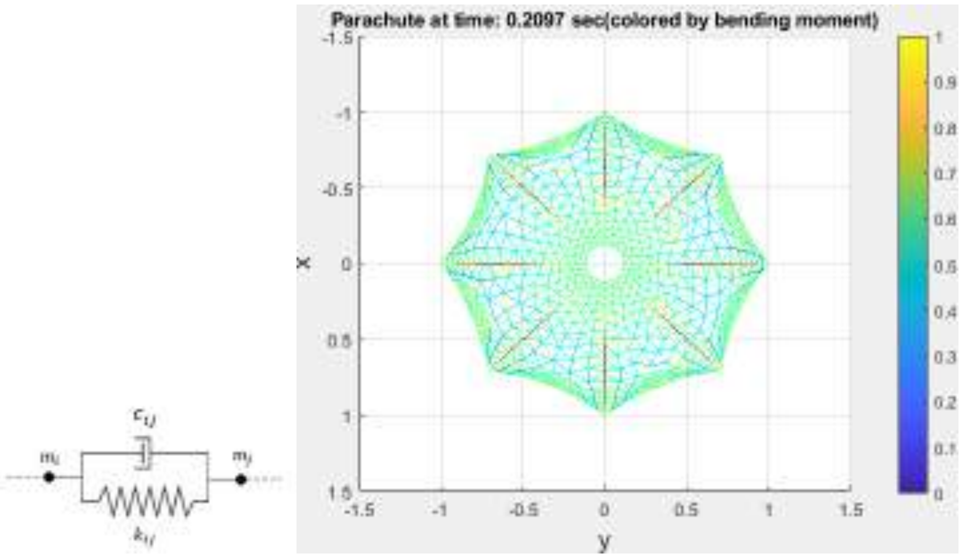


Fig. 2. *Parachute conopy modeled with a net of mass-spring-dampers.*

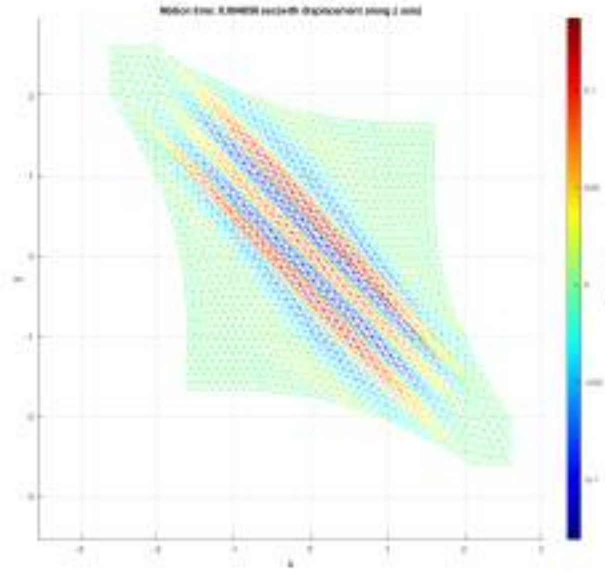


Fig. 3. *Wrinkling fabric test, modeled with a net of mass-spring-dampers.*

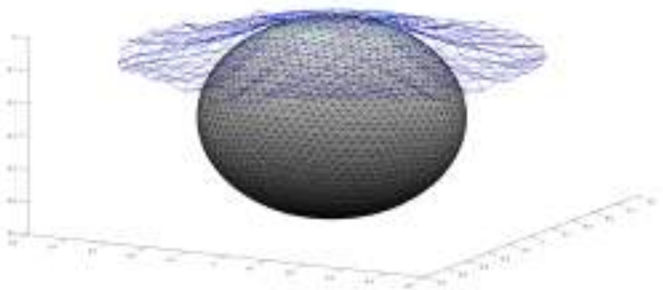


Fig. 4. *Draping fabric test, modeled with a net of mass-spring-dampers.*

Keywords: Numerical simulations; mass models of tissue shock absorber springs; virtual mechanical tests

Unit cell analytical computation: sensitivities and optimal design

Generating eco efficient materials that can be substituted to classical aerospace materials is a challenging problem. One solution is to create digital materials based on unit cell optimization and apply this multiscale topology scheme.

Skills:
Mastering of finite element methods, structural mechanics
Basics of ecodesign
Interest for MDO/optimization

Scientific challenges:
To develop a numerical framework for several unit cell in an analytical way (2D) (Verification with [2]). Solve Effective modulus with periodic boundary conditions [3]
Solve a realistic optimization problem with manufacturing constraints. Use automatic differentiation in Python and EMTO or GGP framework [4,5]
To demonstrate the impact of ultralight digital structures for Aerospace in the entire life cycle of the structure

External/Internal references
Ashby, M. F., & Gibson, L. J. (1997). Cellular solids: structure and properties. Press Syndicate of the University of Cambridge, Cambridge, UK, 175-231.
Ongaro, F. (2018). Estimation of the effective properties of two-dimensional cellular materials: a review. Theoretical and Applied Mechanics Letters, 8(4), 209-230.
Agrawal, Y., & Ananthasuresh, G. K. (2021). Towards optimal heterogeneity in lattice structures. Structural and Multidisciplinary Optimization, 64(4), 2489-2512.
Duriez, E., Morlier, J., Charlotte, M., & Azzaro-Pantel, C. (2021). A well connected, locally-oriented and efficient multi-scale topology optimization (EMTO) strategy. Structural and Multidisciplinary Optimization, 64(6), 3705-3728.
Coniglio, S., Morlier, J., Gogu, C., & Amargier, R. (2020). Generalized geometry projection: a unified approach for geometric feature based topology optimization. Archives of Computational Methods in Engineering, 27, 1573-1610.

How to design Ultralight digital Aerostructures manufactured with robots?

Additive fabrication of digital lattices could fulfil a new paradigm of voxel specific properties at large scale. This will provide several benefits from enabling autonomous fabrication and structural monitoring or repair, to highly tailorable meta-structures. The State of the art (MIT & NASA) demonstrated bulky, low performance structures, difficult to fabricate and inspect.
The objective of the project is to develop an eco-efficient optimization linked to lattice selection and discrete lattice fabrication at large scale.
This lattice can be metallics, and or a combination of fiber/resin. The CO2 footprint can be a metric of optimization
Skills:
Mastering of finite element methods, robotics
Basics of jointing/assembling
Basics of multi-scale homogenization.

Scientific challenges:
To develop autonomous fabrication of an optimized discrete lattice structure integrating optimization, path planning, robot coordination algorithms and robotic hardware using opensource framework.
To Manage complexity of tightly coupled design (space time decomposition).
To demonstrate the impact of ultralight and sustainable digital structures for Aerospace

External/Internal references
K. C. Cheung and N. Gershenfeld, “Reversibly Assembled Cellular Composite Materials,” Science, vol. 341, pp. 1219–1221, Sept. 2013. Publisher: American Association for the Advancement of Science Section: Report.
M. M. J. Opgenoord and K. E. Willcox, “Design for additive manufacturing: cellular structures in early-stage aerospace design,” Structural and Multidisciplinary Optimization, vol. 60, pp. 411–428, Aug. 2019.
Hu, Jingyu, Yunfeng Luo, and Shutian Liu. "Three-scale concurrent topology optimization for the design of the hierarchical cellular structure." Structural and Multidisciplinary Optimization65, no. 5 (2022): 1-19.
B. Jenett, S. Calisch, D. Cellucci, N. Cramer, N. Gershenfeld, S. Swei, and K. C. Cheung, “Digital Morphing Wing: Active Wing Shaping Concept Using Composite Lattice-Based Cellular Structures,” Soft Robotics, vol. 4, pp. 33–48, Mar. 2017.
Wang, Weiming, Dirk Munro, Charlie CL Wang, Fred van Keulen, and Jun Wu. "Space-time topology optimization for additive manufacturing." Structural and Multidisciplinary Optimization61, no. 1 (2020): 1-18
Duriez, Edouard, Joseph Morlier, Miguel Charlotte, and Catherine Azzaro-Pantel. "A well connected, locally-oriented and efficient multi-scale topology optimization (EMTO) strategy." Structural and Multidisciplinary Optimization 64, no. 6 (2021): 3705-3728
Bhat, Krishnaraj Vilasraj, Gabriele Capasso, Simone Coniglio, Joseph Morlier, and Christian Gogu. "On some applications of Generalized Geometric Projection to optimal 3D printing." Computers & Graphics 102 (2022): 199-212.

Global stability of two-dimensional steady flows via different numerical approaches

Theory of stability and transition in wake flows has always been a subject of interest. Their frequent occurrence in nature and practical interest in many engineering applications have dealt these open shear flows an archetypal importance. In particular, flows around wings have drawn even more interest since they count several practical examples in aeronau- tical, civil, mechanical, and naval engineering and, at the same time, are characterised by fundamental physical phenomena which influence the aerodynamic characteristics, such as separation, transition and wake formation.

The aim of this research project is to investigate the global stability of steady spanwise- homogeneous separated flows developing around periodic wings for different flow regimes (see figure 1). The objective is to provide a complete description of the stability properties of the global eigenmodes using different numerical approaches: finite volumes, spectral differences and finite elements.

Optimization of toroidal propeller

Key words: CFD, BEMT, code development, numerical simulation, propeller

Description:

Toroidal propellers are an innovation in propeller design, characterized by their ring-shaped or closed-loop structure. Unlike traditional propellers, which have straight or slightly curved blades, toroidal propellers wrap the blades around themselves, creating a torus-like geometry. This unique shape offers distinct advantages for drone applications, particularly in terms of noise reduction and propulsion efficiency.

One of the primary benefits of toroidal propellers for drones lies in their ability to reduce noise. Conventional propellers generate significant noise due to tip vortices, which occur when air escapes from the rotating blade tips. In a toroidal propeller, the looped shape restricts the formation of these tip vortices, significantly reducing the noise produced. For drones operating in urban environments or sensitive applications (such as surveillance, residential area deliveries, or industrial inspections), this noise reduction represents a major advantage.

Beyond acoustic benefits, toroidal propellers can also improve the energy efficiency of drones. The ring shape helps stabilize the airflow around the blades and reduces drag induced by turbulence, leading to better propulsion efficiency. Drones equipped with toroidal propellers can thus use less energy for the same level of thrust, extending their range and lowering operational costs.

Optimizing toroidal propellers for drones and other applications involves various methods to improve performance, noise reduction, and energy efficiency. The most promising shape optimization techniques include:

Computational Fluid Dynamics (CFD) Modeling : This approach uses numerical simulations to analyze airflow behavior around toroidal propellers. With CFD, engineers can observe airflow patterns, turbulence, and pressure points, helping them optimize the propeller shape to minimize drag and noise. By adjusting parameters such as ring curvature or angle of attack, CFD allows for virtual testing of multiple configurations without the need for physical prototypes.

Parametric Optimization : In this method, different geometry parameters of the propeller (ring diameter, blade thickness, curvature radius, etc.) are systematically adjusted to determine the combination that offers the best performance. Parametric optimization can be assisted by genetic algorithms or machine learning techniques to quickly explore a large solution space and converge on an optimal shape.

Experimental tests : Although costly, wind tunnel testing remains a reliable method for evaluating and adjusting toroidal propeller performance. These tests can recreate real-world flight conditions and gather precise data on noise reduction, thrust, and energy efficiency. The results from these tests are often used to validate numerical models and make fine adjustments to propeller prototypes.

Additive Manufacturing and Rapid Prototyping : Additive manufacturing, or 3D printing, enables the rapid production of toroidal propeller prototypes with different geometries. This method offers flexibility in shapes and facilitates quick modifications, allowing engineers to test and refine multiple versions of the propeller in a short time. Rapid prototyping is especially useful for acoustic optimization iterations, as it allows for continuous adjustments based on test results.

The Blade Element Momentum Theory (BEMT) : Is a method used to analyze the performance of propellers and rotors, particularly in aeronautics and renewable energy. It combines two theories: Momentum Theory, which models the change in air velocity through the propeller to estimate thrust, and Blade Element Theory, which divides each blade into sections or elements to locally analyze lift and drag forces. In BEMT, the propeller or rotor is split into segments, and each segment is treated as a small wing generating lift and drag based on its angle of attack, speed, and shape. By applying aerodynamic equations to each element and then integrating the results over the full blade length, BEMT provides an estimate of thrust, torque, and overall efficiency of the propeller or rotor. This method is particularly useful for optimizing blade geometry to maximize performance.

The optimization of toroidal propellers through these various methods paves the way for quieter, more efficient drones that are better suited to the demands of urban mobility.

In this project, students will be asked to work on two aspects:

- **Research the most suitable numerical methods to simulate this type of geometry.**
- **Develop a low-fidelity code (Matlab) to apply the BEMT on a toroidal propeller.**
- **Perform some CFD simulations as references for the low-fidelity code.**
- **Consider an optimization method, using the low-fidelity code to determine the best shape(s).**
- **Possibility to print and test on an experimental performance test bench of some propeller.**

Drone with toroidal propellers

Example of the shape of toroidal helices

UAV noise study for noise pollution reduction

Description:
Key words: CFD, Aeroacoustic, UAV noise
The use of unmanned air vehicles (UAVs) in many diverse applications, record videos for the cinema industry, perform inspections of cracks in dams, perform reconnaissance missions, just to name a few uses, increased in the last decades (see Figure 1). This development may increase noise pollution in the near future and lead aviation agencies to put in place more stringent certification standards to protect the civilian population from UAV noise. It will then become important to understand UAV noise generation mechanisms in order to develop efficient noise reduction strategies. The main noise components for UAVs are due to rotors, rotor-rotor, rotor-fuselage and rotor-duct interactions. While a relatively large number of experimental and numerical analysis were recently carried out on isolated rotors at low Reynolds and Mach numbers [1-2] only a few addressed rotor fuselage interaction noise [3-4]. Zawodny et al. [3] showed first that the presence of a beam in the rotor wake increases the acoustic level at the harmonics of the blade passing frequency (BPF).

Figure 1: Airbus air taxi (left) and quadcopter drone (right).

The Aerodynamics, Energy and Propulsion Department (DAEP) at ISAE-SUPAERO experimentally investigated that phenomenon for different beam dimensions, positions and geometries [4]. We found that, with the presence of a beam, the highest amplitude for the harmonics can be as high as the amplitude for the BPF, as it can be seen on Figure 2 (right). It seems from analytical studies [5-6] that the main acoustic source for these harmonics is the potential effect of the rotor blade on the beam This extra noise source dues to the presence of the beam is well reproduced by numerical simulation [7-8] and analytical formulations [6], except at 2xBPF (Figure 2). At this frequency, the difference could be explained by diffraction of a noise source coming from the rotor by the beam (or the opposite). This is currently not taken into account in the current numerical setup and lead to the present student project.

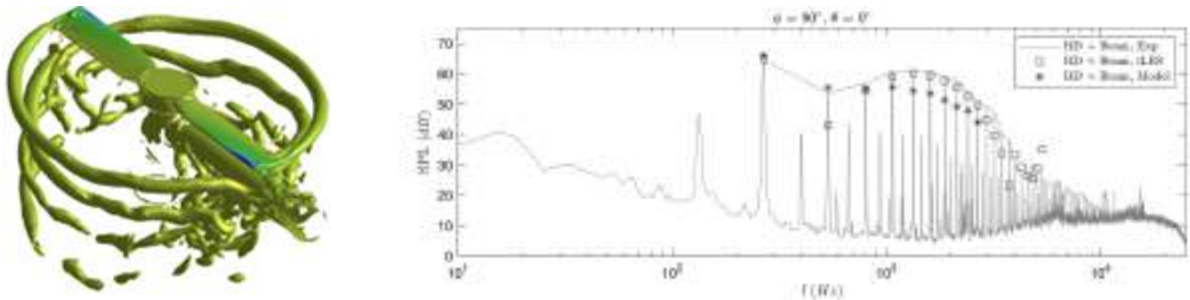


Figure 2. Left: Vortex tip impingement on the beam computed by CFD (iso-contour of Q criterion = 10^6 $1.s^{-2}$) [5]. Right : far field acoustic spectra radiated in the rotor disk plane[5].

The current CFD simulations used the Ffowcs-Williams and Hawkings (FWH) analogy to propagate the pressure fluctuations from the wall surfaces to the farfield, which allows to analyze farfield acoustics. This method has the advantage to be very fast to compute. However, it does not consider the reflection of acoustic waves on the rotor and beam walls. That could explain the gaps of acoustic levels for the first and second harmonics of the BPF between the CFD simulations and experimental measurements, while these acoustic levels are in very good agreement for the harmonics of higher order. A way to take into account the reflection of acoustic waves on the rotor and beam walls is

the use of the Perturbed Convective Wave Model (PCWM) in Star CCM+ (commercial CFD software). This model is a hybrid acoustics model using a wave equation to calculate, inside the CFD simulation, the sound generation and propagation in incompressible flows [9-10].
The objective of this project will be to validate first the calculation method of the volumetric acoustic sources of the PCWM by simulating the acoustic radiation generated by vortex pairing in a mixing layer [11]. Then a configuration of an isolated rotor (without beam) will be simulated and compared to our experimental measurements to validate the numerical setup. Figure 3 shows an example of a first simulation. The beam will be finally added to study the interaction between the rotor and the beam and its effect on the acoustic radiation. Depending the advancement of the project and/or the number of students, a configuration with a ducted rotor could also be studied.

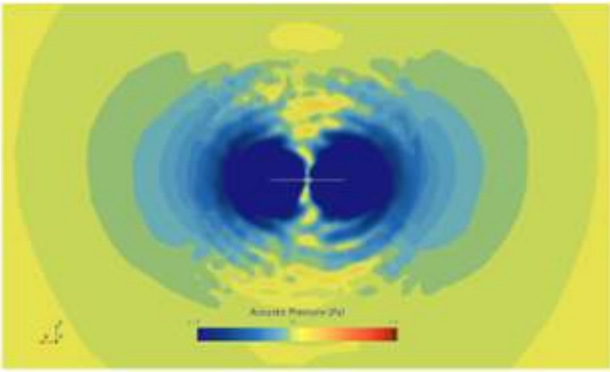


Figure 3: Acoustic pressure between -1 Pa and +1 Pa in the wingspan plane section for the isolated rotor.

The project will be organized :

- **Bibliography for vortex pairing in mixing layer**
- **Learn the use of the PCWM in Star CCM+**
- **Simulation of vortex pairing in mixing layer to validate the calculation of volumetric acoustic sources of the PCWM**
- **Simulation and validation of the isolated rotor configuration**
- **Simulation and investigation of rotor-beam interaction to understand which phenomena increase the acoustic levels of the 2nd harmonic of the BPF**
- **Simulation and investigation of rotor-duct interaction to assess the impact of the duct on the acoustic levels**

References:

[1] Zawodny, N., Boyd Jr, D., and Burley, C.: Acoustic characterization and prediction of representative, small-scale rotary-wing unmanned aircraft system components. In: AHS 72nd Annual Forum . West Palm Beach, Florida, USA (2016).
[2] Gojon, R., Jardin, T., and Parisot-Dupuis, H., “Experimental investigation of low Reynolds number rotor noise,” J. Acoust. Soc. Am. 149(6), 3813–3829 (2021).
[3] Zawodny, N., and Boyd, D.: Investigation of Rotor–airframe Interaction Noise Associated with Small-Scale Rotary-Wing Unmanned Aircraft Systems. Journal of the American Helicopter Society. Vol. 65, No. 1, 2020, pp. 1–17.
[4] Gojon, R., Jardin, T., Parisot-Dupuis, H., and Mellot B.: Aeroacoustic radiation of low Reynolds number rotors in interaction with beams. Journal of the Acoustical Society of America (2023).

[5] Wu, Y., Kingan, M., and Go, S.: Propeller-strut interaction tone noise. Physics of Fluids. Vol. 34, No. 5, p. 055116 (2022).
[6] Vella, E., Gojon, R., Parisot-Dupuis, H., Doué, N., Jardin, T., & Roger, M., Mutual Interaction Noise in Rotor-Beam Configuration. In 30th AIAA/CEAS Aeroacoustics Conference (2024) (p. 3318).
[7] Gojon, R., Doué, N., Parisot-Dupuis, H., Jardin, T.: Aeroacoustic radiation of a low Reynolds number two-bladed rotor in interaction with a cylindrical beam. 28th AIAA / CEAS Aeroacoustics Conference, (2022).
[8] Doué, N., Gojon, R., Jardin, T.: Numerical investigation of the acoustics radiation of a two-bladed rotor in interaction with a beam. Fluid-Structure-Sound Interaction & Control. Busan, South Korea (2023).
[9] Seo, J. H., Moo,, Y. J[RG5] .: Linearized perturbed compressible equations for low Mach number aeroacoustics. Journal of Computational Physics 218 (2006) 702–719.

[10] Piepiorka, J., Von Estorff, O.: Numerical investigation of hydrodynamic/acoustic splitting methods in finite volumes including rotating domains. 23rd International Congress on Acoustics, Aachen, Germany (2019).
[11] Bogey, C., Bailly, C., & Juvé, D., Numerical simulation of sound generated by vortex pairing in a mixing layer. AIAA journal, 38(12), 2210-2218 (2000).

Numerical study of airflow convection and, scattering effect on sound wave propagation

Description:
Key words: CFD, Aeroacoustics, signal processing
Global economic and demographic expansion is leading to an increase in noise pollution associated with land and air transportation as well as new energy developments (wind power). Noise pollution regulations are becoming increasingly strict and the reduction of aerodynamic noise sources, known as aeroacoustics, is therefore a major challenge. The characterization of aeroacoustic sources in the pre-project phase by means of wind tunnel tests is a key step to help understand noise generation mechanisms and to be able to develop effective noise reduction strategies.
For this purpose, microphone array measurements are performed during wind tunnel tests in order to characterize noise sources. Arc antenna are used to obtain the directions where the radiated acoustic energy is maximum, this is called the directivity of sources. Big 2D arrays are used to perform sound source localization which allows to obtain source maps on the mockup for each frequency of interest (see Figure 1). This process is based on the interpretation of propagation delays measured between each microphone of the array for a given source-array distance [1]. However, this methodology assumes spherical sound wave propagation in free field without airflow, which is not the case for wind tunnel measurements. This can lead to inaccuracies of source maps in terms of source positioning, level and spatial resolution. The source positioning can be corrected by using a calibration procedure or analytical models assuming that the shear layer is very thin [2]. However, it is not always easy to correct, especially for the source level.

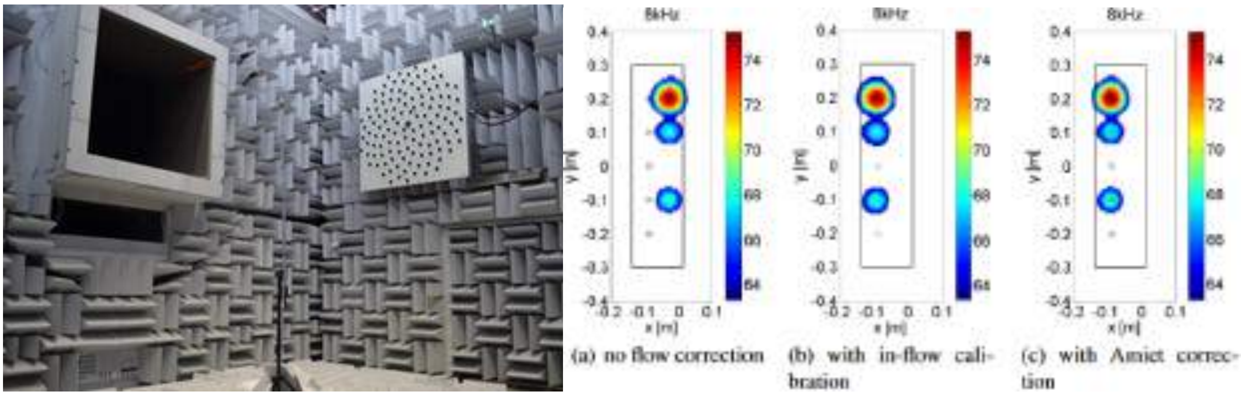


Figure 1: Sound source localization array in ISAE-SUPAERO aeroacoustic wind tunnel (left). Source positionning correction on source map for an airframe with cavities [2] (right).

The objective of this research project is to study numerically the effect of an airflow on microphone array measurements in order to be able to derive corrections. For that, the first step will be to simulate using large eddy simulations (LES) a monopole source and its radiation towards different arrays without airflow, and then with uniform airflows of various mean and fluctuating velocities, to assess the impact of wave convection and aerodynamic fluctuations on virtual microphone array measurements. The second step will be to simulate a monopole source radiating inside a jet of different velocities to study the effect of sound wave scattering due to the shear layer [3]. This can lead to a broadening of the sound power spectral density due to a non-linear interaction between the source radiation and low frequencies spanwise velocity fluctuations in the shear layer (see Figure 2). The same kind of study will be then performed with vertical and horizontal dipoles as sources, that are representative of the sound radiated by an obstacle in airflow. Depending the advancement of the project, spectral post processing could be applied to numerical data to better highlight the interaction between sound waves and airflows. Two dimensional fast Fourier transform (FFT2D) could be used to separate the acoustic and aerodynamic components of the fluctuating pressure measured by microphones. This already allows to decompose aerodynamic and acoustic fields generated by a supersonic impinging jet (see Figure 3). Spectral proper orthogonal decomposition (SPOD) [4] could also be applied to these numerical data to understand the link between the characteristics of the shear layer and the modification of the sound wave propagation due to the crossing of the shear layer [5]. A perspective of this work is to apply the developed methodology on more realistic aeroacoustic configurations, as in-flow cylinder noise, allowing a correction of wind tunnel microphone measurements.

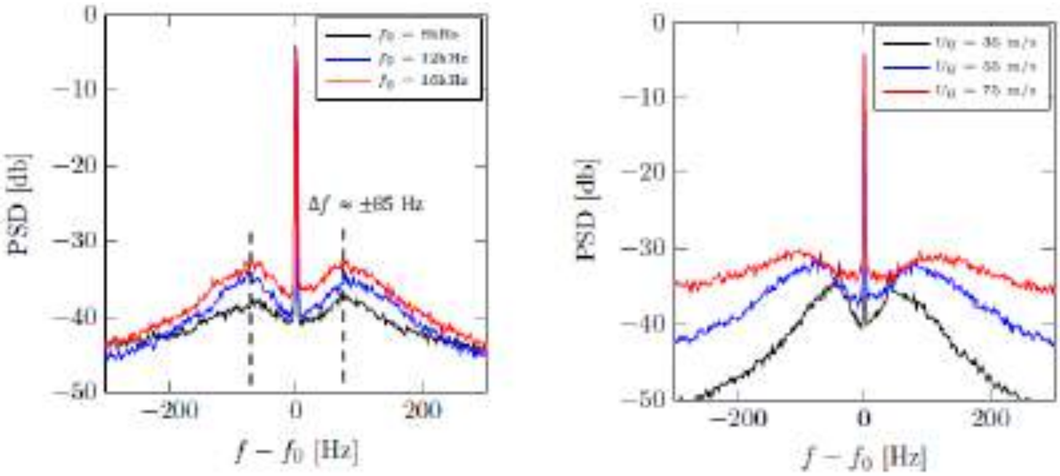


Figure 2: Illustration of spectral broadening measured by a microphone outside the jet of a wind tunnel for several source frequencies and airflow velocities [3].

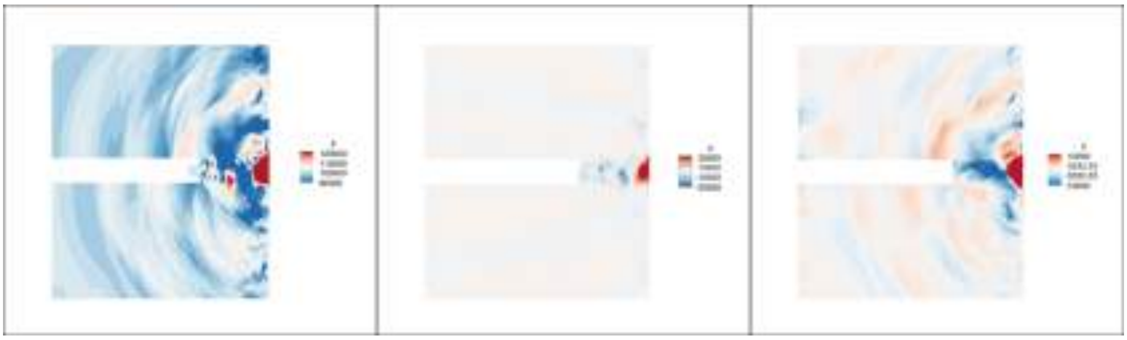


Figure 3: Fluctuating pressure field (left) decomposition into hydrodynamic (center) and acoustic (right) components using SPOD for a supersonic impinging jet [3].

References

[1] J. Billinsgley and R. Kinns, « The acoustic telescope », J. Sound Vib. 48 (4), 485-510 (1976).

[2] Kroeber, Stefan, et al. "Systematic comparison of microphone array measurements in open and closed wind tunnels." 16th AIAA/CEAS Aeroacoustics Conference. 2010.

[3] F. Scarano, E. Salze, A. Pereira, E. Jondeau, V. Clair and J. Chambon, Investigation of spectral broadening of sound waves by a turbulent shear layer using particle image velocimetry, In 30th AIAA/CEAS Aeroacoustics Conference, AIAA 2024-3164 (2024).

[4] Towne A, Schmidt OT, Colonius T. Spectral proper orthogonal decomposition and its relationship to dynamic mode decomposition and resolvent analysis. JFluid Mech 2018;847(1):821–67. http://dx.doi.org/10.1017/jfm.2018.283.

[5] M. Fiore, H. Parisot-Dupuis, B. Etchebarne and R. Gojon, Spectral proper orthogonal decomposition of coupled hydrodynamic and acoustic fields: Application to impinging jet configurations, Computers and Fluids, 241, 105484, 2022.

An adaptive Software Defined Radio (SDR) secure communication scheme for satellite systems

Since its introduction in one of Shannon's most celebrated papers [1] and its elaboration by Wyner [2] and Csisz ár & K örner [3], physical layer security has proved to be a promising means of securing communications by exploiting the inherent non-reproducible randomness in the communication links (noisy channels, fading channels, quantum optical channels ,...).

One of the most common settings in physical layer security is the wiretap channel, in which a legitimate transmitter (Alice) wishes to communication a message to a legitimate receiver (Bob) whilst keeping it secret from an evesdropper (Charlie). As opposed to cryptographic security which distills secrecry from the advantage given to Bob over Charlie by the knowledge of a secrect key, physical layer security distills secrecy from the advantage of Bob having a better communication link than Charlie. This physical advantage stems from either a better signal to noise ratio (SNR), higher number of receive antennas, better antenna gains etc.

For such a setting, security is achieved through the so-called wiretap coding principle, for which practical constructions have been proposed using different error correction codes [4, 5] and successfully simulated through MATLAB [6], leading to the deployment of the PHYSEC test-bench in the Telecommunications Laboratory of ISAE-SUPAERO.

The PHYSEC testbench is based on Software Defined Radio (SDR) components, i.e., USRPs, which constitute highly adaptive and low complexity of-the-shelf solutions for the emulation of realistic communication links. The main purpose of this project is to improve the USRP implementation of the polar wiretap code [7] to allow for a dynamic adaptation of the code to varying communication conditions, namely adapt to varying SNRs of both Bob and Charly, which is necessary to accomodate LEO and MEO-type satellites.

Misc.

This project will take place in the Department of Electronics, Optronics and Signal processing (DEOS) of ISAE Supa éro, in the Telecommunication Laboratory of the ComIT research group (Communication and Information Theory).

The supervisor of the project is Meryem Benammar (meryem.benammar@isae-supaero.fr). Major proposed by supervisors: SANS.

References

1. [1] C. E. Shannon, "Communication theory of secrecy systems," Bell Labs Technical Journal, vol. 28, no. 4, pp. 656–715, 1949.
2. [2] A. D. Wyner, "The wiretap channel," Bell Labs Technical Journal, vol. 54, no. 8, pp. 1355– 1387, 1975.
3. [3] I. Csiszar and J. Korner, "Broadcast channels with confidential messages," IEEE trans- actions on information theory, vol. 24, no. 3, pp. 339–348, 1978.
4. [4] M. Hayashi and R. Matsumoto, "Construction of wiretap codes from ordinary channel codes," in 2010 IEEE International Symposium on Information Theory, pp. 2538–2542, 13-18 June 2010.
5. [5] O. O. Koyluoglu and H. El Gamal, "Polar coding for secure transmission and key agree- ment," IEEE Transactions on Information Forensics and Security, vol. 7, no. 5, pp. 1472– 1483, 2012.
6. [6] K. Taleb and M. Benammar, "On the information leakage of finite block-length wiretap polar codes," in 2021 IEEE International Symposium on Information Theory (ISIT), pp. 61–65, 2021.
7. [7] K. Taleb and M. Benammar, "Finite blocklength wiretap polar codes: construction and performance," in 2022 International Wireless Communications and Mobile Computing (IWCMC), pp. 389–394, 2022.

Modeling compressor water injection for hybrid hydrogen-based propulsion cycles

Description:

Context and motivation

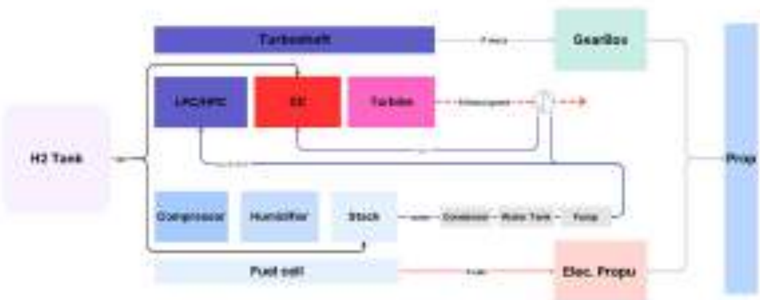
The injection of water in the compressor stages or the combustion chamber of a gas turbine engine is well-known for its enhancement of cycle efficiency, as well as its potential for significantly reducing NOx emissions. Indeed, a humidified mass flow, having a higher heat capacity, contains significantly more extractable energy than dry air. Furthermore, adding a diluent to the combustion zone reduces the formation of thermal NOx by lowering the temperature. Inlet and compressor water injection was applied to a number of the first generation turbofan engines in the 1950s in order to enhance the take-off thrust. The injection of steam into the gas turbine cycle is commonly used in stationary power plant technology, with the steam typically produced using the gas turbine exhaust heat.

These concepts typically require a liquid water supply, which makes its application in aviation a challenge. The recent concept of a Water-Enhanced-Turbofan engine, currently under technology demonstration [1], features an in-flight water-recovery from the gas turbine exhaust flow via a condenser, and a steam generator recuperating the exhaust heat. More recently, Seitz et al. [2] introduced a fuel cell – gas turbine hybrid propulsion concept exploiting the synergies between the water produced by a hydrogen fuel cell and steam injection in the burner. The water from the fuel cell (at low pressure) is conditioned through a process of condensation, pressurisation and re-vaporization before being injected in the burner at a higher pressure.

Deliberate water injection is also beneficial to compressor stage efficiency by reducing the temperature rise during the compression. However, there is a tradeoff to find between the beneficial cooling effect of the liquid injection, and the detrimental entropy generation during the evaporation [3] by carefully selecting the water-to-air ratio and the droplet size of the spray at injection.

Proposed work

The aim of this project is to propose a system-level model of a gas turbine cycle featuring water and steam injection, with a focus on compressor water injection. The work will begin with a literature review to understand the phenomenology and screen available modeling approaches as well as validation test-cases in the literature. We will propose and implement (in Matlab) a low-order model, such as a 0D or a multi-stage compressor model. We will use the developed models to study the cycle of the DGEN turbofan (at the DAEP turbofan test bed) and to discuss the potential of this set-up as a future validation test case. Finally, we aim to use the developed models to study hybrid fuel-cell, gas-turbine cycles in more detail, for examples with different levels of integration exploiting water and/or steam injection.



This project is motivated and co-supervised by VirajH2, a start-up company funded in 2024, and the DAEP.

References

Schmitz, Oliver, Hermann Klingels, and Petra Kufner. 2021. "Aero Engine Concepts Beyond 2030: Part 1—the Steam Injecting and Recovering Aero Engine." *Journal of Engineering for Gas Turbines and Power* 143 (February). https://doi.org/10.1115/1.4048985.

Seitz, Arne, Markus Nickl, Florian Troeltsch, and Kathrin Ebner. 2022. "Initial Assessment of a Fuel Cell—Gas Turbine Hybrid Propulsion Concept." *Aerospace* 9 (January): 68. https://doi.org/10.3390/aerospace9020068.

Zawadzki, Natan, Artur Szymański, and Uyioghosa Igie. 2024. "Effect of Water Injection on Turbofan Engine Compressor Operation and Aerodynamics." In *ASME Turbo Expo 2024: Turbomachinery Technical Conference and Exposition*. American Society of Mechanical Engineers. https://doi.org/10.1115/gt2024-125908.

Design of axisymmetric supersonic nozzles with low cost models

Description:
In non-ideally expanded supersonic jets, several acoustic components, including screech noise, mixing noise and broadband shock-associated noise are observed. The screech noise is due to an aeroacoustic feedback mechanism established between the turbulent structures propagating downstream and the acoustic waves propagating upstream. This mechanism was described by Powell in the 50s [1] and then by Raman [2], who proposed that the turbulent structures developing in the jet shear layers and propagating in the downstream direction interact with the quasi-periodic shock cell structure of the jet, creating upstream propagating acoustic waves. The resonant loop is closed at the nozzle lips, where sound waves are reflected back and excite the shear layers. Moreover, for round jets, Powell identified four modes, labelled A, B, C, and D, based on the screech frequency evolution with the ideally expanded Mach number. Each mode is dominant for a specific ideally expanded Mach number range and frequency jumps for the tones are noted between the modes. Davies [3] studied the oscillation modes of the jets associated with the screech modes. They found that A mode is linked to axisymmetric oscillation modes of the jet, B to sinuous and sometimes helical modes, C to helical modes and D to sinuous modes. Oscillation modes A and C have already been quite well explained in the literature [4,5]. However, oscillation modes B and D are still subject to discussion as they do not follow the classical instability modes of an axisymmetric jet (axisymmetric, helical, or superior azimuthal order modes). That is why we would want to study in depth the B oscillation mode of screeching jets for different nozzles, with different exit design Mach numbers. A fast and reliable tool to design such a nozzle is thus needed in a first place in order to design several nozzles with several design Mach numbers that will then be used in numerical and experimental studies at the lab.

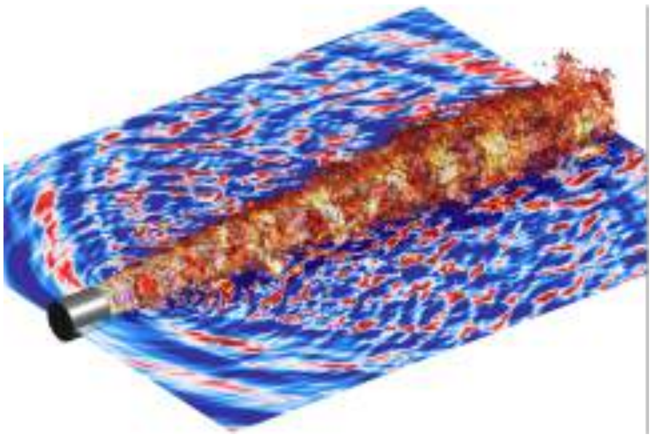


Figure 1: Non-ideally supersonic jet experiencing screech along the C stage. Representation of several density isosurfaces, and of the pressure fluctuations. The nozzle is in gray. [4]

Program of internship
The aim of this project is to develop a low-cost Python code that is able to compute the supersonic flow in axisymmetric/planar configurations for a given geometry/flow parameters with the purpose of designing nozzles or supersonic air intakes. This code will use the method of characteristics or Mach lines as in [6] in different situations (isentropic or not, 2D planar or 2D axisymmetric). In particular, for nozzles, the purpose will be to prevent shocks inside of the nozzle and obtain a uniform flow at the exit. Once developed, the code will be used to generate different nozzle geometries generating the exact equivalent ideally expanded Mach number (mode B) but different exit Mach number. By doing so, the strength of the shock cells downstream of the nozzle will be changed, and in future studies, this parameter will be analyzed with respect to the radiated noise.

Bibliography

[1] A. Powell. On the mechanism of choked jet noise. Proceedings of the Physical Society. Section B, 66(12):1039–1056, 1953.
[2] G. Raman. Supersonic jet screech: half-century from powell to the present. J. Sound Vib., 225(3):543–571, 1999.
[3] M.G. Davies and D.E.S. Oldfield. Tones from a choked axisymmetric jet. ii. the self excited loop and mode of oscillation. Acta Acust. United Ac., 12:267–277, 1962.
[4] Gojon, R., Bogey, C., Mihaescu, M.: Oscillation modes in screeching jets. AIAA J. 56(7), 2918–2924 (2018)

[5] Edgington-Mitchell, D., Jaunet, V., Jordan, P., Towne, A., Soria, J., Honnery, D.: Upstream-travelling acoustic jet modes as a closure mechanism for screech. J. Fluid Mech. 855 (2018)
[6] Argrow, B. M., Comparison of minimum length nozzles, J. Fluids Eng. 110(3), 283–288 (1988).

Science and Sports - Drag Reduction by Laminar Flow Extension on Rowing Shells (DolFin)

Description:

<u>Background</u>
From a hydrodynamic design perspective the traditional shell design was focused mainly on the wave drag and form (pressure) drag reduction which is quite significant. The form drag has naturally been reduced by streamlining the shell towards the leeside; however this results in a much larger wetted area which causes an increase in the skin friction drag especially in the presence of a turbulent boundary layer. From live experiments on a full scale rowing shell as shown in figure 1 hot-film sensors have confirmed that the boundary layer undergoes laminar to turbulent transition in the upstream region of the racing shell/boat. Early boundary layer transition would result in a rapid rise in skin friction drag and therefore increase in the overall drag penalty. From basic laminar hydrofoil or aerofoil design several strategies have already been proposed for delaying transition. Hence, there is a potential of skin friction drag reduction by revisiting the design of the profile shape of racing shells, canoes, kayaks or other watercraft in general, inspired by laminar flow profile design.

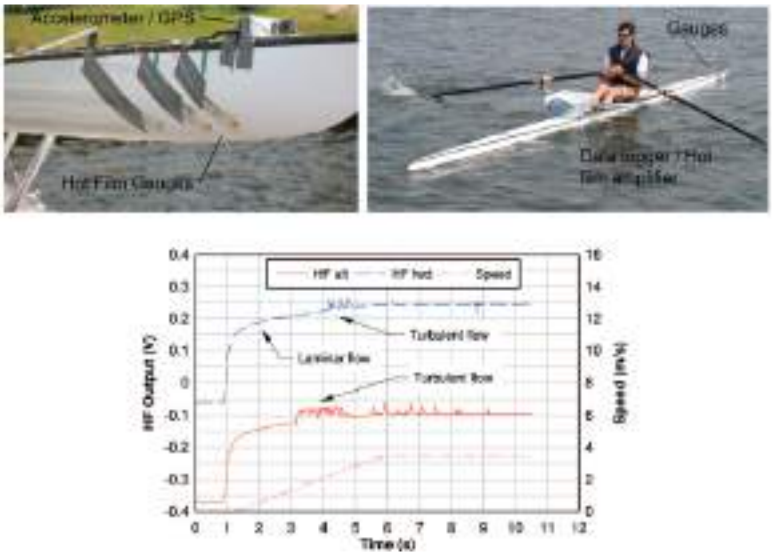


Figure 1: Live experiment on transition detection over a rowing shell, Day et al. 2011, J. Sports Sci.

<u>Objectives</u>
This project is under the Science and Sports initiative of ISAE-SUPAERO and part Science2024 project which aimed at assisting professional athletes in improving their performance during the Paris Olympic and Paralympic Games in 2024 and we are now in the process of building the project for 2028. Here we aim at using an existing aerofoil design optimisation technique that calculates the pressure distribution and the development of the boundary layer and the linear stability of the boundary layer to infer transition. This is modeled in the DolFin suite developed previously by A. Zimmermann in his MAE research project, which at this stage can optimise the 2D section shape of rowing shells in a quasi steady scenario. This project will look at the design of the full 3D shell followed by CFD simulation of real rowing scenario which follows an oscillating forward speed profile. This oscillating pressure field will generate an oscillating boundary layer and the stability of this boundary layer will be further evaluated using the transition prediction code LoLA. In order to capture this effect the boundary layer and stability analysis code will have to be modified and this will constitute the second phase of the project. This study will serve as a precursor for the design and manufacturing of a test model for wind or water tunnel testing, followed by a full scale prototype which can be tested under real rowing conditions.
<u>Roadmap</u>

- Literature review on rowing shell or boat design
- Familiarisation with DolFin for 2D section optimisation
- Design and optimisation of full 3D rowing shell
- StarCCM+ simulation of full 3D shell with real forward velocity profile

- Boundary layer stability analysis for prediction of transition
- Further optimisation

Prerequisites

- High motivation for wind tunnel testing
- Good grasp of basic aerodynamics or fluid mechanics
- Good grasp of applied maths
- CFD simulation skills – grid generation, simulation and post processing (not compulsory)
- Coding skills in Matlab or other tools for further data post-processing and analysis
- Research oriented attitude

Bibliography

1. A. Zimmermann, “Drag reduction by laminar flow extension on rowing shells”, Masters in Aerospace Research Project, ISAE-SUPAERO, 2021
2. https://sciences2024.polytechnique.fr/

Operation in highly perturbed environment inspired by nature

Background
The operation of micro aerial vehicles (MAVs) and, light manned and unmanned aerial vehicles (UAVs) in urban environment can be rapidly compromised due to the highly perturbed environment as a result of the heterogeneous topography as shown in figure 1. Whilst the majority of man-made transport system will very rapidly encounter their limits in highly perturbed environment, in nature many species can still operate and even exploit the perturbations to reduce energy expenditure. For instance albatrosses exploit the environmental perturbation to navigate over very long distances, during the process of dynamic soaring over the oscillating sheared velocity profiles at the interface of air and sea water. This is achieved by either changing the trajectories or by wing morphing to adapt to the rapidly varying upstream conditions.

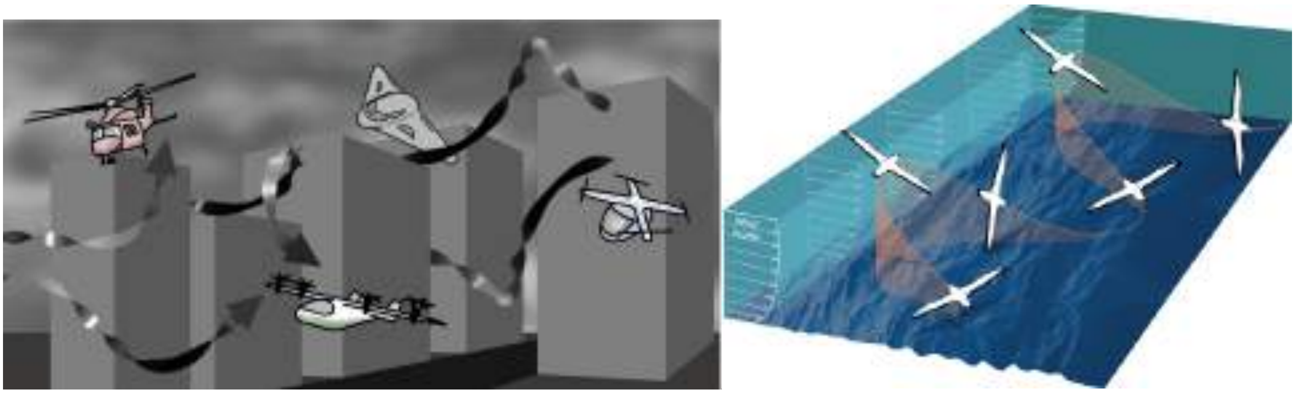


Figure 1: Schematic of the operation in highly perturbed environment (LHS) and dynamic soaring of albatross (RHS).

Objectives
Further understanding of the strategies employed by birds and the underlying flow behaviour could help in the implementation of bio-inspired technologies for high stability and manoeuvrability and for the development of loads and gusts alleviation techniques. Here we aim to further develop tools which would allow us to approach more realistic perturbations encountered in nature. This could be achieved by the implementation of systems for generating oscillating freestream flows such as gusts in the wind tunnel over dynamic and deformable models which results in very strong fluid and structure interaction leading to further complexity for numerical simulations.
Roadmap
Following an extensive literature review on the types of perturbations encountered in urban environment, these perturbation will be simulated numerically and recreated experimentally to validate the numerical simulations, their effect on first a rigid static wing model will be studied and followed by the implementation of gust alleviation strategies such as trajectory optimisation or morphing. The project will be later divided into two parts first we would like to generate and characterise oscillatory freestream flow such as gusts. The highly unsteady flow with strong viscous effects at low Reynolds number poses significant challenges for numerical simulations; therefore the complimentary experimental campaign will be conducted to validate the numerical approach that will be used for further simulation.
<u>Numerical simulation</u>
- Familiarisation with grid generation and numerical simulation tools (StarCCM+).
- Star CCM+ simulation on rigid wing with and without oscillatory freestream flow.
- Analysis and validation with experimental results on rigid wing.
- Star CCM+ simulations repeated with gust alleviation strategies.
- Analysis and validation with experimental results.

<u>The wind tunnel experiment</u>
- Characterisation of the oscillatory freestream flow with and without wing models for a series of oscillating velocity profile using hot wire anemometry.
- Aerodynamic force measurement on rigid wing model with and without oscillatory freestream flow – force balance.
- Characterisation of the flow over the rigid wing with and without oscillatory freestream flow with hot wire anemometry and PIV.
- Aerodynamic force measurement to demonstrate gust alleviation techniques.
- Flow characterisation to understand the physical mechanisms responsible for gust alleviation.

Prerequisites (not compulsory)

- CFD simulation skills
- Wind tunnel testing skills
- Python or Matlab for further data post-processing and analysis
- Ability to work autonomously

Science and Sports - Design of hydrofoils inspired by nature

Background
Over the period of evolution nature has optimised the wings and fins in such a way that they are very efficient at producing the required aerodynamic or hydrodynamic forces for propulsion at minimum drag. Therefore, these solutions can be exploited to improve the aerodynamic or hydrodynamic efficiency of manmade systems, such as foils employed to lift up aquatic vehicles or boards off the water surface and thus reducing drag as shown in figure 1 below. Foil designs have already been inspired by the flippers of the humpback whales which gained a lot of interest in many other engineering applications, so here we would like to assess the benefits other species that exist in nature for the design of hydrofoils with both a higher lift and lower drag characteristics, especially induced drag using the approach developed during previous Master research project [1].

Figure 1: Concepts applying hydrofoils to lift off from the water to reduce skin friction drag. Images from Wikipedia.

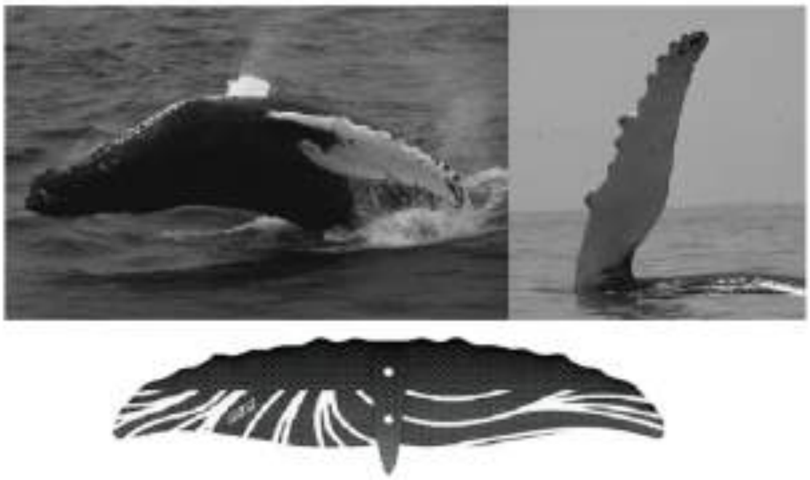


Figure 2: Leading edge tubercles of humpback whale's flippers (top) and a hydrofoil with wavy leading edge inspired by leading edge tubercles. Image from surfshop.fr.

Objectives
To design and optimise the hydrofoil with higher hydrodynamic performance using low order wing design tool, while inspiring from concepts that already exist in nature.
This project is slit into two parts with 1 part looking at the design of the hydrofoil fully immersed in water and the second part looking at mast which is partly immersed in water and partly in air. In the latter case a two phase flow at an interface will be considered
Roadmap
The project will be divided into two parts; first the wing planform will be designed by inspiring from lifting surface inspired nature, identified during the literature. This will be followed by CFD simulation of the optimal planforms and potential wind tunnel testing in the second year together with feasibility studies for some chosen applications.
Literature survey on high lift and low drag wing planform in nature.

- Low order design using XFOIL, XFLR5 or AVL for design and optimisation of hydrofoil planform geometry
- Familiarisation with grid generation and numerical simulation tools
- StarCCM+ simulation optimal configurations

- Wind tunnel testing of optimal designs
- Analysis and validation of results

Prerequisites

High motivation for wing design and optimisation
Good grasp of basic aerodynamics or fluid mechanics
Low order viscous potential flow solvers such as XFOIL, XFLR5 or AVL
CFD simulation skills – grid generation, simulation and post processing (not compulsory)
Wind tunnel testing (not compulsory)
Python, Matlab or other tools for further data post-processing and analysis
Research orientated attitude

SPECTRAL SWP1-2025: OAD & High-fidelity aerodynamic analysis High Aspect Ratio Wing

Context: The SPECTRAL meta project aims at gathering all ISAE-SUPAERO projects related to medium or long range transport aircraft design. From some years, ISAE-Supaero & ONERA have been developing a preliminary design platform FAST-OAD-CS25 and current effort aims at enriching that design tool with models related to emerging technologies (H2, high aspect ratio, UHBR engines…).
AIRBUS ZEROe concept (2020)
This project is established in a context of a need for aviation to reduce its carbon foot-print. One way of achieving this goal is the reduction of drag by means of wings with higher aspect ratio wings. The proposed activity will be specifically targeted to the introduction of such technology. Deeper insight into Overall Aircraft Design challenges and expected performance of such a concept is expected from this work.
Objectives:

- State of the art on high aspect ratio wings, including identification of current challenges, appropriate simulation tools (with priority given to open-source ones), aeroelastic issues
- Identify specific aerodynamic effects due to high-aspect ratio wings technologies (strut-braced wing, upper/conventional wings…).
- Further developments of mesh generation methods, with possible parametrization of the CAD geometry.
- Running CFD on high aspect ratio configurations (conventional wings and SBW) with SU2 (and potentially with ADFlow).
- Methodologies will be then applied to a single-aisle airliner.

Pre-requisites
Overall Aircraft Design

- Transonic and low speed aerodynamics
- Structural airliner wing layout
- Notions of CFD (SU2, ADFlow, OpenFOAM…)
- Notions of CFD meshing (tool: PointWise)
- Python coding
- Catia v5
- Writing technical reports – code documentation
- Curiosity, autonomy

Angiology in Microgravity (AIM)

Description:
CONTEXT
Space exploration poses a threat to human health through factors such as reduced gravity loading, radiation and confinement. In 2019, a blood clot was identified in an astronaut's internal jugular vein by accident, during an experiment. Since then, several cases of reverse flow and stagnation patterns have been reported during medical check-ups. This problem raises serious concerns as the detachment of a blood clot could lead to a pulmonary embolism, a pathology that cannot be treated in space. Space agencies, including the European Space Agency, have thus established a task force to investigate this issue and understand how to prevent this phenomenon from happening during a long-term mission.
AIM
In this context, ESA Young Professionals are currently working on an experiment named Angiology in Microgravity (AIM), as part of the YPSat-2 program. This payload, which is the continuation of Artery in Microgravity, an experiment developed by students from ISAE-SUPAERO, aims to study haemodynamics in space conditions. It will investigate changes and disturbances in the internal jugular vein blood flow, as well as wall distention induced by long-term exposure to the space environment, with a specific focus on the deep vein thrombosis (DVT) pathophysiology. As DVT is considered one of the most critical risks associated with long-duration missions, it needs to be further investigated to facilitate the search for prevention plans and potential treatments. The payload, which should be launched into space late 2026 during the maiden flight of Space Rider, thus aims at understanding the blood clot formation process in microgravity. Moreover, the AIM experiment will serve as a technology demonstrator for the Particle Image Velocimetry (PIV) technique, that for the first time will be implemented in space to study blood flow dynamics. The research holds potential benefits for terrestrial medicine, as cardiovascular diseases stand as the leading global cause of death.
The payload will therefore be built in collaboration with ESA medical team and two institutions:

- The Biomedical Engineering Laboratory team of Politecnico di Torino provides expertise and experience in both designing and operating cardiovascular experiments;
- ISAE-SUPAERO is actively supporting the AIM Science Team with expertise and experience related to the design of the first iteration of the payload, and providing a Master's student (the author of this proposal) to support externally the project within the scope of his studies.

EXPERIMENTAL SETUP
The experimental setup aims to replicate the system in a simplified way. It includes a reservoir which will host the blood mimicking fluid (made of a water-based background and red blood cells replica), a peristaltic pump regulating the flow rate, the internal jugular vein phantom, a control valve regulating the pressure, as well as sensors required to collect meaningful data. To perform PIV, a high-speed camera will be placed at a right angle to the vein, which will be illuminated by a laser. A mirror will be used to direct the laser sheet perpendicularly to the cameras' field of view. The images obtained will be used to reconstruct the velocity path of the blood mimicking fluid particles. A low-speed camera will allow to obtain 2D measurement of sedimentation, as well as potential clotting or stagnation patterns, observed along the vein. LEDs will be used to ensure the camera has enough light to perform its duties.
TIMELINE AND TASKS
The payload just passed the Preliminary Design Review stage. The next targets are the Critical Design Review (CDR) scheduled for May 2025, the validation of the payload prototype planned in October 2025 and the Qualification of the Flight Model scheduled for March 2026.
YPSat-2 would like the author of this proposal to close some key knowledge gaps and help with the advancement of the payload towards its launch, while reinforcing the collaboration between the Agency and ISAE-SUPAERO, which designed and built the first iteration of AIM.
Under the supervision of Stéphanie Lizy-Destrez (Professor at ISAE-SUPAERO) and Gloria Gelosa (Head of the AIM Science Team at ESA), the student would be actively engaged in the following research activities (to be narrowed down based on the team's most urgent needs):

- Design of the fluid system and selection of system components, ensuring smooth and reliable operations, as well as precise data acquisition in a microgravity environment;
- Research state-of-the-art PIV setup and implementation of the most appropriate solution in order to observe blood flow dynamics in space conditions;
- Evaluation of the available techniques to analyze PIV data using image processing software, to measure flow velocities and identity potential stagnation patterns;
- Development of test plans to qualify the fluid system and PIV setup following ECSS standards, to guarantee the integrity of the payload and the flight vehicle;

- Prototype assembly, testing and potential troubleshooting (to be performed in ESA ESTEC laboratories during the summer of 2025, after passing the CDR stage);
- Analysis of fluid dynamics data obtained in the laboratory during the testing phase to quantify blood flow rates, pressure changes and red blood cell distribution.

POTENTIAL IMPACT AND APPLICATIONS

AIM addresses a critical research gap in astronauts' health and has substantial potential to improve our understanding of blood flow dynamics in microgravity. The interdisciplinary nature of this project, supported by ESA and academic collaborators, positions it as a valuable contribution to both space and terrestrial medicine. Beyond space exploration, AIM's findings may offer insights into cardiovascular pathologies on Earth. Understanding DVT risks in microgravity could lead to breakthroughs in blood clot prevention and management, providing new diagnostics or therapeutic approaches for patients with cardiovascular issues.

2024-2025: LOADS & HIGH LIFT DEVICES FOR OAD

Context:
The new HERA meta project aims at gathering all ISAE-SUPAERO projects related to regional type aircraft design. From some years, ISAE-Supaero has worked with ATR on hybrid regional aircraft and a specific aircraft design platform FAST-OAD-CS25/Reg has emerged from that work. It is very likely that this type of work will be extended through futures collaborations with ATR and other EU partners (new Clean Aviation project HERA, https://project-hera.eu/home).
Source: Aviationsourcenews.com
The purpose of the project is to explore the impact of high lift devices on the spectrum of loads to analyze for sizing a typical regional aircraft wing box. A key output from the project is to enrich the current level of load cases considered at OAD level, in particular for HERA configurations:
Objectives:

- Extensive state of the art on: (a) importance of flap loads on wing sizing, (b) methods used to determine loads, (c) range of flap performance & modelling (relevance of available database, DLR, UK NLF…), (d) transfer of loads on wing box structure;
- Second, to evaluate the existing mass model allocated to high lift devices implemented within the open source OAD platform (https://github.com/fast-aircraft-design/FAST-OAD) (CS25 & GA) and to propose an evolution of that model based on the literature review;
- Based on 2D performance (potential use of surrogate type approach, see SMT toolbox) and HERA geometry, to compute loads and associated sizing of the wing box and to evaluate the impact on FAST-OAD-CS25/Reg results;
- To summarize the methodology and findings into a competitive research paper, potentially shared with all contributors of the HERA research community.

Numerical analysis of the impact of the tip gap size on the performances of a centrifugal compressor stage

Description:
About the chair CASTOR:
Decarbonization requires some significant evolutions in the architecture of transportation systems. Many subsystems, such as the propulsive unit or the air conditioning systems, are impacted, and the turbomachinery involved in those subsystems needs to be adapted. In the electrification process, the efficient operating range requirement for turbines and compressors must be significantly increased compared with the current state of the art. The chair CASTOR is a research program of cooperation between Liebherr Aerospace, a recognized air conditioning system provider for the aeronautic industry, and ISAE-Supaero, who has acknowledged research activity in the off-design operation of radial turbomachinery. The main objective of the chair is to increase the operability range of centrifugal compressors and radial turbines. A research team of ISAE-Supaero faculty members, Liebherr experts,
Post-doctoral searchers, PhD students, Technical staff and students in internship will be dedicated to this ambitious four-year program.

Objective of the project:
The project focuses on the performance analysis of a centrifugal compressor stage. In the usual configuration, a tip gap is imposed between the blade tip and the fixed shroud to allow the impeller rotation. This tip gap is responsible for a tip leakage flow and a tip leakage vortex, which tend to reduce the efficiency of the machine and the operating range. One way to avoid tip leakage flow is to use a shrouded impeller. In this configuration, the tip gap is suppressed, and the shroud rotates with the blades. There are two objectives in this project. The first one is to numerically investigate both configurations, with and without tip gap to quantify the impact on the efficiency and the operating point. The second one is to understand the role of the tip leakage flow in the efficiency reduction of the centrifugal compressor stage.

Methodology:
The project will be based on numerical analysis. Therefore, 3D RANS and U-RANS simulations will be performed on different configurations having different tip gap sizes to allow a deep understanding of the impact of the tip leakage flow. The main parts of the project will consider the 3D mesh generation, the numerical set-up, and the 3D analysis of the steady and the unsteady simulation.