From PDEs to certified computational models: this is the motto of CARDAMOM. We aim at providing a robust model development methodology, as well as a quantitative approach to model certification, allowing to assess the robustness of the model w.r.t. each of its components (equations, numerical methods, etc), and to assess the variability of the outputs w.r.t. random variations of the data.

We will achieve this objective working toward a unified set of tools for the engineering analysis of complex flows involving moving fronts. Examples of such flows be found in civil, industrial, and aerospace engineering: industrial hazards (explosions), free surface hydraulics (coastal hydrodynamics, floods, etc), energy conversion facilities (gas-vapour, liquid-vapour systems, wave energy conversion, etc), space-reentry (chemically reacting fronts, ablating walls, rarefied/continuous flows), wing de-icing systems (ice-air flow), etc.

Simulating, optimising, and controlling these systems in a robust manner is far from being a simple task, especially in a real life. There is still a large number of open scientific challenges. These are related to the intrinsic nature of these flows necessitating:

- an appropriate PDE formulation taking into account the physics relevant to the engineering applications while remaining computationally affordable in an operational context
- efficient adaptive discretizations allowing to optimize the computational effort, while providing a sharp and accurate resolution of the physics
- a certification step quantifying the uncertainty in engineering outputs due to all modelling choices, both physical, and mathematical (continuous and discrete)

To develop a robust and accurate model means to be able to quantify and control the effects of the choices made in each of the above steps. The development of robust models tailored to the applications mentione above is the objective of CARDAMOM.

### Context

**Scientific priorities:** Modeling and Simulation

These simulations involve combining very different skills including mathematical analysis, digitalization, linear algebra and high-performance computing (HPC). This is a considerable challenge that must in particular exploit the tremendous advances in computational architectures.

**Scientific Research context:**

In-flight icing is an important problem for aviation safety. In addition to performance degradation due to ice accretion on the wings, icing can cause vibration or pumping of the engine, engine’s extinction, or even permanent degradation when blades are broken as a result of ice block detachment. These problems must be anticipated from the design phase of the aircraft. With the introduction of new, more stringent regulations and the use of new materials, manufacturers would like to have reliable numerical simulation tools available during the design and certification phase.

This is one of the most strategic research topics in the CARDAMOM team.

The goal of this work will be to continue the development of an efficient high fidelity numerical method allowing to compute ice block trajectories. The problem to be solved is a coupled problem since the airflow is modified by the motion of the ice block.
The initial work conducted in the FP7 STORM European project [1, 2], has led to the development of adaptive tools for the prediction of ice blocks shed into the air flow. These tools involve an immersed boundary method applied on unstructured adaptive meshes, and coupled with a level-set representation of the moving solids. Adaptive mesh deformation enables to refine the mesh at the fluid-solid interface. The potential of the proposed approach has been demonstrated in 2d for compressible laminar flows using linear continuous finite elements.

The objective of this proposal is to extend this method in 3d and to couple this approach with complex physical models/modules (turbulence models, ice accretion modules, etc). To be able to perform 3d simulations in a reasonable computational time, all the tools have to run in parallel. In particular, the adaptive mesh deformation should be couple in parallel with the immersed boundary method.

Assignment
The milestones of the project are the following:

1. Development of an unsteady scheme for 3d penalized laminar Navier-Stokes equations
2. Parallel coupling of the mesh deformation tool with the previous
3. Benchmarking on three dimensional tests involving complex geometries
4. Development of Spallart Almaras turbulence model
5. Coupling of these tools with this turbulence model

The development will be done in an object oriented parallel C++ platform (Aerosol) allowing both continuous and discontinuous finite element computations on general hybrid meshes.

Main activities
Keywords: CFD, immersed boundary methods, finite elements, unsteady mesh adaptation, fluid solid interaction

References:


Skills
Scientific programming, and in particular very high proficiency with object oriented programming in C++ ; numerical analysis, and in particular finite element methods ; basic notions of fluid mechanics

Benefits package
- Subsidised catering service
- Partially-reimbursed public transport

Remuneration
2653€ / month (before taxes)