RÉPUBLIQUE FRANÇAISE
Liberté
Égalité
Fraternité

2025 *Activity Report*

RESEARCH CENTRE: **Inria Centre at the University of Bordeaux**

IN PARTNERSHIP WITH: **CNRS, Université de Pau et des Pays de l'Adour**

Project-Team

CAGIRE

Computational AGility for internal flows sImulations and compaRisons with Experiments

In collaboration with Laboratoire de mathématiques et de leurs applications (LMAP)

Project-Team CAGIRE

Creation of the Project-Team: 2016 May 01

Each year, Inria research teams publish an Activity Report presenting their work and results over the reporting period. These reports follow a common structure, with some optional sections depending on the specific team. They typically begin by outlining the overall objectives and research programme, including the main research themes, goals, and methodological approaches. They also describe the application domains targeted by the team, highlighting the scientific or societal contexts in which their work is situated. The reports then present the highlights of the year, covering major scientific achievements, software developments, or teaching contributions. When relevant, they include sections on software, platforms, and open data, detailing the tools developed and how they are shared. A substantial part is dedicated to new results, where scientific contributions are described in detail, often with subsections specifying participants and associated keywords. Finally, the Activity Report addresses funding, contracts, partnerships, and collaborations at various levels, from industrial agreements to international cooperations. It also covers dissemination and teaching activities, such as participation in scientific events, outreach, and supervision. The document concludes with a presentation of scientific production, including major publications and those produced during the year.

Keywords

Computer sciences and digital sciences

A6.1.1. – Continuous Modeling (PDE, ODE)

A6.2.1. – Numerical analysis of PDE and ODE

A6.2.7. – HPC for machine learning

A6.5.2. – Fluid mechanics

Other research topics and application domains

B2. – Digital health

B4. – Energy

B4.2. – Nuclear Energy Production

B5.2.1. – Road vehicles

B5.2.3. – Aviation

B5.2.4. – Aerospace

Contents

1 Team members, visitors, external collaborators

Research Scientists

- Rémi Manceau [Team leader, CNRS, Senior Researcher, HDR]
- Pascal Bruel [CNRS, Researcher, until Mar 2025, HDR]
- Vincent Perrier [INRIA, Senior Researcher, HDR]
- Kevin Schmidmayer [INRIA, ISFP]

Faculty Member

- Jonathan Jung [UPPA, Associate Professor]

Post-Doctoral Fellows

- Puneeth Bikkanahally Muni Reddy [UPPA, Post-Doctoral Fellow, until Sep 2025]
- Anthony Bosco [UPPA, until Feb 2025]
- Daniel Eduardo Inzunza Herrera [UPPA, Post-Doctoral Fellow, from May 2025]
- Mandeep Saini [UPPA, Post-Doctoral Fellow, from Jun 2025]

PhD Students

- Adedotun Ade [INRIA]
- Anju Albi Lilly [UPPA, until Mar 2025]
- Qa'Im Bekkali [UPPA, from Nov 2025]
- Esteban Coiffier [CEA, until Nov 2025]
- Josephine Gauthier [EDF R D, from Oct 2025]
- Jules Mazaleyrat [SAFRAN, CIFRE]
- Corina Sanz Souhait [EDF, CIFRE]
- Romaric Simo Tamou [UPPA, until Mar 2025]
- Felice Taglialatela [UPPA, from Oct 2025]

Technical Staff

- Alexis Altolaguirre [UPPA, Engineer, from Oct 2025]
- Abderrahman Benkhalifa [UPPA, until Sep 2025]
- Luca Cirrottola [INRIA, Engineer]
- Daniel Eduardo Inzunza Herrera [INRIA, Engineer, from Feb 2025 until Apr 2025]
- Lucas Martin De Fourchambault [UPPA, from Oct 2025]

Interns and Apprentices

- Fatima Bouhenni [CEA, Intern, from Mar 2025 until Aug 2025]
- Lucas Martin De Fourchambault [CEA, Apprentice, until Sep 2025]

Administrative Assistants

- Fabienne Cuyollaa [INRIA]
- Nathalie Robin [INRIA]

External Collaborator

- Matthieu Haefele [CNRS]

2 Overall objectives

The project-team CAGIRE is an interdisciplinary project, which brings together researchers with different backgrounds (applied mathematics and fluid mechanics), who elaborated a common vision of what should be the *numerical simulation tools in fluid dynamics* of tomorrow. The targeted fields of application are mainly those corresponding to the aeronautical/terrestrial transportation and energy production sectors, with particular attention paid to the issue of energy transition and the reduction of environmental impacts. This panel has been extended to medical applications recently, where numerical simulation plays an increasingly important role. Through our numerous industrial collaborations, we have been able to refine our vision of the future of numerical simulation, which is subject to ambitious industrial objectives, constant evolution of computing resources and increasingly present environmental constraints.

The flows under consideration involve many physical phenomena: they can be turbulent, compressible, multiphase, anisothermal. Even if these phenomena are not necessarily present at the same time, our strategy for developing models and numerical schemes must take them into account. Turbulence plays a central role insofar as it is a dimensioning constraint for CFD in most industrial configurations. It is indeed the comparison of the requirements in terms of scale of description, numerical accuracy and computational cost that guides the choice of physical models and numerical methods.

Because such flows are exhibiting a multiplicity of length and time scales resulting from complex interactions, their simulation is extremely challenging. Even though various simulation approaches are available and have significantly improved over time, none of them does satisfy all the needs encountered in industrial and environmental configurations. We consider that different methods will be useful in the future in different situations, or regions of the flow if combined in the same simulation, in order to benefit from their respective advantages wherever relevant, while mutually compensating for their limitations. For instance, for turbulent flows, it will thus lead to a description of turbulence at widely varying scales in the computational domain. The RANS[1] method may cover regions where turbulence is sufficiently close to equilibrium, leaving to LES[2] the regions where the RANS description is insufficient, leadind to a hybrid RANS-LES approach. Similarly, for two-phase flows, one of the greatest challenges is to be able to tackle simultaneous and dynamical modelling of the multi-scale features and their transition, e.g., from cavitation pockets to tiny bubbles. The models and numerical methods must also be flexible enough to accurately represent all the above-mentioned phenomena in complex geometries, with efficient and robust resolution algorithms to preserve an optimal computational cost. It is this flexibility and adaptability of models and numerical methods that we call “computational agility", which is in the title of the CAGIRE team: Computational AGility for internal flow sImulations and compaRisons with Experiments.

Therefore, the long-term objective of this project is to develop, validate, promote and transfer original and effective approaches for modeling and simulating generic flows representative of configurations encountered in applications, in various fields, such as transportation, energy production and medicine. In order to progress in this direction, many building blocks have to be assembled, which motivates a variety of research topics described in the following sections and divided into four main research axes. The topics addressed, ranging from advanced physical modelling to high-order numerical discretization, require the multi-disciplinary skills that constitute the CAGIRE project-team:

- Turbulence modelling
- High-order numerical methods and efficient algorithms

[1] Reynolds-Averaged Navier-Stokes
[2] Large-Eddy Simulation

- Compressible and multiphase flows
- Analysis and simulation of turbulent flows and heat transfer

3 Research program

3.1 Turbulence modelling

In the “agile" simulation methods introduced above, a flexible representation of turbulence is essential: in the same simulation, depending on the regions of the flow, it is necessary to be able to switch from a fine-grained to a coarse-grained representation of turbulence. Numerous methods, called hybrid RANS/LES, go in this direction, by associating LES and RANS. In order to ensure such a flexibility, it is preferable not to rely on a preliminary partition of the domain (the so-called *zonal* approach), but rather on a continuous transition from one model to the other (the so-called *continuous* approach).

Various questions then arise: how can we improve the RANS models so as to accurately represent most of the physical phenomena in order to avoid having to switch to LES in large regions; how to play on the terms of the models, and on which criteria, to switch from RANS to LES; how to improve the robustness of the method with respect to the choices made by the user (in particular the mesh). Our research work, described below, aims at answering these questions.

Today, even though the industrial demand for more accurate and robust RANS models is very significant, very few academic teams are active in this field (for instance, [129, 95, 62, 136]), most of them being participants to the European ERCOFTAC SIG-15 group of which we are a coordinating member. In France, we collaborate or have recently collaborated with most of the teams, mainly in the industry (EDF, Dassault, PSA, SAFRAN) or applied research organizations (ONERA, CEA). The CAGIRE team is particularly renowned for its work on the interaction between turbulence and the wall by elliptic blending (EB-RSM, [106, 109]), and is solicited by these partners to improve the representation of complex effects on turbulence (buoyancy, conjugate heat transfer, adverse pressure gradients, impingement, *etc.*).

Concerning the development of original hybrid RANS/LES approaches, the main contributions in France are due to ONERA (ZDES [77] and PITM [73]); IMF Toulouse in collaboration with the ECUADOR team of the Inria center of Sophia-Antipolis (OES [70, 116]) and CAGIRE (HTLES [104, 61, 85, 68]). The originality of our work is two-fold: (i) through temporal filtering, a formally consistent link is provided between the equations of motion and the hybridization method in order to reduce the level of empiricism, which is, for non-homogeneous turbulence, along with the additive filter method [90, 60], one of only two methods capable of providing such a consistent framework; (ii) through the development of an *active* approach based on the Anisotropic Linear Forcing (ALF) [49] and an *adaptive* strategy that autonomously determines the LES zone and refines the mesh based on physical criteria [55], a new *Continuous Embedded LES* paradigm is proposed, which is a realisation of the *agility* concept at the center of our project.

3.2 High-order numerical methods and efficient algorithms

When dealing with RANS models, a second order finite volume method is usually used. In our project, we aim at addressing hybrid RANS/LES models, which include some regions in which essentially unstationary processes are approximated in LES regions. This usually requires to use low dissipative high order numerical methods. If a consensus has emerged for years on second order finite volume methods for the approximation of RANS models, investigations are still ongoing on finding the high order method that would be the best suited with the compressible Navier-Stokes system.

As far as high order numerical methods are concerned, they are addressed at Inria essentially by the Atlantis, Makutu, Poems and Rapsodi teams for wave-matter interaction problems, the Serena and Coffee project-team on porous media, the Tonus team on plasma physics problems, and the Acumes, Gamma, Cardamom and Memphis teams for systems that are closer of ours (shallow-water or compressible Euler). As far as we know, only the Cardamom and Gamma teams are using high order methods with turbulence models, and we are the only one to aim at hybrid RANS/LES models with such methods.

Our objective is to develop a fast, stable and high order code for the discretization of compressible Navier-Stokes equations with turbulence models (Reynolds-stress RANS models and hybrid RANS/LES methods) on unstructured meshes. From a numerical point of view, this raises several questions: how to derive a stable numerical scheme for shocks without destroying the order of accuracy, how to derive stable boundary conditions, how to implement the method efficiently, how to invert the system if implicit methods are used?

Concerning aeronautical applications, several groups are working on discontinuous Galerkin methods: in Europe, some of the groups participated to the TILDA project [3] (DLR, ONERA, CERFACS, Imperial College, UCL, Cenaero, Dassault, U. of Bergamo). As far as we know, none of them considered Reynolds-stress RANS models or hybrid RANS/LES models. Worldwide, we believe the most active groups are the MIT group [4], or Ihme's group[5] which is rather oriented on combustion. Concerning HPC for high order methods, we carefully follow the advances of the parallel numerical algorithm group at Virginia Tech, and also the work around PyFR at Imperial College. Both of these groups are considering imperative parallelism, whereas we have chosen to consider task based programming. Task based parallelism was considered in the SpECTRE code [102] based on the Charm++ framework, and within a European project[6], based on IntelTBB, but only for hyperbolic systems whereas we wish to address the compressible Navier-Stokes system.

3.3 Compressible and multiphase flows

In this section, we are interested in two specific regimes of compressible flows: low Mach number flows and compressible multiphase flows.

Low Mach number flows (or low Froude for Shallow-Water systems) are a singular limit, and therefore raise approximation problems. Two types of numerical problems are known: if convective time scales are considered, semi-implicit time integration is often preferred to explicit ones, because the acoustic CFL is very restrictive compared with the convective one in the low Mach number limit [78]. The second numerical problem at low Mach number is an accuracy problem. The proposed fixes consist in changing the numerical flux either by centering the pressure [122] or are variant of the Roe-Turkel fix [91]. Over the last years, we have been more focused on the accuracy problem, but our major originality with respect to other groups is to be interested in the acoustic wave propagation in low Mach number flows, which may also raise problems as first remarked in [115].

Understanding and controlling complex and physically rich flows, such as unsteady multiphase compressible flows, is of great importance in various fields such as aeronautics, automotive, aerospace, nuclear energy, naval and also medicine. If we note the efforts established so far to partially respond to the problems linked to these flows, we also note major remaining challenges, particularly when different spatial and temporal scales or multiple physical phenomena, such as phase change, viscoelasticity or more generally interactions with solids, are to be considered. Good examples are cavitating flows such as the ones encountered around naval propellers where cavitation pockets form at the vicinity of the blades and lead to a turbulent bubbly flow in the wake [123]. Or in biomedical applications such as in lithotripsy (treatment for kidney stones) [120] or, recently, histotripsy (non-invasive treatment for cancers) [101] where cavitation bubbles, induced by shock waves, laser energy deposit or high-intensity focused ultrasound waves, violently collapse and interact with biomaterials. In this context, we aim to tackle the particularly challenging and ambitious modelling of these extremely complex multiphase compressible flows where numerous scientific and technical obstacles remain to be overcome. Among them, we could cite:

- The modelling of multiscale features including the simultaneous and dynamical computation of sub-grid dynamics (inclusions such as bubbles or drops) and of resolved interfaces. The derivation of averaged compressible multiphase models is currently less active than in the 2000s, and only few teams are interested in such problems. Recent advances were made at RWTH [93], and also mostly in France at EDF R&D by J.M. Hérard or also by Bresch and Hillairet [72]. This low interest in this type of challenging modeling and mathematical analysis was noticed in the review paper [126] as an obstacle for the improvement of numerical methods. Hence, the driving idea of this project to focus our efforts on the modeling of subscale phenomena, in particular by a stochastic process [118].
- The modelling of biomaterials under a fluid-mechanics formulation including viscoelastic behaviour and realistic equations of state, and the modelling of bubbles containing simultaneously condensable

[3] https://cordis.europa.eu/project/id/635962
[4] https://www.gas-turbine-lab.mit.edu/
[5] https://profiles.stanford.edu/werner-ihme
[6] https://data.mendeley.com/datasets/6sz8h6hnpz/1

and non-condensable gases. The simultaneous coupling of compressible, multi-component flow models with viscoelastic solids and mass transfer will enable us, through simulations, to understand the fundamental physics taking place in several medical applications involving bubble dynamics [71]. This will therefore fill the knowledge gap on the subject involving significant range of physical phenomena that are not well understood yet, and for which experiments often lack insight, and spatial and temporal resolution [67]. This will potentially lead to significant improvements of the current and future medical treatments regarding their success rate, cost and safety.

3.4 Analysis and simulation of turbulent flows and heat transfer

The numerous discussions with our industrial partners make it possible to define configurations to carry out comparison between computations and experiments aimed at validating the fundamental developments described in the previous sections. Reciprocally, the targeted application fields play an important role in the definition of our research axes, by identifying the major phenomena to be taken into account. This section gathers applications which essentially deal with turbulent internal flows, most often with heat transfer.

Detailed data are required for a fine validation of the methods. In addition to the active participation and co-organizing of the SIG-15 group of the ERCOFTAC network, which gives us access to various experimental or DNS data and enables us to carry out model and code benchmarking exercises with other European teams [103, 105, 65, 108], we generate experimental data ourselves when possible and develop collaborations with other research groups when necessary (ONERA, institute Pprime, CEA [11], ETH Zurich [28]).

Historically, the scientific convergence between the team members that led to the development of our project and the creation of the CAGIRE project-team in 2016 was based on scientific themes related to aeronautical combustion chambers (hence the term *internal flows* in the name of the team), with our industrial partners SAFRAN and Turbomeca (now SAFRAN-Helicopter Engines). If the scientific and application themes of the team are now much more diverse, these applications to aeronautical combustors are at the origin of the existence of the MAVERIC experimental facility, allowing the study of turbulent flows at low Mach number over multi-perforated walls subjected to a coupling with acoustic waves, representative of the flows in combustors. This wind tunnel is thus complementary to those developed at ONERA, with which we collaborated [121] when it was necessary to add thermal measurements, within the framework of the European project SOPRANO.

4 Application domains

4.1 Aeronautics

Cagire is active in the field of aeronautics through the following activities:

- The combustion chamber wall: the modelling, the simulation and the experimentation of the flow around a multiperforated plate representative of a real combustion chamber wall have been focused on during the recent period. The continuous improvement of our in-house test facility Maveric is also an important ingredient to produce our own experimental validation data for isothermal flows. For non-isothermal flows, our participation in the EU funded program Soprano gave us access to non-isothermal data produced by Onera. This activity was also included in the recently finished E2S-UPPA project Asturies.

- Impinging jets: because of their high heat transfer efficiency, turbulent impinging jets are commonly used in a large variety of applications, and in particular blade cooling systems. Understanding the underlying physics of the mechanisms at play is of prime interest and is still an open question. Additionally, this configuration remains a challenging test case for turbulence models since it embraces many flow features despite a relatively simple geometry, and causes strong discrepancies between standard turbulence closures. Reynolds stress transport models have been shown to be promising candidates but still suffer from a lack of validation regarding this flow configuration. Such models are the subject of a collaboration with Onera and SAFRAN HE (CIFRE PhD thesis of Jules Mazaleyrat).

4.2 Energy

- The prediction of heat transfer in fluid and solid components is of major importance in power stations, in particular, nuclear power plants. Either for the thermohydraulics of the plenum or in the study of accidental scenarii, among others, the accurate estimation of wall heat transfer, mean temperatures and temperature fluctuations are necessary for the evaluation of relevant thermal and mechanical design criteria. These problems are addressed in the framework of a long term collaboration with EDF, started in 2014, leading to the development of innovative RANS models for these industrial applications [110, 59], pursued within the ANR project MONACO_2025 and via the ongoing CIFRE PhD thesis of Corina Sanz Souhait. In addition, one of the problems that has marked the recent history of the nuclear sector is the issue of stress corrosion cracking, which led to the shutdown of many nuclear power plants between 2021 and 2023. This problem is due to secondary flows generated in dead branches of the primary circuit, which cause hot and cold fluids to mix, leading to thermal fatigue, among other things. As part of Joséphine Gauthier's CIFRE thesis, which began in 2025, we are developing decomposition and domain coupling approaches to resolve the key issue of the very different time scales (from seconds to days) governing the various phenomena involved.

- Moreover, the prediction of unsteady hydrodynamic loadings is a key point for operating and for safety studies of PWR power plants. Currently, the static loading is correctly predicted by RANS computations but when the flow is transient (as, for instance, in Reactor Coolant Pumps, due to rotor/stator interactions, or during operating transients) or in the presence of large, energetic, coherent structures in the external flow region, the RANS approach is not sufficient, whereas LES is still too costly for a wide use in the industry. This issue was the main focus of the PhD thesis (CIFRE EDF) of Vladimir Duffal, and pursued within the ANR project MONACO_2025 (PhD of Puneeth Bikkanahally).

- Thermal storage is interesting to decorrelate the production of heat or cold from its use whether for direct operation for a heat network (smoothing of heat supply to meet intermittent needs) or for power generation (phase shift between heat generation and power generation). The challenge is to study, via CFD, the dynamic and thermal behavior of the storage during the loading, resting and discharge phases. This was the focus of the PhD thesis of Alexis Ferré, co-supervised by R. Manceau and S. Serra (LaTeP). This work is pursued through the collaboration with a post-doc at CEA Grenoble.

4.3 Automotive propulsion

- The engine (underhood) compartment is a key component of vehicle design, in which the temperature is monitored to ensure the effectiveness and safety of the vehicle, and participates in 5 to 8% of the total drag and CO2 emissions. Dimensioning is an aerodynamic and aerothermal compromise, validated on a succession of road stages at constant speed and stopped phases (red lights, tolls, traffic jam). Although CFD is routinely used for forced convection, state-of-the-art turbulence models are not able to reproduce flows dominated by natural convection during stopped phases, with a Rayleigh number of the order of 10^{10}, such that the design still relies on costly, full-scale, wind tunnel experiments. Since the ambition of the PSA group is to reach a *full digital design of their vehicles*, i.e., to almost entirely rely on CFD, this issue was the focus of the PhD thesis (CIFRE PSA) of S. Jameel, supervised by R. Manceau, and also a part of the ANR project MONACO_2025, in the framework of which S. Jameel and S.K. Jena were hired as post-docs. This application is currently seeking funding to continue this work.

- The Power & Vehicles Division of IFPEN co-develops a CFD code, CONVERGE, to simulate the internal flow in spark-ignition engines, in order to provide the automotive industry with tools to optimize their design. The RANS method, widely used in the industry, is not sufficiently reliable for quantitative predictions, and is only used as a tool to qualitatively compare different geometries. On the other hand, LES provides more detailed and accurate information, but at the price of a CPU cost unafordable for daily use in the industry. Therefore, IFPEN aims at developing the hybrid RANS/LES methodology, in order to combine the strengths of the two approaches. The PhD thesis of Hassan Afailal, co-supervised by Rémi Manceau, was focused on this issue. In the framework of the collaborative project ASTURIES (E2S-UPPA/Inria/CEA/IFPEN), this collaboration with IFPEN is pursued by the development of high-order methods in the CONVERGE code in order to make it possible to perform highly accurate and low-dissipative LES and hybrid RANS/LES in combustion engines.

4.4 Medical applications

Many medical applications exist where interactions between bubbles and biomaterials appear. CAGIRE is interested in a better understanding of the fundamental physics involved in such interactions, leading to improvements and innovation in current and future medical treatments with regard to their success rate, cost and safety:

- Lithotripsy is a noninvasive (the skin is not pierced) procedure used to treat kidney stones that are too large to pass through the urinary tract. Lithotripsy treats kidney stones by sending focused ultrasonic energy or shock waves directly to the stone first located with fluoroscopy (a type of X-ray "movie") or ultrasound (high frequency sound waves). The shock waves break a large stone into smaller stones that will pass through the urinary system. Lithotripsy allows persons with certain types of stones in the urinary system to avoid an invasive surgical procedure for stone removal. Lithotripsy involves cavitation bubbles as a primary or secondary mechanisms to attack the surface of the stone. Regarding success rates, for patients who are thought to be good candidates for this treatment, about 70 to 90 percent are found to be free of stones within three months of treatment. Furthermore, one should note that lithotripsy may include, but is not limited to, complications such as: bleeding around the kidney, infection, obstruction of the urinary tract by stone fragments, stone fragments left that may require more lithotripsies.
- Histotripsy is the first noninvasive, non-ionizing, and non-thermal ablation technology guided by real-time imaging. Using focused ultrasound delivered from outside the body, histotripsy mechanically destroys tissue through cavitation, rendering the target into acellular debris. The material in the histotripsy ablation zone is absorbed by the body within 1-2 months, leaving a minimal remnant scar. Histotripsy has also been shown to stimulate an immune response and induce abscopal effects in animal models, which may have positive implications for future cancer treatment. Histotripsy has been investigated for a wide range of applications in preclinical studies, including the treatment of cancer, neurological diseases, and cardiovascular diseases. Phase I human trials have shown the initial safety and efficacy of histotripsy to treat patients with malignant liver tumors, BPH, and calcified aortic stenosis. Despite substantial technical, preclinical, and clinical progress to date, there is a large amount of future work necessary for technical development, preclinical research, and human studies before histotripsy can become a wide-spread clinical treatment modality.
- Drug delivery. Gas-filled microbubbles can be designed with drug- and gas-loaded interiors. A stabilizing coating surrounds the bubble which may be targeted to specific tissue by incorporating protein ligands on the surface. Drugs can be incorporated by themselves or, if insoluble in water, in an oil layer. Among the possible therapies, the most exciting is the possibility of the delivery of genetic material to a chosen site. Focused ultrasound is then used to cavitate the gene-loaded microbubble and the shockwaves or microjets thus generated cause the genetic material to be injected into the surrounding cells. This technology can also be combined with endothelial cell barrier opening, which is performed prior the drug delivery. Indeed, the ultrasound irradiation of microbubbles produces jets which open the barrier for a few hours. This promotes the passage of large drug molecules necessary for specific treatments such as that of Alzheimer's disease.

4.5 Defence

- CAGIRE through its collaboration with the CEA Gramat and Aix-Marseille Université is particularly active on the numerical simulation of the mitigation of explosion effects (detonation in a heterogeneous material) using aqueous foam. Indeed, the confinement of an explosive using dry aqueous foams is capable of limiting the destructive effects of detonation in terms of shock waves and blast waves. More recently, the use of foam has also demonstrated the potential to capture micro- and millimeter-sized particles. To conduct more precise analyses of the impact of confinement, such as slowing down or capturing particles, numerical simulation of the phenomenon is required. This is where the ECOGEN code, co-developed with partners, comes into play.

- Atmospheric reentry problem: When a body enters the atmosphere with a high velocity, its trajectory is mainly driven by the hypersonic flow surrounding the body. The integrity of the body is maintained by a shield that is progressively ablated. The sharp control of the motion is possible with a very good knowledge of the surrounding hypersonic flow and of its interaction with the ablated shield. In the Asturies project, the aim was to developed numerical approaches able to deal with these flows. The just-accepted ANR ASTRID project is a continuation of this work.

5 Social and environmental responsibility

Impact of research results

The availability of improved RANS models and hybrid RANS/LES methods offering a better physical representativeness than models currently used in the industry, at a reasonable computational cost, will make it possible to improve the reliability of industrial numerical simulations, and thus to better optimize the systems, in order to reduce the environmental impact of transportation and industrial processes, and to improve the safety of installations and reduce the risks of accidental pollution.

Moreover, previous applications of hybrid RANS/LES methods have shown that it is possible to obtain an accuracy equivalent to LES with a number of grid cells reduced up to a factor of 200. This gain can be considerably increased in a complete industrial simulation with a much higher Reynolds number, leading to a drastic reduction of the environmental impact of the simulations themselves.

6 Highlights of the year

- After a fruitful career dedicated to the understanding, modelling and simulation of turbulence, compressible flows and combustion, Pascal Bruel retired in 2025. Pascal was at the origin of the creation of the CAGIRE team that he leaded until 2019. The team members are indebted to him for his unifying role and scientific influence.

- The flagship RANS model developed by the team, the EB-RSM, is now available in the last version (6.4) of the commercial software COMSOL multiphysics, released in November 2025. After the StarCCM+ and EZNSS commercial codes, EDF's code_saturne (open-source), Dassault Aviation's AETHER, ONERA's CEDRE, and the widely used open-source code OpenFOAM, this marks a new stage in the widespread dissemination of this model towards industry and the scientific community.

- Through the ANR JCJC MSBUB project, which began in 2025, and the Franco-German ANR PRCI DreamTurbulence project, which will begin in April 2026, the CAGIRE team is opening up a new area of research: the enrichment of models (two-phase and turbulence) through machine learning, in particular through symbolic regression approaches.

- At the end of his first PhD year, in October, Felice Edoardo Taglialatela officially joined the team to collaborate with us on numerical methods and turbulence simulation, in particular with Large Eddy Simulations (LES) models, for multiphase compressible flows. He is a PhD student in fluid dynamics, funded by University of Campania "L. Vanvitelli" through the Italian Ministry of University and Research. His supervisors are Prof. Giuliano De Stefano (full professor at University of Campania "L. Vanvitelli") and Kevin Schmidmayer.

- Within the context of the collaboration with the CEA Gramat and Aix-Marseille Université on the numerical simulation of the mitigation of explosion effects using aqueous foam. Lucas Martin de Fourchambault succefully completed his Master apprenticeship under the supervision of Kevin Schmidmayer and is starting his PhD in January 2026, funded by CEA Gramat and supervised by Fabien Petitpas (Aix-Marseille Université), Kevin Schmidmayer and Maxime Reynaud (CEA Gramat).

- The ANR ASTRID project called HYPERSONICS was accepted in 2025. The project aims to develop innovative, robust, and accurate numerical methods for simulating the aerodynamics of hypersonic vehicles. The project partners are the Institute A. Grothendieck of Montpellier (IMAG), the Mathematics Institute of Bordeaux (IMB), the Laboratory of Mathematics and its Applications of Pau (LMAP) and the CEA Cesta.

7 Latest software developments, platforms, open data

7.1 Latest software developments

7.1.1 AeroSol

Keywords: High order finite elements, Parallel computing

Functional Description: The AeroSol software is a high order finite element library written in C++. The code has been designed so as to allow for efficient computations, with continuous and discontinuous finite elements methods on hybrid and possibly curvilinear meshes. The work of the team CARDAMOM (previously Bacchus) is focused on continuous finite elements methods, while the team Cagire is focused on discontinuous Galerkin methods. However, everything is done for sharing the largest part of code we can. More precisely, classes concerning IO, finite elements, quadrature, geometry, time iteration, linear solver, models and interface with PaMPA are used by both of the teams. This modularity is achieved by mean of template abstraction for keeping good performances. The distribution of the unknowns is made with the software PaMPA , developed within the team TADAAM (and previously in Bacchus) and the team Castor.

News of the Year: Highlights for the year 2025 concern:

- Subcell limiting (Alessandro Del Piero)

- Coupling with mesh adaptation, computation of metrics (e.g. Hessian) time-dependent mesh adaptation (Dean Yuan)

- wet-dry handling for shallow water models (Dean Yuan)

- Improvements of PETSc usage (Filipe Forte Tenreiro)

- Refactorization of ICBC allocation in examples (Luca Cirrottola)

- Integration of functional test cases for Axisymmetric problems (Anthony Bosco, Vincent Perrier)

- documentation framework improvement (doxygen) (Luca Cirrottola)

- Maintenance and imrpovements of CMake compilation files and guix packaging (Luca Cirrottola)

- Preliminary tests on code coupling with OASIS (Andrea Filippini)

- Adaptation of the master branch to C++11 (Luca Cirrottola)

- Preparation of DM2 integration (Luca Cirrottola)

- Computation of spurious modes in low Mach number flows with pressure-centred fix (Ibtissem Lannabi)

- Development of code formatting rules and integration with CLANG (Luca Cirrottola)

- Several bug fixes (CI, functional tests, unit tests).

- Wiki and documentation improvement.

Contribution statistics: About 300 commits this year, organized in 16 merge requests that were opened and merged into the master branch this year.

URL: `https://team.inria.fr/cardamom/aerosol/`

Contact: Vincent Perrier

Participant: 11 anonymous participants

7.1.2 DM2

Name: Distributed Mesh and Data Manager

Keywords: HPC, Data parallelism, High order finite elements, Unstructured meshes, Hybrid meshes

Functional Description: DM2 is a C++ library for managing mesh and data on mesh in a MPI parallel environment. It is conceived to provide parallel mesh and data management in high order finite element solvers for continuum mechanics.

The user should provide a mesh file which is read by the library. Then DM2 is able to:

- Read the mesh, and read the data provided in the mesh file, possibly in parallel

- Redistribute the mesh in order to distribute the data on a given set of processors. This redistribution is made through a graph partitioner such as PARMETIS or PT-SCOTCH.

- Allocate the memory in parallel if a number of unknown by entity type is provided by the user.

- Centralize the data.

- Compute the halo required for a numerical method. The halo is adapted for each of the possible discretization.

- Renumber mesh elements for making a difference between mesh elements that need or need not communication.

- Aggregate a mesh based on a metric for developing a multigrid method.

Release Contributions: This version introduces overlap regions ("halos") among distributed mesh partitions. These halos are specialized for discontinuous or continuous schemes, but generic with respect to the (geometric) degree of the mesh cells. These halos allow to synchronize numerical data defined on a set of entities of the distributed mesh. Numerical data is again generic with respect to the degree of their polynomial approximation, the number and combinations of scalar/vector fields, and the size of the vector spaces.

News of the Year: Highlights for the 2025 years:

- Installation of DM2 as a standalone library.

- Library API.

- A specific *compound graph* with local and halo entities for each type of discretization.

- Use namespaces instead of type prefixes.

- Checkpoint restart.

- Vector NetCDF support.

- Code formatting checks with Clang.

- New Docker CI instead of the Cloudstack VMs.

- Refactorings: VTU, iterators, halo graph storage, template graph class, graph constructors, remove legacy code.

- Maintenance: Docker images, Guix time-machine, CMake.

Contributions statistics: about 31 merge requests that were merged into the master branch during the last year.

Contact: Vincent Perrier

Participant: 4 anonymous participants

7.1.3 UHAINA

Keywords: Simulation, Ocean waves, Unstructured meshes, Finite element modelling

Scientific Description: Operational platform for near shore coastal application based on the following main elements:

- Fully-nonlinear wave propagation.

- Wave breaking handled by some mechanism allowing to mimic the energy dissipation in breakers.

- A high order finite element discretization combined with mesh and polynomial order adaptation for optimal efficiency.

- An efficient parallel object oriented implementation based on a hierarchical view of all the data management aspects cared for by middle-ware libraries developed at Inria within the finite element platform Aerosol.

- A modular wrapping allowing for application tailored processing of all input/output data (including mesh generation, and high order visualization).

- Spherical coordinates based on a local projection on a real 3D spherical map (as of 2021)

- Compilation with GUIX available (as of 2022)

- Homogenization and standardization of code outputs and hazard quatification (as of 2022)

- Correction of the management of dry/wet fronts in the presence of structures represented by a single high point (as of 2022)

- Use of FES for the calculation of the tide directly in UHAINA through an API. New compilation option for activation (as of 2022)

- Boundary conditions accounting tides from FES and corrected with the effect of the inverse barometer, for the simulation of the tidal propagation and the surge on domains at the regional scale (as of 2022)

- Hydraulic connections (e.g. sewers) in the simulation of urban flooding (as of 2022)

- Mass source term, for the injection of the volume of water overtopping structures not accounted in the elevation model during flooding episodes by sea surges (as of 2022)

Functional Description: Waves simulation

Contact: Mario Ricchiuto

Participant: 4 anonymous participants

Partners: EPOC, IMAG, IMB

7.1.4 ECOGEN

Name: Evolutive Compressible Flows Open Source Genuine Easy and N-phase

Keywords: Computational Fluid Dynamics, Compressible flows, Compressible multimaterial flows, Mesh adaptation, Unstructured meshes, Finite volume methods, 3D

Scientific Description: https://doi.org/10.1016/j.cpc.2019.107093

Functional Description: ECOGEN is a CFD platform dedicated to the numerical simulation of compressible multiphase flows. ECOGEN offers features such as: - Multi-model option (single-phase, multi-phase with or without equilibrium), so you can choose the model best suited to the physical phenomenon you're going to simulate. - Multi-physics option (heat transfer, viscosity, surface tension, mass transfer), allows you to add physical phenomena or effects to be considered in the simulation. - Multi-mesh option (Cartesian, unstructured, AMR), to simulate the phenomenon with different levels of precision and performance (calculation time) and adaptability to the geometry of the object on which the fluids are flowing. - HPC.

Release Contributions: A new release, ECOGEN_v5.0, is available on GitHub. It includes new features and fixes bugs.

Major points - Possibility to use second-order numerical scheme on unstructured meshes. Although still in beta. - Acoustic waves can now be modelled through source terms (sinusoidal or Gaussian pulses). This follows the work of _Maeda & Colonius_ [https://doi.org/10.1016/j.wavemoti.2017.08.004](https://doi.org/10.1016/j.wav for single-phase flows and has been extended here for multiphase systems of equations. - For developers: Improvement of the CI now including code coverage, clang-formatting, better artefact management, use of address sanitizer and pre-commit. - Clang-format applied to the whole code.

Minor points - Gnuplot scripts added for pdf generation of results with GNU output format - Add velocity in x-direction to the 1D output using Gnuplot. - Correction of translation: restart -> resume. - The CFL length estimation has been improved for quadrangle elements. - Possibility to add record frequency for cuts. - Correction of memory errors (deallocations). - Possibility to compile the code using ‘cmake‘ instead of ‘makefile‘ (for advanced users). - Update of documentation and test cases, as usual. - Use ‘VTK‘ key instead of ‘XML‘ one for ‘outputMode‘ tag of ‘main.xml‘ file. - Modify the deepness of all 1D and 2D test cases to unity. This was made to avoid possible issues during the visualisation of the results with tools such as Paraview.

Fixes - Update names of manuals to be coherent with input files from the test cases. - Update Cv from air EOS. - Fix bug and variables names update on ‘ModUEq::solveRIemannInletInjTemp‘. - Fix EOS assignment in boundary conditions and add that the EOS key must be present in ‘dataFluid‘ tag of boundary conditions (must be consistent with model EOS). - Fix ‘Euler‘ test cases missing EOS key in ‘dataFluid‘ tag of boundary conditions. - Fix memory errors during termination of the execution when EOS is missing. - Fix unstructured-mesh file parsing using msh v4 format (working now, although only for serial simulations). - Fix wrong elapsed time from simulations (was using CPU time, now uses wall time).

News of the Year: A new release, ECOGEN_v5.0, is available on GitHub. It includes new features and fixes bugs.

Points majeurs - Possibilité d’utiliser un schéma numérique d’ordre deux sur des maillages non structurés. Toujours en version bêta. - Les ondes acoustiques peuvent désormais être modélisées via des termes sources (impulsions sinusoïdales ou gaussiennes). Cela s’inspire des travaux de [_Maeda & Colonius_](https://doi.org/10.1016/j.wavemoti.2017.08.004) pour les écoulements monophasiques et a été étendu ici aux systèmes d’équations multiphasiques. - Pour les développeurs : amélioration de l’intégration continue (CI) incluant désormais la couverture de code, le formatage avec clang-format, une meilleure gestion des artefacts, l’utilisation de l’AddressSanitizer et des pré-commits. - Application de clang-format à l’ensemble du code.

Points mineurs - Ajout de scripts Gnuplot pour la génération de résultats en format PDF avec le format de sortie GNU. - Ajout de la vitesse selon x dans la sortie 1D avec Gnuplot. - Correction de traduction : *restart* → *resume*. - Amélioration de l’estimation de la longueur CFL pour les éléments quadrilatères. - Possibilité d’ajouter une fréquence d’enregistrement pour les coupes. - Correction des erreurs de mémoire (désallocations). - Possibilité de compiler le code avec ‘cmake‘ au lieu de ‘makefile‘ (pour les utilisateurs avancés). - Mise à jour de la documentation et des cas tests, comme d’habitude. - Utilisation de la clé ‘VTK‘ au lieu de ‘XML‘ pour la balise ‘outputMode‘ du fichier ‘main.xml‘. - Modification de la profondeur de tous les cas tests 1D et 2D à l’unité. Cela a été fait pour éviter d’éventuels problèmes lors de la visualisation des résultats avec des outils comme Paraview.

Corrections - Mise à jour des noms des manuels pour qu’ils soient cohérents avec les fichiers d’entrée des cas tests. - Mise à jour du Cv pour l’équation d’état de l’air. - Correction d’un bug et mise à jour des noms de variables dans ‘ModUEq::solveRIemannInletInjTemp‘. - Correction de l’assignation de l’EOS dans les conditions aux limites et ajout de l’obligation de présence de la clé EOS dans la balise ‘dataFluid‘ des conditions aux limites (doit être cohérent avec l’EOS du modèle). - Correction des cas tests ‘Euler‘ manquants de la clé EOS dans la balise ‘dataFluid‘ des conditions aux limites. - Correction des erreurs de mémoire lors de la terminaison de l’exécution en cas d’absence d’EOS. - Correction de l’analyse des fichiers de maillages non structurés au format msh v4 (fonctionne désormais, bien que

uniquement pour les simulations série). - Correction du temps écoulé incorrect dans les simulations (utilisait le temps CPU, utilise désormais le temps réel).

URL: https://code-mphi.github.io/ECOGEN/overview/

Publications: hal-01781830, hal-05124535, hal-03649359

Contact: Kevin Schmidmayer

Participant: 12 anonymous participants

Partners: Aix-Marseille Université, IUSTI, UMR CNRS 7343, CNES, California Institute of Technology, ETHZ, Université de Pau et des Pays de l'Adour

8 New results

8.1 Turbulence modelling

8.1.1 Improvement of the EB-RSM RANS model

Participants: Rémi Manceau, Corina Sanz Souhait, Jules Mazaleyrat.

External collaborators: E. Laroche (ONERA), G. Bonneau (Safran HE), Th. Grosnickel (Safran HE), Y. Smith (Safran HE), S. Benhamadouche (EDF), J.-F. Wald (EDF).

In order to accurately represent the complexity of the phenomena that govern the evolution of turbulent flows, an important part of our research focuses on the development of Reynolds-stress RANS models that take into account the wall/turbulence interaction by an original approach, elliptic blending [106, 109]. Although this approach, has been successfully applied to various configurations (for instance [66]), in order to take into account more subtle effects, during the theses of A. Colombié and G. Sporschill, in collaboration with ONERA and Dassault Aviation, respectively, we identified the importance of introducing a specific pressure diffusion model to correctly reproduce the dynamics of turbulence in impingement regions and in boundary layers subject to adverse pressure gradients, paving the way towards a wider application of the EB-RSM in aeronautics [130, 74, 131, 132, 133]. This activity is continued via the PhD thesis of J. Mazaleyrat in collaboration with SAFRAN HE and ONERA in the framework of turbine blade cooling by jet impingement.

Another aspect, relevant to the nuclear industry, is an accurate modelling of the temperature variance close to solid boundaries, since the penetration into the solid material of the temperature fluctuations generated by the turbulent flow is important for thermal fatigue. Since very different type of fluids can be involved (eg, air, water, pressurized water, liquid metals), the models for the turbulent heat flux and the temperature variance must be valid for a wide range of Prandtl numbers, from 0.001 to 10. Such a physical complexity requires solving the transport equation for the dissipation of the temperature variance. The PhD thesis of Corina Sanz Souhait, in collaboration with EDF, is dedicated to this type of models [21].

8.1.2 Extension of RANS turbulence models to mixed and natural convection

Participants: Rémi Manceau, Puneeth Bikkanahally.

External collaborators: S.M. Saad Jameel (formerly Cagire, now Plastic Omnium), V. Herbert (PSA-Stellantis), F. Dehoux (formerly Cagire, now Framatome), S. Benhamadouche (EDF), S.K. Jena (formely Cagire, now Bosch), M. Raba (CEA), D. Duri (CEA), A. Girard (CEA).

In the mixed and natural convection regimes, as presented in three invited lectures [107, 108, 53], the interaction mechanisms between dynamic and thermal fluctuations are complex and very anisotropic due to buoyancy effects [46], so that the natural turbulence modelling level to take them into account is second-moment closure, i.e., Reynolds-stress models. When associating the EB-RSM and the EB-DFM, several modifications had to be introduced in natural convection for the scrambling term, the length scale of the elliptic blending, and especially by substituting a mixed time scale for the dynamic time scale in the buoyancy production term of the dissipation equation, which has a drastic positive impact on the predictions in the natural convection regime. This work, carried out in collaboration with EDF, leads to the first linear Reynolds-stress model able to accurately represent the wall/turbulence interaction in forced, mixed and natural convection regimes [79]. However, some industrial partners, in particular PSA Group (now Stellantis), who encounter natural convection flows in the underhood compartment of vehicles, do not wish to use such sophisticated models, so we have developed an algebraic version of the Reynolds stress equation which thus constitutes an extension of the eddy-viscosity models (buoyancy-extended Boussinesq relation), within the framework of S. Jameel's thesis [96, 98, 97], which can be implemented into any industrial and/or commercial CFD code easily. The application of such models to various configurations of differentially-heated cavities showed that, depending on the situation, such buoyancy extensions can have an influence ranging from very significant to negligible [99]. A part of the PhD thesis of Corina Sanz Souhait, in collaboration with EDF, is also concerned with the extension of models for the turbulent heat flux and the temperature variance to these regimes [21].

A collaboration was launched in 2024 with CEA (Department for low-temperature systems, DSBT). Joint experimental and numerical studies are dedicated to understanding and modelling the flow in a vertical boundary layer in a cryogenic facility (liquiq helium, 4K, Ra=10^{14}) in order to investigate the behaviour of turbulence in the natural convection regime at very high Rayleigh number. An intern, Fatima Bouhenni, was hired at UPPA, financially supported by CEA, to investigate these type of flows with turbulence models.

8.1.3 HTLES: an original hybrid RANS/LES model

Participants: Rémi Manceau, Puneeth Bikkanahally.

External collaborators: Martin David (formerly Cagire, now University of Perpignan), Mahitosh Mehta (formerly Cagire, now EDF), Vladimir Duffal (formerly Cagire, now EDF), B. de Laage de Meux (EDF).

Regarding hybrid RANS/LES, we have developed the HTLES (hybrid temporal LES) approach. The wall/turbulence interaction being fundamental for the applications of interest to EDF, V. Duffal's thesis [83] focused on the precise control of the transition from RANS to LES when moving away from the wall, through the improvement of the theoretical link between the turbulent scales and the form of the model equations, as well as the introduction of two different shielding functions to avoid the classical grid-induced separation and log-layer mismatch [85, 84, 52, 83]. In the framework of the ANR project Monaco_2025, HTLES was extended to natural convection: in differentially heated cavities, due to the coexistence of turbulent boundary layers and a laminar region in the centre, the shielding function introduced by V. Duffal causes a deterioration of the results. Good results are obtained by using instead a new shielding function based on the resolution of an elliptic relaxation equation [68, 51].

8.1.4 Towards embedded LES

Participants: Rémi Manceau, Pascal Bruel, Puneeth Bikkanahally.

External collaborators: Martin David (formerly Cagire, now University of Perpignan), Mahitosh Mehta (formerly CAGIRE, now EDF), Fabien Dupuy (GD-Tech), Olivier Jegouzo (GD-Tech).

In the framework of hybrid RANS/LES, a particularly attractive approach is Embedded LES, which consists in reserving the LES to a small area included in a global RANS domain, which is a particular strategy for using the zonal hybrid RANS/LES. However, the zonal approach is characterized by a pre-division between RANS and LES zones and a discontinuous interface, which prohibits any evolution of the scale of description of turbulence during the calculation, which would allow an adaptability of the model according to physical criteria determined during the calculation. Our objective is therefore to develop embedded LES in the context of continuous approaches (CELES, Continuous Embedded LES), in which the interface between RANS and LES is now a diffuse interface. In these approaches, the domain is not split into sub-domains, but the model evolves in a continuous manner so that it tends towards a RANS model or towards a LES model. The diffuse interface (grey area) is the transition area in which the model transitions from a RANS model to a LES model. It is then necessary, as in the zonal approach, to enrich the RANS solution by adding synthetic turbulence to avoid the drastic decrease of the total turbulent stress at the beginning of the LES zone which would strongly degrade the results. In the framework of the hybrid RANS/LES approach developed by Cagire, HTLES, this aspect consists in developing a volume enrichment approach based on a fluctuating force [113, 49, 33]. The development of such a CELES approach is the main purpose of the CELTIC project (post-doc of P. Bikkanahally), in collaboration with the SME GD-Tech [19, 20, 22]. An adaptive determination of the RANS and LES regions based on physical criteria was the subject of the post-doc of M. David, in the framework of the Asturies project [43]. All the recent progress made on this topic was presented at the invited opening conference of the Lille turbulence program [15].

8.1.5 Turbulent premixed combustion in the flamelet regime: developing a closure model for the time filtered reaction rate

Participants: Pascal Bruel.

External collaborator: S. Elaskar (Universidad Nacional de Córdoba, Argentina).

With the objective of extending the temporal large-eddy simulation to turbulent reactive flows, the flamelet regime of isenthalpic turbulent premixed combustion was first considered. In such a regime, the combustion process can be represented through the evolution of a single progress variable whose time evolution resembles a bi-valued telegraph signal. We first concentrate on the reaction rate, leaving aside for the moment the question of the closure of the filtered scalar flux. In a RANS approach, corresponding to an infinite time filter width, many models are available to close the mean reaction rate as a function of the mean progress variable. So the question raised now is: what happens to the relation between the filtered reaction rate and the filtered variable when the time filter width remains finite ? To guide our thinking, the development of the capacity of generating and filtering synthetic telegraph signals was deemed necessary [44]. After considering in [56] the possibility of using the so-called "poor man Navier-Stokes" approach, we develop a fortran code aimed at directly generating synthetic telegraph signals satisfying some a priori constraints so as to mimick real signals measured in such a combustion regime. With such a tool, we were able to recover the behavior of the filtered reaction rate for quasi-infinite time filter widths e.g. the RANS behavior. Our future activity will now concentrate on the numerical study of filtering the synthetic progress variable signal with finite time filter widths.

8.2 High-order numerical methods and efficient algorithms

8.2.1 Efficient implementation of flux-reconstruction methods for combustion

Participants: Vincent Perrier, Romaric Simo-Tamou.

External collaborators: Julien Bohbot (IFPEN), Julien Coatléven (IFPEN), Quang Huy Tran (IFPEN).

In the framework of the PhD thesis of Romaric Simo-Tamou, flux-reconstruction methods were implemented, first in AeroSol for the Navier-Stokes system, and then in the Converge CFD code for high order computation of combustion and for benefiting of AMR in this code. For these schemes, new analyses of their dissipation and dispersion properties were performed. The Phd thesis of Romaric Simo-Tamou was defended in Macrh 2025 [30]

8.3 Compressible and multiphase flows

8.3.1 Low-Mach-number schemes

Participants: Jonathan Jung, Vincent Perrier, Esteban Coiffier.

External collaborators: Ibtissem Lannabi (formerly Cagire, now Serena Inria Paris), Michael Ndjinga (CEA).

The use of pressure centered type fixes can provide oscillating numerical solutions. These oscillations on the velocity field appear on triangular and quadrangular meshes and have a catastrophic impact on the solution because they may jeopardize the mesh convergence [58]. A detailed study of these oscillating modes has been carried out in the triangular casea and it was proved that the dimension of the space generated by these oscillating modes is greater than the number of internal nodes in the mesh, which is very large. Then, filtering these oscillating modes does not appear to be a reasonable solution. Moreover, a basis of these oscillating modes has been numerically generated. This work was published in [12]

The previous study on the oscillating modes revealed that the fix developed in [45], in addition to not being Galilean invariant, could also be subject to these oscillating modes. This led us to tackle the problem in a different way from the numerical flux modification. Based on the link between the low Mach number solution of the Euler system and the long time limit of the wave system done in [100], a numerical scheme is low Mach number accurate if its low Mach number acoustic development has a consistent long time behaviour. Then, the general problem of conservation of vorticity for the wave system is addressed. In [48], we propose to enrich the approximation space for the velocity, then the Hodge-Helmholtz context developed for triangular meshes in [81] can be recovered in the quadrangular mesh case. This leads to a numerical scheme for the wave system that naturally preserves the vorticity and is long time limit consistent if the numerical flux conserves exactly contacts. Using this new approximation space for Euler system, the resulting numerical scheme is accurate for both steady and acoustic problems at low Mach number. This work was disseminated in [32].

We also looked at numerical schemes based on staggered discretizations (velocity at faces and pressure at cells), initially developed for incompressible fluid mechanics [94], with the aim of adapting the method to compressible fluid mechanics. We began by studying the energy stability and vorticity conservation of the wave system discretizations. The stability study revealed that in order to obtain an explicit dissipative energy scheme, it is necessary to have numerical diffusion on both variables, which would lead us to change the schemes usually used, which are either partially centred or completely centred. By reinterpreting the staggered numerical schemes in terms of finite element schemes, we were able to determine the discrete diffusion operators that naturally preserve the discrete vorticity. More precisely, the velocities at the faces are interpreted as a finite element approximation in Raviart-Thomas space, while the pressure is interpreted as a discontinuous approximation of degree 0. From the discrete de-Rham structure that links these approximation spaces, the proposed explicit scheme guarantees the consistency of the long-time limit. The thesis of Esteban Coiffier on staggered schemes was defended in December 2025 [29]. This work was also presented in the conferences [23, 24]

8.3.2 Preservation of differential constraints in hyperbolic systems

Participants: Jonathan Jung, Vincent Perrier, Esteban Coiffier.

External collaborators: Michael Ndjinga (CEA).

Our work on accurate low Mach number schemes led us to be interested in the wider problem of the preservation of implicit involutions. Implicit involutions are additional differential equations that are ensured for continuous system, this is for example the vorticity for the wave system, or the divergence of the magnetic field for the Maxwell equations or the magneto-hydro-dynamic system. As these involutions are implicit, they represent additional contraints with respect to the original system, and their preservation at the discrete level is known to be a complicated problem.

Starting from the triangular case, we were able to derive a distributional de-Rham complex for the classical approximation spaces for vectors, on which we could prove that the discrete and continuous cohomology spaces are matching on periodic domains. These properties give another point of view on the previously proven results for low Mach number flows on triangular meshes published in [47]. We were also able to prove that the low order approximation space proposed on quadrangular meshes in [48] can be easily extended to higher order. The proof of these properties and the derivation of the quadrangular approximation space were published in [14].

Based on these approximation spaces, we were able to derive discontinuous Galerkin schemes on which an implicit preservation of the curl or the divergence can be easily proven. The article was accepted this year [13].

This work was disseminated in the following conferences [25, 26, 27], and the following invited talks [17, 16]

8.3.3 ImEx methods with multigrid methods for low Froude shallow-water equations

Participants: Vincent Perrier, Daniel Inzunza.

Within the ANR Lagoon project, we aim at developing high order numerical methods for which the CFL number is based on the convective CFL, and independent of the velocity of the acoustic waves. For this, we rely on a plitting of the shallow-water system which makes appear a nonlinear wave system that should be integrated implicitly. As the stiffness of this system is similar, with large time steps, to the stiffness of the Laplace equation, we plan, we plan to work on efficient multigrid methods and full approximation schemes.

A preliminary result was presented in [31].

8.3.4 Multi-scale multiphase flows

Participants: Vincent Perrier, Kevin Schmidmayer, Qa'im Bekkali.

As far as multiphase models are concerned, based on the ideas of [82], we have revisited the derivation of Baer-and-Nunziato models [63]. Usually, models are derived by averaging the Euler system; then the system of PDE on the mean values contains fluctuations which are modeled, often leading to relaxation terms and interfacial velocity and pressure which should also be modeled. This can be achieved by using physical arguments [125] or by ensuring mathematical properties [75]. In [118], we have followed a slightly different path: we have supposed that the topology of the different phases follows an explicit model: the sign of a Gaussian process. Some parameters of the Gaussian process (mean, gradient of the mean) are linked with the averaged values of the flow (volume fraction), whereas others (auto-correlation function) are linked with the subscale structure of the flow. The obtained system is closed provided the parameters of the Gaussian process are known. Also, the system dissipates the phase entropies. Under some hypotheses that can be interpreted physically, asymptotic models can be derived in the interface flow limit or in the limit where the two fluids are strongly mixed. In these limits, different previously proposed models are recovered [125, 89], which does not necessarily ensure the same phase entropy dissipation properties. This work was disseminated this year in the conference [54]. A project was granted this year for working on multiscale multiphase flows. The project is funded by the Région Nouvelle Aquitaine, and provides half a Phd grant, the other half being provided by Pau University. Qa'im Bekkali has started his PhD on this project since November 2025.

8.3.5 Shock-induced cavitation within a droplet

Participants: Kevin Schmidmayer, Mandeep Saini.

External collaborators: L. Biasiori-Poulanges (ETH Zürich, Switzerland), F. Denner (Polytechnique Montréal, Canada).

In [42] and [50], we investigated the shock-induced cavitation within a droplet which is highly challenged by the multiphase nature of the mechanisms involved. Within the context of heterogeneous nucleation, we introduced a thermodynamically well-posed multiphase numerical model accounting for phase compression and expansion, which relies on a finite pressure-relaxation rate formulation. We simulated (i) the spherical collapse of a bubble in a free field, (ii) the interaction of a cylindrical water droplet with a planar shock wave, and (iii) the high-speed impact of a gelatin droplet onto a solid surface. The determination of the finite pressure-relaxation rate was done by comparing the numerical results with the Keller–Miksis model, and the corresponding experiments of Sembian et al. and Field et al., respectively. For the latter two, the pressure-relaxation rate was found to be $\mu = 3.5$ and $\mu = 0.5$, respectively. Upon the validation of the determined pressure-relaxation rate, we ran parametric simulations to elucidate the critical Mach number from which cavitation is likely to occur. Complementing simulations with a geometrical acoustic model, we provided a phenomenological description of the shock-induced cavitation within a droplet, as well as a discussion on the bubble-cloud growth effect on the droplet flow field. The usual prediction of the bubble cloud center, given in the literature, was eventually modified to account for the expansion wave magnitude.

More recently, we are working on improving the sub-grid bubble modelling. Indeed, resolving every single bubble in direct numerical simulations is extremely expensive and often impractical, making subgrid modelling essential. We therefore model a bubble cluster as a subgrid mixture of the gas inside the bubbles and the surrounding liquid. The numerical framework is based on the model explained above. We build on this approach and extend the model to bubble cluster dynamics by proposing closures based on three different ideas: dimensional analysis, solving a spherical Riemann problem at the subgrid bubbles, and modelling bubbles with the Rayleigh–Plesset equation. We show that all three approaches lead to the same pressure-relaxation rate between the bubble cluster and the surrounding fluid. We validate the model using two test cases: the collapse of a single bubble and the evolution of a bubble cluster inside a cylindrical droplet impacted by a Mach 2.4 shock wave. The proposed model provides a practical tool for design and control of systems in which bubble clusters play a key role, such as lithotripsy (kidney-stone treatment), noise reduction using bubble curtains in offshore drilling, or shock-induced droplet atomisation for aeronautics applications.

8.3.6 Modelling of visco-elastic solids in multiphase flows

Participants: Kevin Schmidmayer, Adedotun Ade, Algiane Froehly.

External collaborators: N. Favrie (Aix-Marseille Université).

As a work in progress, an extension of the model of diffuse solid–fluid interfaces [117, 86] is proposed to deal with arbitrary complex materials such as porous materials in presence of plasticity and damage [34]. These are taken into account through Maxwell-type models and are cast in the standard generalized materials. The specific energy of each solid is given in separable form: it is the sum of a hydrodynamic part of the energy depending only on the density and the entropy, an elastic part of the energy which is unaffected by the volume change, and a compaction part taking into account the compaction effects. It allows us to naturally pass to the fluid description in the limit of vanishing shear modulus. In spite of a large number of governing equations, the model has a simple mathematical structure. The model is well posed both mathematically and thermodynamically, *i.e.* it is hyperbolic and compatible with both laws of thermodynamics. The resulting model can be applied in situations involving an arbitrary number of fluids and solids. In particular, we are showing the ability of the model to describe complex plasticity (Gurson [92]) and damage (Mazars [112]) models. The first and fundational paper of this project has been submitted at the end of 2025.

8.3.7 Impulse-driven release of gas-encapsulated drops

Participants: Kevin Schmidmayer.

External collaborators: G. T. Bokman (ETH Zürich, Switzerland), L. Biasiori-Poulanges (ETH Zürich, Switzerland), B. Lukić (ESRF, France), C. Bourquard (Dynamics and Control, Netherlands), E. Baumann (ETH Zürich, Switzerland), A. Rack (ESRF, France), B. J. Olson (Lawrence Livermore National Laboratory, USA), O. Supponen (ETH Zürich, Switzerland).

Gas-encapsulated drops, much like antibubbles, are drops enclosed in a bubble within a liquid. They show potential as payload carriers in fluid transport and mixing techniques where sound waves can be leveraged to induce the collapse of the gas core and the subsequent release of the drop. In [69], the interaction of millimeteric gas-encapsulated drops with impulsive laser-induced shock waves is investigated to gain fundamental insights into the release process. Experimental synchrotron X-ray phase contrast imaging, which allows the drop dynamics to be visualised inside the encapsulating bubble, is complemented by numerical simulations to study the intricate physics at play. Three drop dynamical release regimes are discovered, namely the *drop impact*, *partial deposition* and *jet impact* regimes. The regime type is mainly dependent on the shape of the bubble interface impacting the drop and the associated Weber and Reynolds numbers. The drop dynamics of the *drop impact* and *partial deposition* regimes show similarities with the canonical configuration of drops impacting flat liquid surfaces, while the *jet impact* regime resembles binary drop collisions, which allows existing scaling laws to be applied to describe the underlying processes. The release of the drop is investigated numerically. The time evolution of the drop dissemination within the surrounding liquid discloses enhanced mixing for dynamics involving high Weber and Reynolds numbers such as the *drop impact* and *jet impact* regimes.

8.3.8 Experiment, modelling and simulation of shock-wave lithotripsy

Participants: Kevin Schmidmayer, Adedotun Ade.

External collaborators: A. Sieber (ETH Zürich, Switzerland), C. Brewer (ETH Zürich, Switzerland), G. T. Bokman (ETH Zürich, Switzerland), B. Lukić (ESRF, France), G. Shakya (ETH Zürich, Switzerland), O. Supponen (ETH Zürich, Switzerland), M. Belau (Storz Medical, Switzerland), A. Kühl (Storz Medical, Switzerland), M. Schlötter (Storz Medical, Switzerland).

Extracorporeal shock wave lithotripsy (ESWL) is a non-invasive medical procedure where kidney stones are fragmented in smaller pieces allowing them to naturally evacuate the kidney. After the shock wave is delivered, cavitation bubbles usually form around the stone. The collapse of these bubbles is considered one of the important mechanisms that contribute to the fragmentation of the stone, in addition to the mechanical effects resulting from the shock wave-stone interaction. However, due to the demanding spatiotemporal scales of the phenomenon and optical challenges due to dense bubble clouds and opaqueness of the stone, direct observations disclosing the exact mechanisms in play remain scarce. To address these experimental challenges, we conducted in-situ ultrafast X-ray imaging of shock wave-stone interactions at the ID19 beamline of the European Synchrotron Radiation Facility, using a medical lithotripter to generate the shock waves. These visualizations were complemented by high-resolution post-impact microtomography to assess the location and progression of damage within the stone. The work in progress reports the patterns of damage development observed in both phantoms and real kidney stones, with the aim to relate these patterns to the different mechanisms of stone fragmentation.

To complement the experiments done, modelling and simulation efforts are employed to analyse cavitation-induced erosion, playing a significant role alongside direct mechanical loading in the fragmentation process [124]. However, the mechanisms governing multi-bubble interactions remain poorly understood. This study investigates dual-bubble dynamics through combined ultra-high-speed X-ray phase-contrast imaging (100 kfps), conducted at ESRF-ID19, and diffuse-interface numerical simulations using ECOGEN [128, 41]. We examine interactions between stone-attached and distant cavitation bubbles with BEGO stone (kidney-stone phantom) which is subjected to medical-grade shock waves (peak pressure 92 MPa, incident compression phase of the shock is approximately 1 μs) generated by the MODULITH® SLK ≫intellect≪ lithotripter (Storz Medical AG). The numerical simulations accurately reproduce experimentally observed bubble dynamics, demonstrating good qualitative and temporal agreement in jet formation, penetration, and collapse sequences. Our simulations reveal that dual-bubble configurations generate peak pressures of 87.1 Mpa, with sustained and localised high pressures for approximately 25 μs, at the jet-impact location, compared to 10.4 MPa for single stone-attached bubble cases, representing an order-of-magnitude amplification that substantially exceeds typical kidney stone compressive strengths (3.2–6.2 Mpa). This amplification results from constructive interference when the distant bubble jet penetrates the stone-attached bubble and impacts the stone surface coincident with the larger bubble's collapse. Systematic parametric studies (distant bubble standoff distance, size and aspect ratio) establish that the standoff distance is the dominant parameter governing pressure amplification. Simulations correctly predict erosive loading locations consistent with observed surface erosion patterns, providing a foundation for future investigations incorporating fluid–structure interaction and damage mechanics, with implications not only for optimising ESWL treatment protocols through controlled bubble dynamics but also for advancing the broader understanding of cavitation erosion in biomedical and engineering systems such as burst wave lithotripsy (BWL) and hydraulic machinery.

8.3.9 Numerical simulation of the mitigation of explosion effects using aqueous foam

Participants: Kevin Schmidmayer, Lucas Martin de Fourchambault.

External collaborators: F. Petitpas (Aix-Marseille Université), M. Reynaud (CEA Gramat), P. Graumer (CEA Gramat).

Studies have shown [80, 64] that confining an explosive with dry aqueous foams can limit the destructive effects of detonation in terms of shock waves and blast waves. More recently, the use of foam has also highlighted the potential capture of micro- and millimeter-sized particles [114]. More precise analyses of the impact of confinement, including the slowing down or capture of particles, require numerical simulation of the phenomenon.

The developed numerical tools must account for various physical aspects to accurately represent both the detonation phenomenon and particle transport (up to potential capture) by aqueous foam. The challenges are diverse:

- Compressibility of phases.
- Computating the propagation of shock waves and detonation waves within heterogeneous materials (mixtures of solids, gases, etc.) [119].
- Flows being multi-velocity during the interaction phase between particles and aqueous foam, a velocity non-equilibrium model is necessary.
- Fragmentation of solids and foam.

The ECOGEN computational tool [128, 41], co-developed mainly with Aix-Marseille Université (IUSTI), is sufficiently advanced to eventually perform complex numerical simulations on this issue. One of the critical aspects of future developments involves accounting for velocity non-equilibrium.

The goal of the ongoing project is to introduce new, precise, and robust numerical models and methods into ECOGEN to enable the simulation of multiphase flows (liquid, gas, solid) with velocity non-equilibrium between phases. These methods should be extended to higher orders to improve result accuracy. The detonation phenomenon must be considered through the development of stiff source term integration methods to enable the simulation of an explosion confined by aqueous foam. This study is able to rely on existing experimental data coming from, in particular, the CEA Gramat.

8.3.10 Towards Large Eddy Simulation (LES) of multiphase compressible flows

Participants: Felice Edoardo Taglialatela, Kevin Schmidmayer.

External collaborators: G. De Stefano (University of Campania “L. Vanvitelli").

Turbulent two-phase flows are present in many industrial applications: liquid atomisation in fuel injection, metal casting and many others. All of these processes share the following: the two immiscible fluids separated by a distinct interface undergo several phenomena, including deformation, breakup, and coalescence. More interestingly, all of the mentioned interface transformations have their characteristic time and length scales. For this reason, performing a DNS is still impractical. For this reason, the strategy to keep the computational cost reasonable is to solve the large scales of the flow (responsible of transporting the majority of the conserved properties) and to model the smallest one that the computational grid is not able to capture. The first set of the two is what is commonly referred to as *large eddies*, while the second one is the set of the *sub-grid scales*. The whole procedure is called Large Eddy Simulation (LES).

LES of two-phase compressible flows is an even more challenging task due to the interface-turbulence interaction. Obtaining the LES governing equations involves performing the filtering operation (*i.e.*, averaging over space) on the Navier–Stokes and scalar equation. This procedure closely resembles the approach used in the single-phase context; however, it gives rise to an expanded set of sub-grid terms, among which only a subset coincides with those appearing in the single-phase formulation. Using Favre filtering and *a priori* results can reduce the number of sub-grid terms to account for.

The current state of this work is the following: a coarse DNS of twin droplet aerobreakup is presented as a case study [134]. Then, the state-of-the-art closure for the sub-filter scales (SFS) in the LES formulation for single-phase flows is employed. The model is implemented in the open-source finite-volume code ECOGEN [128, 41], where an *a posteriori* analysis is conducted to evaluate the correctness and performance of the implementation.

8.3.11 Machine learning for relaxation coefficients in multiphase compressible flows

Participants: Alexis Altolaguirre, Kevin Schmidmayer, Rémi Manceau.

In the realm of multiphase compressible flows, flows such as the atomisation of a droplet by an high-speed flow, cavitation activities around propellers, or simply waves interacting with interfaces, involve multi-scale features for which both experiments and simulations can't comprehend and evaluate correctly. However, while the larger features may be observable and well captured by computational tools, the smallest features are hidden, although they have a significant importance on the behaviour of the whole flow.

Among the existing numerical methods available in the literature, the diffuse-interface method is a particularly attractive one since it allows to simulate both interface and mixture flows, and new interfaces (phase appearance) happen naturally. In this method, the interface is diffused and can therefore be seen as a thin mixture region. While this feature is an advantage regarding robustness and the fact it involves a unique solver for all the flow regions, it can be problematic when solving interfaces that behave like mixtures where the speed of sound differs significantly. A phenomenon of wave trapping can happen and alter the solution resulting from the interaction of waves with interfaces [127]. Another scenario is when you have both interface scale and real mixtures in a unique simulation, such as what happens during droplet shock-induced cavitation [42]. In such scenarios, the relaxation terms, dealing with the natural behaviour of mixtures, need to be adapted to properly model the various flow features. However, at the state of the art, this adaptation is done by hand and is test case dependent, making it unsuitable for studying unknown flows where experimental observations are lacking.

In this context, the aim of this exploratory project is to leverage machine learning algorithms to evaluate the value of the relaxation coefficients to apply for given conditions. This evaluation will be performed in each cell of the mesh and at each time step of the simulation. Only this evaluation would be done by machine learning, meaning the whole solver stays intact, which allows to guarantee its mathematical and mechanical properties directly linked to robustness, efficiency and accuracy. The training process will be integrated into CFD simulations, leveraging known input parameter values at all times for the machine learning algorithm. However, unlike conventional approaches where expected values of the relaxation coefficients would be known, we only have expectations for the thermo-mechanical outcomes. Consequently, the objective is to determine the relaxation coefficients that minimize the discrepancy between the relaxation solver's output, given these coefficients, and the expected thermo-mechanical results. This approach ensures that the machine learning model adapts to produce the most accurate relaxation coefficients for achieving the desired thermo-mechanical behaviour. Several machine-learning algorithms would be tested in order to assess their performance. A particularly interesting option would be the use of gene-expression programming [88, 135] which would allow for retro-engineering and obtaining a physically comprehensive formulation. The work therefore consists of developing, training and testing these algorithms. The first attempt will rely on solving the interaction of waves with interfaces, similarly to what was done in [127], where exact solutions are known at all times and spatial locations. A second test will consist of solving the droplet shock-induced cavitation without choosing *a priori* a value for the pressure relaxation coefficient.

If positive results emerge, new and mainly unknown multi-scale flows could be studied with precision for the first time. Another aspect of this study would be to gain valuable, yet unknown, information about multiphase mixtures and possibly better assess the behaviour of the relaxation coefficients in various configurations.

8.4 Analysis and simulation of turbulent flows and heat transfer

8.4.1 Effusion cooling

Participants: Rémi Manceau, Pascal Bruel.

External collaborators: Martin David (formerly Cagire, now university of Perpignan), Ph. Reulet (ONERA), E. Laroche (ONERA), D. Donjat (ONERA), F. Mastrippolito (formerly CAGIRE, now SAFRAN HE).

As regards wall cooling by effusion (jets in crossflow), our MAVERIC experimental facility does not allow us to carry out thermal measurements, so we approached ONERA Toulouse to collaborate on the effects of gyration (angle of the jets with respect to the incident flow) on the heat transfer between the fluid and the wall, within the framework of the European project SOPRANO. We then took up the challenge of carrying out RANS simulations with the EB-RSM model on a configuration of unprecedented complexity for us, consisting of 10 rows of 9 holes, in 90-degree gyration, representative of effusion cooling problems in aeronautical combustion chambers. Comparisons between calculations and experiments have shown the relevance of using the EB-RSM model and the importance of taking into account conjugate heat transfer [121, 111]. In the framework of the Asturies project, the case, the database of a jet in crossflow measured in the MAVERIC facility has been investigated with the active hybrid RANS/LES we have developed, in order to serve as a demonstrator of this agile simulation method.

8.4.2 Thermocline energy storage

Participants: Rémi Manceau.

External collaborators: Alexis Ferré (formerly Cagire, now NALDEO), S. Serra (LaTEP, UPPA), J. Pouvreau (CEA), A. Bruch (CEA), M. Rudkiewicz (CEA).

In the framework of a collaboration with the CEA LITEN in Grenoble and the LaTEP of UPPA on thermocline energy storage, an experimental facility has been developed at the CEA and URANS simulations were carried out to understand the dynamics of this type of flows, to determine the influence of the turbulence generated by the filling of the tank on the quality of the thermocline, in order to optimize the system and provide data to support the development of 1D models used in the optimization of heat networks. A particular attention has been paid to the approximation used for variations of density with temperature. Due to the wide range of temperatures, it was shown that the standard Boussinesq approximation is not valid but a quadratic Boussinesq approximation was proposed, which gives results very close to the more complex low-Mach number approximation, with a computational cost reduced by a factor of two and an improved numerical stability [87, 57]. As concerns turbulence modelling, the coupled mechanisms between buoyancy, turbulence and the thermocline was identified for the first time. Buoyancy completely suppresses turbulence right at the top of the thermocline, which explains its asymmetry.

8.4.3 Thermal fatigue and stress corrosion cracking

Participants: Rémi Manceau, Joséphine Gauthier.

External collaborators: S. Benhamadouche (EDF), J.-F. Wald (EDF), A. Morente (EDF).

Thermal fatigue and stress corrosion cracking are two phenomena that can cause cracks of varying severity to appear in the welds of pipes in the primary circuit of a pressurized water nuclear reactor. This problem caused the shutdown of 12 EDF nuclear reactors in the winter of 2022-2023.

The problem is caused by the branch pipe (dead arm), which is not supposed to have any flow, but in which a secondary flow can be observed that can penetrate the dead arm by several diameters. Quantitative predictions of the vortex penetration length, possible thermal stress frequencies, and the consequences of these phenomena on thermo-hydraulic behavior can be made using the tools and models available today, but with prohibitive computation times, making in-depth R&D analyses too time-consuming and rendering industrial exploitation impossible from an engineering standpoint.

Joséphine Gauthier's CIFRE thesis, funded by EDF, consists of developing a new computational approach that allows for reasonable return times for calculations. The main idea is to divide the computational domain into three coupled subdomains: turbulent zone, rapidly evolving chaotic zone, and very slowly evolving laminar zone. Each sub-domain will have its own specific model, with the main objective being to decouple the numerical time steps. The method targeted is therefore similar to a multiscale approach. To our knowledge, there are no published studies on similar approaches, and the main challenge will be the modelling of the interface conditions between the sub-domains, which will require, among other things, the adaptation of the ALF (Anisotropic Linear Forcing) method developed by the team [76] to this problem.

9 Bilateral contracts and grants with industry

Participants: Rémi Manceau, Vincent Perrier, Jonathan Jung, Pascal Bruel, Kevin Schmidmayer, Anthony Bosco, Romaric Simo Tamou, Alexis Ferré, Esteban Coiffier, Lucas Martin de Fourchambault, Corina Sanz Souhait, Jules Mazaleyrat.

9.1 Bilateral contracts with industry

- CEA: "Agile simulation of turbulent internal flows", contract in the framework of the Asturies project.
- CEA: "Collaboration contract for the PhD thesis of E. Coiffier".
- IFPEN: "Collaboration contract for the PhD thesis of Romaric Simo-Tamou".
- CEA DAM: "Collaboration contract for the Master 2 apprenticeship and engineer contract of L. Martin de Fourchambault, on the implementation of a multi-velocity model into ECOGEN".
- CEA DAM: "Collaboration contract for the PhD thesis of L. Martin de Fourchambault, on the numerical simulation of the mitigation of explosion effects using aqueous foam".
- EDF: "Collaboration contract for the PhD thesis of Corina Sanz Souhait"
- SAFRAN HE and ONERA: "Collaboration contract for the PhD thesis of Jules Mazaleyrat"

9.2 Bilateral grants with industry

- CEA: "Development of Fast, Robust and Accurate numerical methods for turbulence models on Complex Meshes" (1/2 Grant), PhD student Anthony Bosco.
- CEA: "Numerical analysis and simulation of staggered discretizations in two-phase thermohydraulics", PhD student Esteban Coiffier.

- IFPEN: "Development of high-order schemes for a Cartesian / AMR solver for LES combustion modeling", PhD student Romaric Simo-Tamou.
- EDF: "Industrialization of advanced RANS models with heat transfer for forced, mixed and natural convection", PhD student Corina Sanz Souhait.
- SAFRAN HE: "Improved numerical modeling of jet impingement cooling", PhD student Jules Mazaleyrat.

10 Partnerships and cooperations

10.1 International initiatives

10.1.1 Participation in other International Programs

Collaboration with Waterloo university within the Fonds Commun de Recherche France-Canada

Participants: Vincent Perrier.

Title: de-Rham complex extension of summation-by-parts methods with application to non-linear hyperbolic systems

Partner Institution(s): • University of Waterloo

Date/Duration: 2 years

The aim of this project is to adapt the stability analysis framework for hyperbolic systems developed at Waterloo University (Summation By Part) to the new space approximations developed within the team for preserving divergence or curl constraints in order to get numerical schemes that ensure both preservation of implicit involutions and provable entropy/energy stability.

10.2 International research visitors

10.2.1 Visits of international scientists

Other international visits to the team

Felice Edoardo Taglialatela, a PhD student in fluid dynamics, funded by University of Campania “L. Vanvitelli" through the Italian Ministry of University and Research, visited the team for a week in July, before officially joining the team to collaborate with us on numerical methods and turbulence simulation, in particular with Large Eddy Simulations (LES) models, for multiphase compressible flows. He is visiting the team for 6 months since October.

10.3 National initiatives

10.3.1 ANR LAGOON

Participants: Vincent Perrier, Abderrahman Benkhalifa, Daniel Inzunza.

The ANR project Lagoon was funded by ANR in 2021 within the section CE46 - Modèles numériques, simulation, applications.

Coastal areas host around 10% of the world’s population and a huge amount of economic activities. Climate change is expected to increase coastal flooding hazard in years to come. In this project, we propose to develop a numerical tool for the stormsurges predictions.

For four years, a joint effort between the partners of this project among others has been done for the development of a numerical tool able to tackle planetary computations with high resolution at the coast: the Uhaina code, based on top of the AeroSol library. The scope of this project is to increase the computational performance of our modelling platform, in order to upgrade it as an efficient and accurate tool for storm-surge predictions in different future climate scenarios. To achieve this goal and producing results which go beyond the state-of-the-art, our efforts will be focused on the following numerical and informatics developments, devoted to decrease the run time of the model in operational conditions:

- Development of low-Froude accurate Implicit-Explicit (ImEx) time integration strategy.
- Development of scalable aggregation-based multigrid methods for addressing the efficiency of the inversion of the (non)linear systems induced by implicit time stepping. For the data generation, two stages IO, in-situ and in-transit data post-processing are strategies that will be evaluated with existing technologies and will be implemented to improve the performance of the production code.
- The numerical tool will be validated on 1979-2014 sea level reanalysis, and be used for the generation of a database of sea level projections on future climate CMIP6 projections.

The code developed within this project will be freely distributed, with a strong effort put on reproducibility of results. Data generated for both the sea level reanalysis and the database of sea level projection for future climate projections will be distributed towards the community.

10.3.2 ANR MSBUB

Participants: Kevin Schmidmayer, Adedotun Ade, Mandeep Saini, Algiane Froehly, Alexis Altolaguirre, Rémi Manceau.

Call: ANR young researcher project within the section CE51 – Sciences de l'ingénierie et des procédés.
Dates: 2024-2029
Partners: Aix-Marseille Université (IUSTI) ; ETH Zürich ; European Synchrotron Radiation Facility ; Storz Medical
Project title: Modelling and simulation of bubble dynamics near a kidney stone
Cavitating flows appear in numerous fields, including in biomedical applications such as in lithotripsy (treatment for kidney stones) where cavitation bubbles, induced by shock waves, laser energy deposit or high-intensity focused ultrasound waves, violently collapse and interact with biomaterials. In this context, the young researcher and his team, experts on modelling and study of multiphase compressible flows, including solids, for industrial and biomedical applications, aims to tackle the particularly challenging and ambitious modelling of the dynamics of bubbles near a kidney stone where numerous scientific and technical obstacles remain to be overcome. Among them, we could cite major obstacles such as the modelling of biomaterials under a fluid-mechanics formulation including viscoelastic behaviour and realistic equations of state, and the modelling of cavitation (phase change). The simultaneous coupling of compressible, multi-component flow models with viscoelastic solids will enable us, through simulations, to understand the fundamental physics taking place and therefore fill the knowledge gap on the subject involving significant range of physical phenomena that are not well understood yet, and for which experiments often lack information, and spatial and temporal resolution. This will potentially lead to significant improvements of the current and future lithotripters regarding their success rate, cost and safety. In addition to its initial purpose, this project participates to the funding of exploratory research regarding the enrichment of two-phase flow models through machine learning, in particular through symbolic regression approaches.

10.3.3 National collaborative effort on detonation in heterogeneous materials

Participants: Kevin Schmidmayer, Lucas Martin de Fourchambault.

Master Apprenticeship and PhD thesis funded by CEA Gramat.
Dates: 2024-2029
Partners: Aix-Marseille Université (IUSTI) ; CEA Gramat ; ISAE-ENSMA (PPrime, Poitiers)
Project title: Numerical simulation of the mitigation of explosion effects using aqueous foam

Studies have shown [80, 64] that confining an explosive with dry aqueous foams can limit the destructive effects of detonation in terms of shock waves and blast waves. More recently, the use of foam has also highlighted the potential capture of micro- and millimeter-sized particles [114]. More precise analyses of the impact of confinement, including the slowing down or capture of particles, require numerical simulation of the phenomenon.

The developed numerical tools must account for various physical aspects to accurately represent both the detonation phenomenon and particle transport (up to potential capture) by aqueous foam. The challenges are diverse:

- Compressibility of phases.
- Computating the propagation of shock waves and detonation waves within heterogeneous materials (mixtures of solids, gases, etc.) [119].
- Flows being multi-velocity during the interaction phase between particles and aqueous foam, a velocity non-equilibrium model is necessary.
- Fragmentation of solids and foam.

The ECOGEN computational tool [128, 41], co-developed with Aix-Marseille Université (IUSTI), is sufficiently advanced to eventually perform complex numerical simulations on this issue. One of the critical aspects of future developments involves accounting for velocity non-equilibrium.

The goal of the ongoing project is to introduce new, precise, and robust numerical models and methods into ECOGEN to enable the simulation of multiphase flows (liquid, gas, solid) with velocity non-equilibrium between phases. These methods should be extended to higher orders to improve result accuracy. The detonation phenomenon must be considered through the development of stiff source term integration methods to enable the simulation of an explosion confined by aqueous foam. This study is able to rely on existing experimental data coming from, in particular, the CEA Gramat.

10.4 Regional initiatives

10.4.1 MODEM

Participants: Vincent Perrier, Kevin Schmidmayer, Qa'im Bekkali.

Call: AAP Région Nouvelle-Aquitaine.
Dates: 2024-2028
Project title: Modélisation multi-échelle des écoulements multiphasiques

Understanding and mastering complex and physically rich flows, such as compressible multiphase flows, often unsteady, is of great importance in various fields such as aeronautics, automotive, aerospace, nuclear, naval, and even medicine.

It is observed that very simple resolved interface flow configurations can quickly give rise to flows containing very small inclusions (bubbles, droplets), leading to intrinsically multi-scale flows. This multi-scale nature presents challenges for both the modeling and simulation of such phenomena.

The goal of this project is to contribute to improving models and numerical methods for simulating compressible multiphase flows. The guiding principle of this project will be the modeling of subscale phenomena using a stochastic process.

The project will be carried out in three stages:

1. First Stage: We will develop a 1D *ab initio* simulation tool in which the interfaces will be fully resolved. The phase distribution will be determined by drawing from a stochastic process that satisfies macroscopic constraints. The goal is to compute the mean flow limit obtained by averaging over

numerous draws of the stochastic process and to relate the properties of the mean flow to the prescribed macroscopic quantities.

2. Second Stage: We will develop both a 1D model and a numerical scheme based on the same stochastic approach. The objective is to establish a model and numerical scheme capable of directly simulating the mean flow. We will mathematically study the robustness of the numerical scheme. This scheme will be implemented and compared to the numerical results obtained with the *ab initio* simulations from the first stage.

3. Third Stage: We will extend the model and numerical method to 2D and 3D spatial dimensions and implement the method. The results will be compared to existing numerical and/or experimental results in three types of configurations:

   - Study of interface instabilities governing fragmentation processes (*e.g.*, Rayleigh-Taylor, Kelvin-Helmholtz). These flows start with well-resolved interfaces that deteriorate into increasingly smaller inclusions, requiring modeling.
   - Study of the interaction between a wave and a cloud of bubbles, involving a significant response of the dilute phase.
   - Study of cavitation induced by a shock wave in a droplet, where interactions occur between waves and interfaces (at the mesh scale) as well as with mixtures (cavitation bubbles at the submesh scale). Unlike the first two configurations, numerous scales are involved within a single configuration.

11 Dissemination

Participants: Rémi Manceau, Jonathan Jung, Vincent Perrier, Kevin Schmidmayer, Pascal Bruel.

11.1 Promoting scientific activities

11.1.1 Scientific events: organisation

- V. Perrier organized a minisymposium at the conference ICOSAHOM 2025

Member of the organizing committees

- Member of the scientific committee of the International Symposium on Turbulence, Heat and Mass Transfer since 2006 [R. Manceau]

Reviewer

- Reviewing of 6 papers for International Symposium on Turbulence, Heat and Mass Transfer since 2006 [R. Manceau]

- Reviewing of 3 papers for CFM (Congrès français de mécanique, Metz, 2025) [R. Manceau]

11.1.2 Journal

Member of the editorial boards

- Advisory Board of International Journal of Heat and Fluid Flow [R. Manceau]

- Advisory Board of Flow, Turbulence and Combustion [R. Manceau]

Reviewer - reviewing activities

- Journal Fluid Mechanics [3]
- Physical review fluids [1]
- Journal of Computational Physics [3]
- International Journal of Heat and Fluid Flow [1]
- Applied thermal engineering [1]
- Nuclear Eng. Design [1]
- European Journal of Mechanics / B Fluids [1]
- Computer and Fluids [2]
- Proceedings of the International Conference Zaragoza - Pau on Mathematics and its Applications [1]

11.1.3 Invited talks

[15] R. Manceau. 'The active approach of hybrid RANS/LES modeling for continuous embedded LES'. in: Lille turbulence programme 2025, Opening workshop on turbulent flows. Lille, France, 17th June 2025. URL: https://inria.hal.science/hal-05137465

[18] K. Schmidmayer, F. Petitpas and N. Favrie. 'ECOGEN: An open-source software for simulation of cavitation bubble dynamics'. In: Workshop on Bubbles in Complex Media and Confinements. Monte Verità, Switzerland, 18th June 2025. URL: https://inria.hal.science/hal-05124535

[17] V. Perrier and J. Jung. 'Conservation of divergence or rotational type constraints of hyperbolic systems with the discontinuous Galerkin method.' In: Journées Ondes du Sud Ouest. Pau, France, 18th Mar. 2025. URL: https://hal.science/hal-05465215

[16] V. Perrier. 'Méthodes de Galerkin discontinu d'ordre élevé préservant les contraintes différentielles d'un système hyperbolique'. In: Journée du LRC ANABASE 2024. Saint-Médard d'Eyrans, France, 17th Jan. 2025. URL: https://hal.science/hal-05465156

11.1.4 Leadership within the scientific community

- R. Manceau co-chairs the Standing committee (S. Jakirlić, F. Menter, S. Wallin, D. von Terzi , B. Launder, K. Hanjalić, W. Rodi, M. Leschziner, D. Laurence) of the Special Interest Group *Turbulence modelling* (SIG-15) of ERCOFTAC with S. Jakirlć. The main activities of the group are to organize international workshops and thematic sessions in international congresses.
- K. Schmidmayer co-pilots the working group "Valorisation et durabilité" of the Software and Source Codes College of the Committee for Open Science from the Ministère de l'enseignement supérieur et de la recherche.
- Vincent Perrier coordinates the ANR Project LAGOON, a 4-year project started in 2022. The partners are: the CARDAMOM project-team of Inria Bordeaux and the BRGM.

11.1.5 Scientific expertise

- Review for an ANR ASTRID project [K. Schmidmayer (1)]
- Review of one ANR project (Vincent Perrier)

11.1.6 Research administration

- Member of the LMAP council [Rémi Manceau].
- Member of the CDT, in charge of the evaluation of software projects at the Inria Bordeaux center [Vincent Perrier].
- Member of the CT3-Num committee of Pau University, in charge of managing the computing resources and projects at Pau University [Vincent Perrier].
- Member of the comité des utilisateurs des moyens de calcul at INRIA [Vincent Perrier]
- Co-head of Comité Parité et Égalité des Chances, since 2025 (Vincent Perrier)
- Vincent Perrier has been appointed as a member of the Mathematics evaluation panel of the Swiss National Science Foundation for 2 years (2025-2027)
- Member of the National Council of Universities section 26 (CNU 26) [Jonathan Jung]

11.2 Teaching - Supervision - Juries - Educational and pedagogical outreach

11.2.1 Teaching

(Legend: L1-L2-L3 corresponds to the 3 years of undergraduate studies, leading to the BSc degree; M1-M2 to the 2 years of graduate studies, leading to the MSc degree; E1-E2-E3 to the 3 years of engineering school, equivalent to L3-M1-M2, leading to the engineer/MSc degree)

- L1 [J. Jung]: Research and innovation (lectures: 1.5h/year), Université de Pau et des Pays de l'Adour, Pau, France.
- L1 [J. Jung]: Mathematical Algorithms 1 and Python (lectures: 9h/year) Mathematics, University of Pau (UPPA).
- L2 [J. Jung]: Numerical analysis for vectorial problems (lectures: 10.5h/year), Mathematics, Université de Pau et des Pays de l'Adour, Pau, France.
- L2 [J. Jung]: Scientific computing (labs: 58.5h/year), Informatics, University of Pau (UPPA).
- M1 [J. Jung]: Tools for scientific computing (lectures: 9.75/year, labs: 9h75/year), MMS, Université de Pau et des Pays de l'Adour, Pau, France.
- M2 [R. Manceau]: Turbulence modelling (in English), 27h30/year, International Master program Turbulence, ISAE-ENSMA/École centrale de Lille, France.
- E3 [R. Manceau]: Industrial codes for CFD (in English), 12h30/year, ISAE-ENSMA, Poitiers, France [38, 37].
- E3 [R. Manceau]: Advanced physics–Turbulence modelling for CFD, 16h/year, ENSGTI, France [39, 40].
- M1 [K. Schmidmayer]: Introduction to Python, 32h/year, Master MSID, Pau, France.
- E2 [K. Schmidmayer]: Coupled heat transfer, 12h/year, ENSGTI, Pau, France.

11.2.2 Supervision

- Defended in 2025: Romaric Simo Tamou, “Development of high-order methods in a Cartesian AMR/Cutcell code. Application to LES modelling of combustion”, IFPEN, E2S-UPPA Asturies project, Vincent Perrier.
- Defended in 2025: Esteban Coiffier, "Analyse et simulation numériques de discrétisations décalées en thermohydraulique diphasique", CEA-Saclay, Vincent Perrier & Jonathan Jung.
- PhD in progress: Jules Mazaleyrat, "Modélisation numérique d’une turbine refroidie par impact de jets : dérivation de modèles RANS adaptés sur la base d’une approche LES”, SAFRAN HE and ONERA, Rémi Manceau.
- PhD in progress: Corina Sanz Souhait, “Industrialisation des modèles RANS avancés avec transferts thermiques pour la convection forcée, mixte et naturelle”, EDF, Rémi Manceau.
- PhD in progress: Joséphine Gauthier, “Modélisation de l’écoulement turbulent et laminaire dans les bras morts avec ou sans coudes”, EDF, Rémi Manceau.
- PhD in progress: Adedotun Ade, “Numerical modelling of cavitation bubbles interacting with biomaterials”, Inria, Vincent Perrier & Kevin Schmidmayer.
- PhD in progress: Lucas Martin de Fourchambault, “Simulation numérique de l’atténuation des effets d’une explosion par une mousse aqueuse”, co-supervised with IUSTI (Aix-Marseille Université) and CEA Gramat, Kevin Schmidmayer.
- PhD in progress: Felice Edoardo Taglialatela, “Large-Eddy Simulation of compressible multiphase flows: algorithm development and application to aerobreakup phenomena”, co-supervised with the University of Campania “Luigi Vanvitelli", Italia, Kevin Schmidmayer.
- PhD in progress: Qa’im Bekkali, “Development of numerical strategies for the computation of multiscale compressible multiphase flows”, Vincent Perrier & Kevin Schmidmayer.

11.2.3 Juries

- Reviewer of the PhD thesis of A. Monot, École Centrale de Nantes [R. Manceau]
- President of the jury of the PhD thesis of Basile Desmolin, ISAE-Supaero, Toulouse [R. Manceau]
- Reviewer of the PhD thesis of A. Tardieu, Université de Bordeaux [V. Perrier]
- President of the jury of the PhD thesis of Solène Schropff, IUSTI, Aix-Marseille Université, Marseille [V. Perrier]

11.3 Popularization

11.3.1 Participation in Live events

[36] K. Schmidmayer and L. Martin de Fourchambault. ‘Lithotripsie’. In: Nuit de la recherche 2025. Pau, France, 26th Sept. 2025. URL: https://hal.science/hal-05286994

[35] R. Manceau. ‘Comment reproduire un iceberg ou une centrale nucléaire en laboratoire : les paramètres de similitude en dynamique des fluides’. In: Nuit de la recherche 2025. Pau, France, 26th Sept. 2025. URL: https://inria.hal.science/hal-05290074

12 Scientific production

12.1 Major publications

[1] L. Biasiori-Poulanges and K. Schmidmayer. 'A phenomenological analysis of droplet shock-induced cavitation using a multiphase modelling approach'. In: *Physics of Fluids* 35 (6th Jan. 2023), p. 013312. DOI: 10.1063/5.0127105. URL: https://hal.archives-ouvertes.fr/hal-03894523.

[2] P. Bruel, S. Delmas, J. Jung and V. Perrier. 'A low Mach correction able to deal with low Mach acoustics'. In: *Journal of Computational Physics* 378 (2019), pp. 723–759. URL: https://hal.inria.fr/hal-01953424.

[3] S. Dellacherie, J. Jung, P. Omnes and P.-A. Raviart. 'Construction of modified Godunov type schemes accurate at any Mach number for the compressible Euler system'. In: *Mathematical Models and Methods in Applied Sciences* (Nov. 2016). DOI: 10.1142/S0218202516500603. URL: https://hal.archives-ouvertes.fr/hal-00776629.

[4] V. Duffal, B. De Laage De Meux and R. Manceau. 'Development and Validation of a new formulation of Hybrid Temporal Large Eddy Simulation'. In: *Flow, Turbulence and Combustion* (2021). DOI: 10.1007/s10494-021-00264-z. URL: https://hal.inria.fr/hal-03206747.

[5] J.-L. Florenciano and P. Bruel. 'LES fluid-solid coupled calculations for the assessment of heat transfer coefficient correlations over multi-perforated walls'. In: *Aerospace Science and Technology* 53 (2016), p. 13. DOI: 10.1016/j.ast.2016.03.004. URL: https://hal.inria.fr/hal-01353952.

[6] E. Franquet and V. Perrier. 'Runge-Kutta discontinuous Galerkin method for the approximation of Baer and Nunziato type multiphase models'. In: *Journal of Computational Physics* 231.11 (Feb. 2012), pp. 4096–4141. DOI: 10.1016/j.jcp.2012.02.002. URL: https://hal.inria.fr/hal-00684427.

[7] J.-M. Hérard and J. Jung. 'An interface condition to compute compressible flows in variable cross section ducts'. In: *Comptes Rendus Mathématiques* (Feb. 2016). DOI: 10.1016/j.crma.2015.10.026. URL: https://hal.inria.fr/hal-01233251.

[8] R. Manceau. 'Recent progress in the development of the Elliptic Blending Reynolds-stress model'. In: *International Journal of Heat and Fluid Flow* (2015), p. 32. DOI: 10.1016/j.ijheatfluidflow.2014.09.002. URL: https://inria.hal.science/hal-01092931.

[9] G. Mangeon, S. Benhamadouche, J.-F. Wald and R. Manceau. 'Extension to various thermal boundary conditions of the elliptic blending model for the turbulent heat flux and the temperature variance'. In: *Journal of Fluid Mechanics* 905.A1 (Dec. 2020), pp. 1–34. DOI: 10.1017/jfm.2020.683. URL: https://hal.inria.fr/hal-02974557.

[10] Y. Moguen, S. Delmas, V. Perrier, P. Bruel and E. Dick. 'Godunov-type schemes with an inertia term for unsteady full Mach number range flow calculations'. In: *Journal of Computational Physics* 281 (Jan. 2015), p. 35. DOI: 10.1016/j.jcp.2014.10.041. URL: https://hal.inria.fr/hal-01096422.

12.2 Publications of the year

International journals

[11] A. Ferre, S. Serra, J. Pouvreau, R. Manceau, A. Bruch, O. Soriano and M. Bois. 'Experimental study of thermocline behavior under high propagation velocities in a water based thermal energy storage'. In: *Journal of Energy Storage* 125.116921 (9th May 2025). DOI: 10.1016/j.est.2025.116921. URL: https://inria.hal.science/hal-05063852 (cit. on p. 9).

[12] J. Jung, I. Lannabi and V. Perrier. 'On the spurious modes associated with the pressure centred low Mach number fix for compressible flows'. In: *Computers and Fluids* 306 (4th Dec. 2025), p. 106945. DOI: 10.1016/j.compfluid.2025.106945. URL: https://hal.science/hal-05465724 (cit. on p. 20).

[13] V. Perrier. 'Development of discontinuous Galerkin methods for hyperbolic systems that preserve a curl or a divergence constraint: the case of linear systems'. In: *Journal of Computational Physics* 544 (10th Oct. 2025), p. 114445. DOI: 10.1016/j.jcp.2025.114445. URL: https://inria.hal.science/hal-04564886 (cit. on p. 21).

[14] V. Perrier. 'discrete de-Rham complex involving a discontinuous finite element space for velocities: the case of periodic straight triangular and Cartesian meshes'. In: *Annales Henri Lebesgue* 8 (2025), pp. 417–452. DOI: 10.5802/ahl.239. URL: https://inria.hal.science/hal-04564069 (cit. on p. 21).

Invited conferences

[15] R. Manceau. 'The active approach of hybrid RANS/LES modeling for continuous embedded LES'. In: Lille turbulence programme 2025, Opening workshop on turbulent flows. Lille, France, 17th June 2025. URL: https://inria.hal.science/hal-05137465 (cit. on pp. 19, 33).

[16] V. Perrier. 'Méthodes de Galerkin discontinu d'ordre élevé préservant les contraintes différentielles d'un système hyperbolique'. In: Journée du LRC ANABASE 2024. Saint-Médard d'Eyrans, France, 17th Jan. 2025. URL: https://hal.science/hal-05465156 (cit. on pp. 21, 33).

[17] V. Perrier and J. Jung. 'Conservation of divergence or rotational type constraints of hyperbolic systems with the discontinuous Galerkin method.' In: Journées Ondes du Sud Ouest. Pau, France, 18th Mar. 2025. URL: https://hal.science/hal-05465215 (cit. on pp. 21, 33).

[18] K. Schmidmayer, F. Petitpas and N. Favrie. 'ECOGEN: An open-source software for simulation of cavitation bubble dynamics'. In: Workshop on Bubbles in Complex Media and Confinements. Monte Verità, Switzerland, 18th June 2025. URL: https://inria.hal.science/hal-05124535 (cit. on p. 33).

International peer-reviewed conferences

[19] P. Bikkanahally and R. Manceau. 'A modelling strategy for log-layer mismatch in channel flows'. In: THMT 2025 - 11th International Symposium on Turbulence, Heat and Mass Transfer. Tokyo, Japan, 21st July 2025. URL: https://inria.hal.science/hal-05245470 (cit. on p. 19).

[20] P. Bikkanahally and R. Manceau. 'Development of a novel strategy for embedded LES based on continuous hybrid RANS/LES methods'. In: 15th Int. ERCOFTAC Symp. on Eng. Turb. Modelling and Measurements. Dubrovnik, Croatia, 2025. URL: https://inria.hal.science/hal-05411500 (cit. on p. 19).

[21] C. Sanz Souhait, J.-F. Wald, S. Benhamadouche and R. Manceau. 'Second moment closure for thermal fluxes for forced and mixed convection'. In: 21st Int. topical meeting on nuclear reactor thermal hydraulics (NURETH-21). Busan, South Korea, 2025. URL: https://inria.hal.science/hal-05411398 (cit. on pp. 17, 18).

National peer-reviewed Conferences

[22] P. Bikkanahally and R. Manceau. 'Towards a model for continuous embedded LES (CELES)'. In: CFM 2025 - 26ème Congrès Français de Mécanique. Metz, France, 25th Aug. 2025. URL: https://inria.hal.science/hal-05245476 (cit. on p. 19).

Conferences without proceedings

[23] E. Coiffier, J. Jung, M. Ndjinga and V. Perrier. 'Numerical staggered conservative scheme for the simulation of low Mach number flows'. In: NumHyp 2025 - Numerical Methods for Hyperbolic Problems. Darmstadt, Germany, June 2025. URL: https://hal.science/hal-05468321 (cit. on p. 21).

[24] E. Coiffier, J. Jung, M. Ndjinga and V. Perrier. ‘Schéma numérique décalé conservatif pour la simulation d’écoulements à bas nombre de Mach’. In: SMAI 2025 - 12ème Biennale de la Société des Mathématiques Appliquées et Industrielles. Carcans-Maubuisson (Gironde), France, 2025. URL: https://hal.science/hal-05468336 (cit. on p. 21).

[25] J. Jung and V. Perrier. ‘Development of discontinuous Galerkin methods That preserve a curl or a divergence Constraint’. In: The 15th international conference on spectral and high order methods (ICOSAHOM 2025). Montréal, France, 13th July 2025. URL: https://hal.science/hal-05465295 (cit. on p. 21).

[26] J. Jung and V. Perrier. ‘How to preserve a curl or a divergence constraint with the discontinuous Galerkin method’. In: 18th U.S. National Congress on Computational Mechanics (USNCCM). Chicago, United States, 20th July 2025. URL: https://hal.science/hal-05465629 (cit. on p. 21).

[27] J. Jung and V. Perrier. ‘On the conservation of curl or divergence constraints by the discontinuous Galerkin method’. In: hyPerbolIC models, numerical AnalysiS and Scientific cOmputation. Malaga, Spain, 24th Mar. 2025. URL: https://hal.science/hal-05465252 (cit. on p. 21).

[28] K. Schmidmayer, G. T. Bokman, A. Sieber, A. Ade-Onojobi, B. Lukic, G. Shakya, A. Kühl, M. Schlötter, M. Belau and O. Supponen. ‘Exploring the interplay of stone-attached and distant cavitation bubbles during extracorporeal shock wave lithotripsy’. In: ICMF 2025 - 12th International Conference on Multiphase Flow. Toulouse, France, 12th May 2025. URL: https://inria.hal.science/hal-05070922 (cit. on p. 9).

Doctoral dissertations and habilitation theses

[29] E. Coiffier. ‘Numerical analysis and simulation of staggered schemes for low Mach number flows’. Université de Pau et des Pays de l ’Adour, 8th Dec. 2025. URL: https://hal.science/tel-05465927 (cit. on p. 21).

[30] R. Simo Tamou. ‘Development of high order methods in a Cartesian/AMR solver for combustion modeling’. Université de Pau et des Pays de l ’Adour, 17th Mar. 2025. URL: https://hal.science/tel-05465982 (cit. on p. 20).

Other scientific publications

[31] D. Inzunza and V. Perrier. ‘Assessment of multigrid preconditioners for solving systems induced by ImEx methods’. In: Numerical methods for hyperbolic problems (NumHyp25). Darmstadt, Germany, 9th June 2025. URL: https://hal.science/hal-05465817 (cit. on p. 21).

[32] J. Jung and V. Perrier. ‘A curl preserving finite volume scheme by space velocity enrichment. Application to the low Mach number accuracy problem’. In: numhyp25 - Numerical Methods for Hyperbolic Problems. Darmstadt, Germany, 9th June 2025. URL: https://hal.science/hal-05117229 (cit. on p. 20).

[33] R. Manceau. *Une approche active de la modélisation hybride RANS/LES pour la LES embarquée continue*. 12th Nov. 2025. URL: https://inria.hal.science/hal-05463034 (cit. on p. 19).

[34] K. Schmidmayer. *Towards a unified framework: Modelling and simulating multiphase compressible flows with cavitation and viscoplastic solids*. 17th June 2025. URL: https://inria.hal.science/hal-05124543 (cit. on p. 23).

Scientific popularization

[35] R. Manceau. ‘Comment reproduire un iceberg ou une centrale nucléaire en laboratoire : les paramètres de similitude en dynamique des fluides’. In: Nuit de la recherche 2025. Pau, France, 26th Sept. 2025. URL: https://inria.hal.science/hal-05290074 (cit. on p. 35).

[36] K. Schmidmayer and L. Martin de Fourchambault. ‘Lithotripsie’. In: Nuit de la recherche 2025. Pau, France, 26th Sept. 2025. URL: https://hal.science/hal-05286994 (cit. on p. 35).

Educational activities

[37] R. Manceau. 'Codes de calcul industriels pour la simulation des écoulements turbulents'. Master. France, 2nd Feb. 2026. URL: https://inria.hal.science/hal-03207435 (cit. on p. 34).

[38] R. Manceau. 'Industrial codes for CFD'. Master. France, 2nd Feb. 2026. URL: https://inria.hal.science/hal-03207431 (cit. on p. 34).

[39] R. Manceau. 'Modelisation de la turbulence pour la CFD'. Master. France, 14th Oct. 2025. URL: https://inria.hal.science/hal-03207437 (cit. on p. 34).

[40] R. Manceau. 'Turbulence modelling for CFD'. Master. France, 14th Oct. 2025. URL: https://inria.hal.science/hal-03207433 (cit. on p. 34).

Software

[41] [SW] K. Schmidmayer, F. Petitpas, E. Daniel, J. Cazé, A. Froehly, B. Dorschner, S. Le Martelot, N. Favrie, F. Z. Gadiri, S. Schropff and A. Ade-Onojobi, *ECOGEN* version 5.0, 7th Apr. 2025. LIC: GNU General Public License v3.0 or later. HAL: ⟨hal-05023172⟩, URL: https://hal.science/hal-05023172, VCS: https://github.com/code-mphi/ECOGEN, SWHID: ⟨swh:1:dir:75944d477a18b5e3039c74e96158df001e935d02⟩ (cit. on pp. 24–26, 31).

12.3 Cited publications

[42] L. Biasiori-Poulanges and K. Schmidmayer. 'A phenomenological analysis of droplet shock-induced cavitation using a multiphase modelling approach'. In: *Physics of Fluids* 35 (6th Jan. 2023), p. 013312. DOI: 10.1063/5.0127105. URL: https://hal.science/hal-03894523 (cit. on pp. 22, 26).

[43] M. David, M. Mehta and R. Manceau. 'On the feasibility of a self-adaptive strategy for hybrid RANS/LES based on physical criteria and its inital testing on low Reynolds number backward-facing step flow'. In: *Flow, Turbulence and Combustion* (2024). DOI: 10.1007/s10494-024-00583-x. URL: https://hal.science/hal-04396836 (cit. on p. 19).

[44] S. Elaskar, P. Bruel and L. Gutiérrez Marcantoni. 'Random Telegraphic Signals with Fractal-like Probability Transition Rates'. In: *Symmetry* 16.9 (8th Sept. 2024), p. 1175. DOI: 10.3390/sym16091175. URL: https://inria.hal.science/hal-04695142 (cit. on p. 19).

[45] T. Galié, J. Jung, I. Lannabi and V. Perrier. 'Extension of an all-Mach Roe scheme able to deal with low Mach acoustics to full Euler system'. In: *ESAIM: Proceedings and Surveys* 76 (2024), pp. 35–51. DOI: 10.1051/proc/202476035. URL: https://univ-pau.hal.science/hal-04164990 (cit. on p. 20).

[46] S. K. Jena and R. Manceau. 'Dynamics of turbulent natural convection in a cubic cavity with centrally placed partially heated inner obstacle'. In: *Physics of Fluids* 38.8 (2024). DOI: 10.1063/5.0222258. URL: https://inria.hal.science/hal-04652604 (cit. on p. 18).

[47] J. Jung and V. Perrier. 'A curl preserving finite volume scheme by space velocity enrichment. Application to the low Mach number accuracy problem'. In: *Journal of Computational Physics* 515 (21st Mar. 2024), p. 113252. DOI: 10.1016/j.jcp.2024.113252. URL: https://inria.hal.science/hal-04563838 (cit. on p. 21).

[48] J. Jung and V. Perrier. 'Behavior of the Discontinuous Galerkin Method for Compressible Flows at Low Mach Number on Triangles and Tetrahedrons'. In: *SIAM Journal on Scientific Computing* 46.1 (5th Feb. 2024), A452–A482. DOI: 10.1137/23M154755X. URL: https://hal.science/hal-04503238 (cit. on pp. 20, 21).

[49] M. Mehta, R. Manceau, V. Duffal and B. de Laage de Meux. 'An active hybrid Reynolds-Averaged Navier-Stokes/Large Eddy Simulation approach for grey area mitigation'. In: *Physics of Fluids* 35 (2023). DOI: 10.1063/5.0174381. URL: https://inria.hal.science/hal-04287062 (cit. on pp. 7, 19).

[50] K. Schmidmayer and L. Biasiori-Poulanges. 'Geometry effects on the droplet shock-induced cavitation'. In: *Physics of Fluids* 35.6 (6th June 2023), p. 063315. DOI: 10.1063/5.0151404. URL: https://hal.science/hal-04100852 (cit. on p. 22).

[51] P. Bikkanahally and R. Manceau. 'Modelling turbulent flows in the natural convection regime using hybrid RANS-LES approaches'. In: 17th ERCOFTAC SIG15/MONACO2025 workshop: Turbulent natural convection flows in differentially heated cavities. Pau, France, 19th Jan. 2023. URL: https://inria.hal.science/hal-03970569 (cit. on p. 18).

[52] V. Duffal, R. Manceau and B. de Laage de Meux. 'Development and Validation of a new formulation of Hybrid Temporal Large-Eddy Simulation'. In: 17th ERCOFTAC SIG15/MONACO2025 workshop: Turbulent natural convection flows in differentially heated cavities. Pau, France, 19th Jan. 2023. URL: https://inria.hal.science/hal-03972235 (cit. on p. 18).

[53] R. Manceau. 'Modélisation de la turbulence en convection naturelle (conférence introductive)'. In: Journées thématiques de la SFT. Convection naturelle : aspects fondamentaux et applications. Orsay, France, 3rd July 2023. URL: https://inria.hal.science/hal-04151728 (cit. on p. 18).

[54] V. Perrier. 'A fully closed one dimensional model for two-phase flows'. In: Sixth Workshop on Compressible Multiphase Flows. Strasbourg (67000), France, 17th June 2024. URL: https://inria.hal.science/hal-04884348 (cit. on p. 22).

[55] M. David, M. Mehta and R. Manceau. 'Towards self-adaptivity in hybrid RANS/LES based on physical criteria'. In: THMT 2023 - 10th International Symposium on Turbulence, Heat and Mass Transfer. Rome, Italy, 11th Sept. 2023. URL: https://hal.science/hal-04210071 (cit. on p. 7).

[56] S. Elaskar and P. Bruel. 'Ecuaciones de Navier-Stokes de hombre pobre applicadas a llamas premezcladas'. In: *Matemática Aplicada, Computacional e Industrial (MACI, ISSN: 2314-3282)*. MACI 2023 - IX Congreso de Matemática Aplicada, Computacional e Industrial. Vol. 9. Santa Fe, Argentina, 8th May 2023. URL: https://inria.hal.science/hal-04186270 (cit. on p. 19).

[57] A. Ferre, J. Pouvreau, S. Serra, R. Manceau and A. Bruch. 'CFD study of thermocline formation in stratified water storage: Consideration of a second-order Boussinesq approximation to model buoyancy effects and its application to assess the impact of operating conditions'. In: *IHTC-17 | ID: 435*. Proceedings of the 17th International Heat Transfer Conference, IHTC-17 14 – 18 August 2023, Cape Town, South Africa. Cape Town, South Africa, 14th Aug. 2023. URL: https://hal.science/hal-04145083 (cit. on p. 27).

[58] J. Jung, I. Lannabi and V. Perrier. 'On the convergence of Godunov scheme with a centered discretization of the pressure gradient'. In: FVCA X 2023 - Finite Volumes for Complex Applications X. Strasbourg, France, 13th Oct. 2023. URL: https://hal.science/hal-04260535 (cit. on p. 20).

[59] J.-F. Wald, G. Mangeon, S. Benhamadouche, R. Manceau and C. Flageul. 'Conjugate heat transfer with different fluid-solid physical properties: a differential flux model based on elliptic blending'. In: THMT 2023 - 10th International Symposium on Turbulence, Heat and Mass Transfer. Rome, Italy, 11th Sept. 2023. URL: https://inria.hal.science/hal-04203590 (cit. on p. 10).

[60] A. Abbà, M. Germano and M. Nini. 'A RANS assisted LES approach'. In: *Direct and Large-Eddy Simulation XI*. Ed. by M. Salvetti, V. Armenio, J. Fröhlich, B. Geurts and H. Kuerten. Vol. 25. ERCOFTAC Series. Springer, Cham, 2019, pp. 159–165 (cit. on p. 7).

[61] A. H. Afailal, J. Galpin, A. Velghe and R. Manceau. 'Development and validation of a hybrid temporal LES model in the perspective of applications to internal combustion engines'. In: *Oil & Gas Science and Technology - Revue d'IFP Energies nouvelles* 74 (2019), p. 56. DOI: 10.2516/ogst/2019031. URL: https://hal-ifp.archives-ouvertes.fr/hal-02162596 (cit. on p. 7).

[62] J.-A. Bäckar and L. Davidson. 'Evaluation of numerical wall functions on the axisymmetric impinging jet using OpenFOAM'. In: *Int. J. Heat Fluid Fl.* 67 (2017), pp. 27–42 (cit. on p. 7).

[63] M. R. Baer and J. W. Nunziato. 'A two-phase mixture theory for the deflagration-to-detonation transition (DDT) in reactive granular materials'. In: *International journal of multiphase flow* 12.6 (1986), pp. 861–889 (cit. on p. 22).

[64] F. Ballanger. 'Confinement de la détonation d'un objet explosif par mousse aqueuse sèche. Etude expérimentale et numérique'. PhD thesis. ISAE-ENSMA Ecole Nationale Supérieure de Mécanique et d'Aérotechique-Poitiers, 2016 (cit. on pp. 25, 31).

[65] S. Benhamadouche, R. Howard and R. Manceau, eds. *Proc. 15th ERCOFTAC (SIG-15)/IAHR Workshop on Refined Turbulence Modelling*. EDF Chatou, France. 2011 (cit. on p. 9).

[66] S. Benhamadouche, I. Afgan and R. Manceau. 'Numerical simulations of flow and heat transfer in a wall-bounded pin matrix'. In: *Flow, Turbulence and Combustion* 104.1 (2020), pp. 19–44. DOI: 10.1007/s10494-019-00046-8. URL: https://hal.archives-ouvertes.fr/hal-02179021 (cit. on p. 17).

[67] L. Biasiori-Poulanges, C. Bourquard, B. Lukic, L. Broche and O. Supponen. 'HIFU-induced cavitation bubble cloud dynamics and its role in model kidney stone erosion'. In: *Bulletin Am. Phis. Soc.* (2022) (cit. on p. 9).

[68] P. Bikkanahally, R. Manceau and F. Mastrippolito. 'Development of a hybrid RANS-LES model based on temporal filtering for natural convection flows'. In: *WCCM - 14th World Congress in Computational Mechanics, ECCOMAS Congress 2020*. Virtual, France, Jan. 2021. URL: https://hal.inria.fr/hal-03114394 (cit. on pp. 7, 18).

[69] G. T. Bokman, L. Biasiori-Poulanges, B. Lukić, K. Schmidmayer, C. Bourquard, E. Baumann, A. Rack, B. J. Olson and O. Supponen. 'Impulse-driven release of gas-encapsulated drops'. In: *J. Fluid Mech.* 1001 (2024), A51 (cit. on p. 23).

[70] R. Bourguet, M. Braza, G. Harran and R. El Akoury. 'Anisotropic Organised Eddy Simulation for the prediction of non-equilibrium turbulent flows around bodies'. In: *J. Fluid Struct.* 24.8 (2008), pp. 1240–1251 (cit. on p. 7).

[71] C. E. Brennen. 'Cavitation in medicine'. In: *Interface Focus* 5.5 (2015), p. 20150022 (cit. on p. 9).

[72] D. Bresch and M. Hillairet. 'A compressible multifluid system with new physical relaxation terms'. In: *Annales Scientifiques de l'École Normale Supérieure* 52.2 (2019), pp. 255–295 (cit. on p. 8).

[73] B. Chaouat and R. Schiestel. 'A new partially integrated transport model for subgrid-scale stresses and dissipation rate for turbulent developing flows'. In: *Phys. Fluids* 17.065106 (2005), pp. 1–19 (cit. on p. 7).

[74] A. Colombié, E. Laroche, F. Chedevergne, R. Manceau, F. Duchaine and L. Gicquel. 'Large-eddy-simulation-based analysis of Reynolds-stress budgets for a round impinging jet'. In: *Physics of Fluids* 33.11 (Nov. 2021), p. 115109. DOI: 10.1063/5.0064009. URL: https://hal-univ-pau.archives-ouvertes.fr/hal-03423068 (cit. on p. 17).

[75] F. Coquel, T. Gallouët, J.-M. Hérard and N. Seguin. 'Closure laws for a two-fluid two-pressure model'. In: *Comptes Rendus Mathematique* 334.10 (2002), pp. 927–932 (cit. on p. 22).

[76] B. de Laage de Meux, B. Audebert, R. Manceau and R. Perrin. 'Anisotropic Linear Forcing for synthetic turbulence generation in LES and hybrid RANS/LES modeling'. In: *Phys. Fluids* 27.035115 (2015). DOI: 10.1063/1.4916019. URL: hal-01246100 (cit. on p. 28).

[77] S. Deck. 'Recent improvements in the Zonal Detached Eddy Simulation (ZDES) formulation'. In: *Theor. Comput. Fluid Dyn.* 26.6 (2012), pp. 523–550 (cit. on p. 7).

[78] P. Degond and M. Tang. 'All speed scheme for the low Mach number limit of the isentropic Euler equations'. In: *Communications in Computational Physics* 10.1 (2011), pp. 1–31 (cit. on p. 8).

[79] F. Dehoux, S. Benhamadouche and R. Manceau. 'An elliptic blending differential flux model for natural, mixed and forced convection'. In: *International Journal of Heat and Fluid Flow* 63 (2017), p. 15. DOI: 10.1016/j.ijheatfluidflow.2016.09.003. URL: https://hal.inria.fr/hal-01391900 (cit. on p. 18).

[80] E. Del Prete, A. Chinnayya, L. Domergue, A. Hadjadj and J.-F. Haas. 'Blast wave mitigation by dry aqueous foams'. In: *Shock waves* 23.1 (2013), pp. 39–53 (cit. on pp. 25, 31).

[81] S. Dellacherie, P. Omnes and F. Rieper. 'The influence of cell geometry on the Godunov scheme applied to the linear wave equation'. In: *Journal of Computational Physics* 229.14 (2010), pp. 5315–5338 (cit. on p. 20).

[82] D. A. Drew and S. L. Passman. *Theory of multicomponent fluids*. Vol. 135. Springer Science & Business Media, 2006 (cit. on p. 22).

[83] V. Duffal. 'Development of a hybrid RANS-LES model for the prediction of unsteady loads at the wall'. Theses. Université de Pau et des Pays de l'Adour, Nov. 2020. URL: https://tel.archives-ouvertes.fr/tel-03038837 (cit. on p. 18).

[84] V. Duffal, B. de Laage de Meux and R. Manceau. 'A new formulation of hybrid temporal large-eddy simulation'. In: *ETMM 2021 - 13th International ERCOFTAC symposium on engineering, turbulence, modelling and measurements*. Rhodes / Virtua, Greece, Sept. 2021. URL: https://hal.inria.fr/hal-03366796 (cit. on p. 18).

[85] V. Duffal, B. de Laage de Meux and R. Manceau. 'Development and Validation of a new formulation of Hybrid Temporal Large Eddy Simulation'. In: *Flow, Turbulence and Combustion* 108 (2022), p. 42. DOI: 10.1007/s10494-021-00264-z. URL: https://inria.hal.science/hal-03206747 (cit. on pp. 7, 18).

[86] N. Favrie, K. Schmidmayer and J. Massoni. 'A multiphase irreversible-compaction model for granular-porous materials'. In: *Continuum Mech. Thermodyn.* 34.1 (2022), pp. 217–241 (cit. on p. 23).

[87] A. Ferré, R. Manceau, S. Serra, J. Pouvreau and A. Bruch. 'Etude CFD de l'impact des conditions d'injection pour un stockage thermocline en eau'. In: *SFT 2022 - 30ème Congrès de la Société Française Thermique SFT*. Valenciennes, France, May 2022. URL: https://hal.science/hal-03692645 (cit. on p. 27).

[88] C. Ferreira. 'Gene expression programming: a new adaptive algorithm for solving problems'. In: *Complex Systems* 13.2 (2001), pp. 87–129 (cit. on p. 26).

[89] E. Franquet and V. Perrier. 'Runge–Kutta discontinuous Galerkin method for the approximation of Baer and Nunziato type multiphase models'. In: *Journal of Computational Physics* 231.11 (2012), pp. 4096–4141 (cit. on p. 22).

[90] M. Germano. 'Properties of the hybrid RANS/LES filter'. In: *Theor. Comput. Fluid Dyn.* 17.4 (2004), pp. 225–231 (cit. on p. 7).

[91] H. Guillard and C. Viozat. 'On the behaviour of upwind schemes in the low Mach number limit'. In: *Computers & fluids* 28.1 (1999), pp. 63–86 (cit. on p. 8).

[92] A. L. Gurson. 'Continuum theory of ductile rupture by void nucleation and growth: Part I–Yield criteria and flow rules for porous ductile media'. In: *J. Eng. Mater. Technol.* 99.1 (1977), pp. 2–15 (cit. on p. 23).

[93] M. Hantke, S. Müller and L. Grabowsky. 'News on Baer–Nunziato-type model at pressure equilibrium'. In: *Continuum Mechanics and Thermodynamics* 33.3 (2021), pp. 767–788 (cit. on p. 8).

[94] F. H. Harlow, J. E. Welch et al. 'Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface'. In: *Physics of fluids* 8.12 (1965), p. 2182 (cit. on p. 20).

[95] S. Jakirlić and R. Maduta. 'Extending the bounds of 'steady' RANS closures: Toward an instability-sensitive Reynolds stress model'. In: *Int. J. Heat Fluid Fl.* 51 (2015), pp. 175–194 (cit. on p. 7).

[96] S. M. S. Jameel. 'Turbulence modelling of mixed and natural convection regimes in the context of the underhood-space of automobiles'. Theses. Université de Pau et des Pays de l'Adour, Dec. 2020. URL: https://hal.inria.fr/tel-03128259 (cit. on p. 18).

[97] S. M. S. Jameel, R. Manceau and V. Herbert. 'A buoyancy extension for eddy-viscosity models for the natural convection regime'. In: *17th European Turbulence Conference (ETC-2019)*. Torino, Italy, Sept. 2019. URL: https://hal.archives-ouvertes.fr/hal-02305788 (cit. on p. 18).

[98] S. M. S. Jameel, R. Manceau and V. Herbert. 'Sensitization of eddy-viscosity models to buoyancy effects for predicting natural convection flows'. In: *HEFAT 2019 - 14th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics*. Proc. HEFAT 2019 - 14th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics. Wicklow, Ireland, July 2019. URL: https://hal.archives-ouvertes.fr/hal-02129153 (cit. on p. 18).

[99] S. K. Jena, P. Bikkanahally and R. Manceau. 'Numerical simulation of a turbulent natural convection flow in a cubic cavity with centrally placed partially heated inner obstacle'. In: *3rd high-fidelity industrial LES/DNS symp. (HiFiLeD)*. Brussels, Belgium, Dec. 2022. URL: https://inria.hal.science/hal-03928060 (cit. on p. 18).

[100] J. Jung and V. Perrier. 'Steady low Mach number flows: identification of the spurious mode and filtering method'. In: *Journal of Computational Physics* (July 2022), p. 111462. DOI: 10.1016/j.jcp.2022.111462. URL: https://hal.science/hal-03736193 (cit. on p. 20).

[101] V. A. Khokhlova, J. B. Fowlkes, W. W. Roberts, G. R. Schade, Z. Xu, T. D. Khokhlova, T. L. Hall, A. D. Maxwell, Y.-N. Wang and C. A. Cain. 'Histotripsy methods in mechanical disintegration of tissue: Towards clinical applications'. In: *Int. J. hyperthermia* 31.2 (2015), pp. 145–162 (cit. on p. 8).

[102] L. E. Kidder, S. E. Field, F. Foucart, E. Schnetter, S. A. Teukolsky, A. Bohn, N. Deppe, P. Diener, F. Hébert, J. Lippuner et al. 'SpECTRE: A task-based discontinuous Galerkin code for relativistic astrophysics'. In: *Journal of Computational Physics* 335 (2017), pp. 84–114 (cit. on p. 8).

[103] R. Manceau. 'Contra-rotating jets: wake/mixing layer interaction'. In: *Proc. 10th ERCOFTAC (SIG-15)/IAHR/QNET-CFD Workshop on Refined Turbulence Modelling*. Ed. by R. Manceau, J.-P. Bonnet, M. A. Leschziner and F. Menter. Laboratoire d'études aérodynamiques, UMR CNRS 6609, Université de Poitiers, France. 2002 (cit. on p. 9).

[104] R. Manceau. 'Progress in Hybrid Temporal LES (invited keynote paper)'. In: *Papers contributed to the 6th Symp. Hybrid RANS-LES Methods, 26–28 September 2016, Strasbourg, France*. Ed. by Y. Hoarau, S.-H. Peng, D. Schwamborn and A. Revell. Vol. 137. Notes on Numerical Fluid Mechanics and Multidisciplinary Design. Springer, 2018, pp. 9–25. DOI: 10.1007/978-3-319-70031-1_2. URL: hal-01391899 (cit. on p. 7).

[105] R. Manceau. 'Turbulent jet impinging onto a rotating disk: analysis of the RANS results'. In: *Proc. 13th ERCOFTAC (SIG-15)/IAHR Workshop on Refined Turbulence Modelling*. TU Graz, Austria. 2008. URL: hal-00385357 (cit. on p. 9).

[106] R. Manceau and K. Hanjalić. 'Elliptic Blending Model: A New Near-Wall Reynolds-Stress Turbulence Closure'. In: *Phys. Fluids* 14.2 (2002), pp. 744–754. DOI: 10.1063/1.1432693. URL: hal-02990466 (cit. on pp. 7, 17).

[107] R. Manceau. 'Modélisation des transferts thermiques turbulents (conférence plénière)'. In: *26e congrès français de thermique*. Pau, France, May 2018. URL: https://hal.inria.fr/hal-01944227 (cit. on p. 18).

[108] R. Manceau. 'Modelling of turbulent natural convection (keynote lecture)'. In: *16th ERCOFTAC SIG15 Workshop on Modelling of wall bounded turbulent natural convection*. Jozef Stefan Institute (IJS). Ljubljana, Slovenia, Oct. 2019. URL: https://hal.archives-ouvertes.fr/hal-02319389 (cit. on pp. 9, 18).

[109] R. Manceau. 'Recent progress in the development of the Elliptic Blending Reynolds-stress model'. In: *International Journal of Heat and Fluid Flow* (2015), p. 32. DOI: 10.1016/j.ijheatfluidflow.2014.09.002. URL: https://hal.inria.fr/hal-01092931 (cit. on pp. 7, 17).

[110] G. Mangeon, S. Benhamadouche, J.-F. Wald and R. Manceau. 'Extension to various thermal boundary conditions of the elliptic blending model for the turbulent heat flux and the temperature variance'. In: *Journal of Fluid Mechanics* 905.A1 (Dec. 2020), pp. 1–34. DOI: 10.1017/jfm.2020.683. URL: https://hal.inria.fr/hal-02974557 (cit. on p. 10).

[111] F. Mastrippolito, R. Manceau and P. Bruel. 'Modélisation des effets de giration pour les jets en écoulement transverse'. In: *Journee thematique SFT : Aerothermique des systèmes propulsifs pour l'aeronautique*. Paris, France, Oct. 2022. URL: https://inria.hal.science/hal-03836384 (cit. on p. 27).

[112] J. Mazars and G. Pijaudier-Cabot. 'Continuum damage theory-application to concrete'. In: *J. eng. mech.* 115.2 (1989), pp. 345–365 (cit. on p. 23).

[113] M. Mehta, R. Manceau, V. Duffal and B. de Laage de Meux. 'An active hybrid RANS/LES approach for grey area mitigation'. In: *DLES13 - Direct and Large Eddy Simulation*. Udine, Italy, Oct. 2022. URL: https://inria.hal.science/hal-03832703 (cit. on p. 19).

[114] M. Mikart. 'Capture, par mousse aqueuse, de particules micrométriques dispersées par explosif. Étude expérimentale et numérique'. PhD thesis. ISAE-ENSMA Ecole Nationale Supérieure de Mécanique et d'Aérotechique-Poitiers, 2019 (cit. on pp. 25, 31).

[115] Y. Moguen, T. Kousksou, P. Bruel, J. Vierendeels and E. Dick. 'Pressure–velocity coupling allowing acoustic calculation in low Mach number flow'. In: *Journal of Computational Physics* 231.16 (2012), pp. 5522–5541 (cit. on p. 8).

[116] C. Moussaed, S. Wornom, B. Koobus, A. Dervieux, T. Deloze, R. El Akoury, D. Szubert, Y. Hoarau and M. Braza. 'VMS and OES-based hybrid simulations of bluff body flows'. In: *Notes on Numerical Fluid Mechanics and Multidisciplinary Design* 133 (2016), pp. 293–308. DOI: 10.1007/978-3-319-27386-0_18 (cit. on p. 7).

[117] S. Ndanou, N. Favrie and S. Gavrilyuk. 'Multi-solid and multi-fluid diffuse interface model: Applications to dynamic fracture and fragmentation'. In: *Journal of Computational Physics* 295 (2015), pp. 523–555 (cit. on p. 23).

[118] V. Perrier and E. Gutiérrez. 'Derivation and Closure of Baer and Nunziato Type Multiphase Models by Averaging a Simple Stochastic Model'. In: *Multiscale Modeling and Simulation: A SIAM Interdisciplinary Journal* 19.1 (Jan. 2021), pp. 401–439. DOI: 10.1137/19M1306609. URL: https://hal.inria.fr/hal-03209659 (cit. on pp. 8, 22).

[119] F. Petitpas, R. Saurel, E. Franquet and A. Chinnayya. 'Modelling detonation waves in condensed energetic materials: Multiphase CJ conditions and multidimensional computations'. In: *Shock Waves* 19(5) (2009), pp. 377–401 (cit. on pp. 25, 31).

[120] Y. A. Pishchalnikov, W. M. Behnke-Parks, K. Schmidmayer, K. Maeda, T. Colonius, T. W. Kenny and D. J. Laser. 'High-speed video microscopy and numerical modeling of bubble dynamics near a surface of urinary stone'. In: *J. Acoust. Soc. Am.* 146 (2019), pp. 516–531 (cit. on p. 8).

[121] P. Reulet, E. Laroche, D. Donjat, F. Mastrippolito, R. Manceau and P. Bruel. *Parametric study of the effect of gyration on the flow and heat transfer of multiperforated plates*. SOPRANO : Soot processes and radiation in aeronautical innovative combustors. H2020 european project (D3.1). 2021 (cit. on pp. 9, 27).

[122] F. Rieper. 'A low-Mach number fix for Roe's approximate Riemann solver'. In: *Journal of Computational Physics* 230.13 (2011), pp. 5263–5287 (cit. on p. 8).

[123] P. S. Russell, D. R. Giosio, J. A. Venning, B. W. Pearce, P. A. Brandner and S. Ceccio. 'Microbubble generation from condensation and turbulent breakup of sheet cavitation'. In: *31st Symposium on Naval Hydrodynamics*. 31st Symposium on Naval Hydrodynamics. 2016, pp. 11–16 (cit. on p. 8).

[124] O. A. Sapozhnikov, A. D. Maxwell, B. MacConaghy and M. R. Bailey. 'A mechanistic analysis of stone fracture in lithotripsy'. In: *J. Acoust. Soc. Am.* 121.2 (2007), pp. 1190–1202 (cit. on p. 24).

[125] R. Saurel and R. Abgrall. 'A multiphase Godunov method for compressible multifluid and multiphase flows'. In: *Journal of Computational Physics* 150.2 (1999), pp. 425–467 (cit. on p. 22).

[126] R. Saurel and C. Pantano. 'Diffuse-interface capturing methods for compressible two-phase flows'. In: *Annual Review of Fluid Mechanics* 50 (2018), pp. 105–130 (cit. on p. 8).

[127] K. Schmidmayer, J. Cazé, F. Petitpas, E. Daniel and N. Favrie. 'Modelling interactions between waves and diffused interfaces'. In: *Int. J. Numer. Meth. Fl.* 95.2 (2023), pp. 215–241 (cit. on p. 26).

[128] K. Schmidmayer, F. Petitpas, S. Le Martelot and É. Daniel. 'ECOGEN: An open-source tool for multiphase, compressible, multiphysics flows'. In: *Computer Physics Communications* 251 (2020), p. 107093 (cit. on pp. 24–26, 31).

[129] J.-K. Shin, J.-K. Byun and Y.-D. Choi. 'A low Reynolds number dissipation rate equation model using the dissipation rate tensor equation and elliptic-blending equation'. In: *J. Mech. Sci. Technol.* 25.5 (2011), pp. 1361–1371 (cit. on p. 7).

[130] G. Sporschill. 'Improved Reynolds-Stress Modeling for Adverse-Pressure-Gradient Turbulent Boundary Layers in Industrial Aeronautical Flow'. Theses. Université de Pau et des Pays de l'Adour, June 2021. URL: https://theses.hal.science/tel-03440794 (cit. on p. 17).

[131] G. Sporschill, F. Billard, M. Mallet and R. Manceau. 'Reynolds stress RANS models for industrial aeronautical applications'. In: *WCCM-ECCOMAS Congress - 14th World Congress in Computational Mechanics and ECCOMAS Congress*. Paris / Virtual, France, Jan. 2021. URL: https://hal-univ-pau.archives-ouvertes.fr/hal-03154441 (cit. on p. 17).

[132] G. Sporschill, F. Billard, M. Mallet and R. Manceau. 'Turbulence modelling improvements for APG flows on industrial configurations'. In: *55th 3AF International Conference on Applied Aerodynamics (AERO2020+1)*. Poitiers (Virtual), France, Mar. 2021. URL: https://hal.inria.fr/hal-03207390 (cit. on p. 17).

[133] G. Sporschill, F. Billard, M. Mallet, R. Manceau and H. Bézard. 'Assessment of Reynolds-Stress models for aeronautical applications'. In: *International Journal of Heat and Fluid Flow* 96 (2022). DOI: 10.1016/j.ijheatfluidflow.2022.108955. URL: https://inria.hal.science/hal-03597506 (cit. on p. 17).

[134] F. E. Taglialatela and G. De Stefano. 'Numerical study of cavitation effects in shock-induced tandem droplet breakup'. In: *Phys. Fluids* 37.9 (2025) (cit. on p. 26).

[135] J. Weatheritt and R. Sandberg. 'A novel evolutionary algorithm applied to algebraic modifications of the RANS stress–strain relationship'. In: *J. Comp. Phys.* 325 (2016), pp. 22–37 (cit. on p. 26).

[136] G. Yang, H. Iacovides, T. Craft and D. Apsley. 'RANS modelling for temperature variance in conjugate heat transfer'. In: *Proc. 5 th World Congress on Mech., Chem., and Material Eng. (MCM'19), Lisbon, Portugal*. 2019 (cit. on p. 7).