RESEARCH CENTRE

Inria Centre at the University of Bordeaux

IN PARTNERSHIP WITH:

CNRS, Université de Pau et des Pays de l'Adour

2024

ACTIVITY REPORT

Project-Team

CAGIRE

Computational AGility for internal flows sImulations and compaRisons with Experiments

IN COLLABORATION WITH: Laboratoire de mathématiques et de leurs applications (LMAP)

DOMAIN

Applied Mathematics, Computation and Simulation

THEME

Numerical schemes and simulations

Contents

Project-Team CAGIRE **1**

1 Team members, visitors, external collaborators **2**

2 Overall objectives **3**

3 Research program **4**
- 3.1 Turbulence modelling . . . 4
- 3.2 High-order numerical methods and efficient algorithms . . . 4
- 3.3 Compressible and multiphase flows . . . 5
- 3.4 Analysis and simulation of turbulent flows and heat transfer . . . 6

4 Application domains **6**
- 4.1 Aeronautics . . . 6
- 4.2 Energy . . . 7
- 4.3 Automotive propulsion . . . 7
- 4.4 Medical applications . . . 8

5 Social and environmental responsibility **9**

6 Highlights of the year **9**

7 New software, platforms, open data **10**
- 7.1 New software . . . 10
  - 7.1.1 AeroSol . . . 10
  - 7.1.2 DM2 . . . 11
  - 7.1.3 UHAINA . . . 12
  - 7.1.4 ECOGEN . . . 12

8 New results **14**
- 8.1 Turbulence modelling . . . 14
  - 8.1.1 Improvement of the EB-RSM RANS model . . . 14
  - 8.1.2 Extension of RANS turbulence models to mixed and natural convection . . . 14
  - 8.1.3 HTLES: an original hybrid RANS/LES model . . . 15
  - 8.1.4 Towards embedded LES . . . 15
  - 8.1.5 Turbulent premixed combustion in the flamelet regime: developing a closure model for the time filtered reaction rate . . . 16
- 8.2 High-order numerical methods and efficient algorithms . . . 16
  - 8.2.1 Efficient implementation of flux-reconstruction methods for combustion . . . 16
  - 8.2.2 Development of high order numerical schemes for axisymmetric turbulent flows . . 17
  - 8.2.3 Application of high order schemes to coastal flood assessment . . . 17
- 8.3 Compressible and multiphase flows . . . 17
  - 8.3.1 Low-Mach-number schemes . . . 17
  - 8.3.2 Multi-scale multiphase flows . . . 19
  - 8.3.3 Shock-induced cavitation within a droplet . . . 19
  - 8.3.4 Modelling of visco-elastic solids in multiphase flows . . . 20
  - 8.3.5 Impulse-driven release of gas-encapsulated drops . . . 20
- 8.4 Analysis and simulation of turbulent flows and heat transfer . . . 20
  - 8.4.1 Effusion cooling . . . 20
  - 8.4.2 Thermocline energy storage . . . 21

9 Bilateral contracts and grants with industry **21**
- 9.1 Bilateral contracts with industry . . . 21
- 9.2 Bilateral grants with industry . . . 22

10 Partnerships and cooperations **22**
10.1 National initiatives . . . 22
10.1.1 ANR LAGOON . . . 22
10.1.2 ASTURIES . . . 23
10.1.3 ANR MSBUB . . . 24
10.2 Regional initiatives . . . 24
10.2.1 MODEM . . . 24

11 Dissemination **25**
11.1 Promoting scientific activities . . . 25
11.1.1 Scientific events: organisation . . . 25
11.1.2 Journal . . . 25
11.1.3 Invited talks . . . 26
11.1.4 Leadership within the scientific community . . . 26
11.1.5 Scientific expertise . . . 26
11.1.6 Research administration . . . 26
11.2 Teaching - Supervision - Juries . . . 27
11.2.1 Teaching . . . 27
11.2.2 Supervision . . . 27
11.2.3 Juries . . . 28
11.3 Popularization . . . 28
11.3.1 Participation in Live events . . . 28

12 Scientific production **28**
12.1 Major publications . . . 28
12.2 Publications of the year . . . 29
12.3 Cited publications . . . 32

Project-Team CAGIRE

Creation of the Project-Team: 2016 May 01

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. – High performance computing

A6.5.2. – Fluid mechanics

Other research topics and application domains

B2. – Health

B4. – Energy

B4.2. – Nuclear Energy Production

B5.2.1. – Road vehicles

B5.2.3. – Aviation

B5.2.4. – Aerospace

1 Team members, visitors, external collaborators

Research Scientists

- Rémi Manceau [Team leader, CNRS, Senior Researcher, HDR]
- Pascal Bruel [CNRS, Researcher, HDR]
- Vincent Perrier [INRIA, Senior Researcher, from Oct 2024, HDR]
- Vincent Perrier [INRIA, Researcher, until Sep 2024, HDR]
- Kevin Schmidmayer [INRIA, ISFP]

Faculty Member

- Jonathan Jung [LMAP, Associate Professor]

Post-Doctoral Fellows

- Puneeth Bikkanahally Muni Reddy [UPPA, Post-Doctoral Fellow]
- Anthony Bosco [UPPA, from Mar 2024]
- Martin David [UPPA, Post-Doctoral Fellow, until Aug 2024]

PhD Students

- Adedotun Ade-Onojobi [INRIA, from Nov 2024]
- Anthony Bosco [UPPA, until Feb 2024]
- Esteban Coiffier [CEA]
- Alexis Ferre [CEA, until Feb 2024]
- Ibtissem Lannabi [UPPA]
- Jules Mazaleyrat [SAFRAN, CIFRE]
- Corina Sanz Souhait [EDF, CIFRE]
- Romaric Simo Tamou [IFPEN]

Technical Staff

- Abderrahman Benkhalifa [UPPA]
- Luca Cirrottola [INRIA, Engineer]
- Algiane Froehly [INRIA, Engineer, from Nov 2024]

Interns and Apprentices

- Zakaria Karzazi [UPPA, Intern, from May 2024 until Jul 2024]
- Lucas Martin De Fourchambault [CEA, Apprentice, from Oct 2024]
- Marius Smagghe [UPPA, Intern, from May 2024 until Jun 2024]

Administrative Assistant

- Fabienne Cuyollaa [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

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

- High-order numerical methods and efficient algorithms
- 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, [127, 98, 68, 132]), most of them being participants to the European ERCOFTAC SIG-15 group of which we are an active 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, [109, 112]), 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 [81] and PITM [78]); IMF Toulouse in collaboration with the ECUADOR team of the Inria center of Sophia-Antipolis (OES [74, 118]) and CAGIRE (HTLES [107, 67, 88, 73]). 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 [92, 66], 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) [54] and an *adaptive* strategy that autonomously determines the LES zone and refines the mesh based on physical criteria [59], 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 [105] 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 [82]. 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 [123] or are variant of the Roe-Turkel fix [94]. 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 [117].

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 [124]. Or in biomedical applications such as in lithotripsy (treatment for kidney stones) [121] or, recently, histotripsy (non-invasive treatment for cancers) [104] 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 [96], and also mostly in France at EDF R&D by J.M. Hérard or also by Bresch and Hillairet [76]. This low interest in this

[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://exahype.eu/

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 [120].

- 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 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 [75]. 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 [72]. 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 [106, 108, 70, 111], we generate experimental data ourselves when possible and develop collaborations with other research groups when necessary (ONERA, institute Pprime, CEA).

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 [122] 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 is also included in the 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).

- 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 is to study these flows by using advanced RANS models.

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 [113, 63], pursued within the ANR project MONACO_2025 and via the ongoing CIFRE PhD thesis of Corina Sanz Souhait.

- 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).

- For the design of high temperature solar receiver for concentrated solar power plants, flows are characterized by strong variations of the fluid properties, such that, even in the forced convection regime, they significantly deviate from isothermal flows, with a possible tendency to relaminarize, which can significantly reduce heat transfer. A better understanding and modeling of the physical mechanisms observed in turbulent flows with strong temperature gradients are important and was the focus of a recent collaboration with the LaTeP laboratory of UPPA.

- 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 recently defended PhD thesis of Alexis Ferré, co-supervised by R. Manceau and S. Serra (LaTeP). This work is pursued through the co-supervision of 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.

- 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.

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 an energy consumption of the simulation reduced by a factor of about 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

- A. Froehly joined the team to contribute to the development of ECOGEN. Her expertise in software engineering provides essential support, enabling the researchers to concentrate on their core tasks. This focus enhances research productivity and amplifies the impact of the projects undertaken, many of which are detailed in this document.

- K. Schmidmayer participated in February 2024 to an experiment at the European Synchrotron Radiation Facility (ESRF). In the framework of the collaboration with ETH Zürich and Storz Medical, this work implies experiments using a medical extracorporeal shock wave lithotripter and X-rays visualizations. The objective is to discover the contribution of cavitation damage in kidney stone ablation [33, 34]. These experiments will also allow us to better assess the quality of future state-of-the-art simulations. The latter will constitute the first stones for following studies using exploratory technology for kidney stone ablation.

- K. Schmidmayer obtained a young researcher ANR grant to support the modelling and the simulation of bubble dynamics near a kidney stone (project MSBUB). This work represents the computational aspect of a larger project tied to the aforementioned experiments. Collaborating institutions include Aix-Marseille Université, ETH Zürich and ESRF. The project focuses on modeling of biomaterials under a fluid-mechanics formulation including viscoelastic behaviour and realistic equations of state. 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.

- A new collaboration started with the Department for low-temperature systems (DSBT) of CEA around the study of high-Rayleigh number natural convection flows. The work, in the framework of a "high-risk research action" of CEA, is dedicated to understanding and modelling the flow in a vertical boundary layer in a cryogenic facility (liquiq helium, 4K, $Ra=10^{14}$) and is intended to prepare for wider collaborative programs involving several partners (supported by ANR and/or CEA).

- V. Perrier and K. Schmidmayer obtained support from both the Région Nouvelle-Aquitaine and the Université de Pau et des Pays de l'Adour to supervise a PhD student working on the modelling of multi-scale two-phase compressible flows (project MODEM). Indeed, the mastery of compressible multiphase flows, which are complex and multi-scale, is crucial in various fields (aeronautics, automotive, nuclear, etc.). These flows, characterized by small inclusions (bubbles, droplets) and multi-scale interactions, present challenges for modeling and simulation. This project aims to improve models and numerical methods by using a stochastic approach to represent subscale phenomena. It is structured in three stages. 1. *Ab initio* simulation in 1D: Development of a tool where interfaces are resolved, and phases are distributed via a stochastic process. The goal is to link the properties of the mean flow to the prescribed macroscopic quantities. 2. Development of a model and numerical scheme in 1D: Design of a robust scheme to directly simulate the mean flow, comparing the results with those from the first stage. 3. Extension to 2D and 3D: Adaptation of the model and scheme to higher dimensions, validated against benchmark configurations: interface instabilities, wave-bubble cloud interactions, and shock-induced cavitation. This work will enhance the understanding and simulation of complex multi-scale phenomena in multiphase flows.

7 New software, platforms, open data

7.1 New software

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 2024 concern:

1. *Functional tests*: reference metrics were added for several tests, allowing to check the convergence order and the execution time (M. Haefele, V. Perrier). 2. Handling of *spherical manifolds* has been merged into the master branch, allowing to support Uhaina on the Lagoon branch from the AeroSol master branch (M. Haefele, V. Perrier). 3. *Tags* have been defined for relevant historical versions of the code. 4. Complete and time-consistent *Guix channels* have been specified to avoid reproducibility issues (L. Cirrottola). 5. Guix packages and channels for legacy compilation of the Uhaina master branch with a legacy AeroSol branch (L. Cirrottola). 6. New test cases for the *curl-/divergence-preserving schemes* (J. Jung, V. Perrier). 7. Several bug fixes (CI, tests). 8. Wiki improvement.

Contribution statistics: About 300 commits this year, organized in 11 merge requests that were opened and merged into the master branch this year.

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

Contact: Vincent Perrier

Participants: Mario Ricchiuto, Vincent Perrier, Héloïse Beaugendre, Christopher Poette, Marco Lorini, Jonathan Jung, Anthony Bosco, Luca Cirrottola, Romaric Simo Tamou, Ibtissem Lannabi, Matthieu Haefele

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 2024 years:

- Refactoring of *mesh iterators*: dealing with constness, views on specified entity types, generalization of normal (uniform) and non-normal (non-uniform) iterators, handling an entity as a generalized reference. - Refactoring of methods for MPI communication into *generic communication algorithms*. - Refactoring of halo creation methods: no shallow entities, better object-oriented creation. - Passage to the C++14 *standard*. - Development of *generic algorithms for graph permutations*. - Specific *mesh graph composition* for DG or CG schemes. - Towards a consistent handling of 1D meshes: better role distinction between faces and points, remove hard-coded constants, template some methods on the mesh dimension. - Boundary handling: assign boundary faces if not found in the mesh file. - Development of input/output in Vtu format. - Development of input/output in NetCDF. - Full specification of Guix channels for avoiding environment reproducibility issues. - Integration of a two-dimensional finite volume test. - Test references for ParaGMSH objects. - Clean repository paths. - Several bug fixes. - Several small refactorings. - Several small contributions: introducing chronometers, storing the MPI communicator in a variable.

Contributions statistics: about 1800 commits, organized in 32 merge requests that were opened and merged into the master branch during the last year.

Contact: Vincent Perrier

Participants: Abderrahman Ben Khalifa, Matthieu Haefele, Vincent Perrier, Luca Cirrottola

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

Participants: Mario Ricchiuto, Philippe Bonneton, David Lannes, Fabien Marche

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.

Release Contributions: A new release, ECOGEN_v4.0, is available on GitHub. It includes new features and fixes bugs.

Major points: Removed the version numbers of the input file names (break compatibility with previous version). Possibility to use finite pressure relaxation on UEq model for more than two phases. More information in: Schmidmayer et al. DOI link Possibility to initialize an unstructured simulation with the result of a previous simulation performed on a similar mesh and/or a different number of CPUs. This feature is particularly useful to fasten steady state convergence on a fine mesh using coarse mesh results. More information in: Caze‘ et al. (2023). Modelling and simulation of the cavitation phenomenon in space-engine turbopumps. DOI link PUEq phase-change model (handled through PTMu relaxation) does not require mass fraction threshold anymore to trigger mass transfer. Also more information in: Caze‘ et al. (2023). DOI link Add Moving Reference Frame coupling of a rotating region with a static one. Also more information in: Caze‘ et al. (2023). DOI link Renamed boundary condition names (break compatibility with previous version). Immersed boundaries can be added in a Cartesian mesh domain (physicalEntity = -1). Similarly to: Trummler et al. DOI link Minor points: Option to record boundary quantities such as pressure forces and shear stress (useful for aerodynamic coefficient computation). Possibility to display cells' reference length on XML output with unstructured mesh. Add a tutorial on mesh mapping and low-Mach preconditionning options. Add scripts related to droplet shock-induced cavitation. More information in: Biasiori-Poulanges & Schmidmayer (2023) and Schmidmayer & Biasiori-Poulanges (2023). DOI link DOI link Improve variable name style for Gnuplot output. Increase code coverage by nonreg. Updated AMR refinement criteria to match the different modelling. Always a little more source code translation from French to English. A wall is now considered as a symmetry BC if viscosity is ignored. Update of documentation. Fixes: Fix the getTemperature() and copyPhase() methods for multiphase models. Fix restart bug when using alphaNull on PUEq model. Fix bug on simulation progress when using iteration control mode. Fix a MPI data type that caused a crash during MPI_Allreduce on some compilers. Minor fix for Euler-Korteweg model. Correction on UEq BC for volume fraction flux, using sM instead of uStar to respect transport equation. Update non-regression scripts to make them compatible with OS X. Correction for phase change with a second-order method.

News of the Year: A new release, ECOGEN_v4.0, is available on GitHub. It includes new features and fixes bugs.

Major points: - Removed the version numbers of the input file names (break compatibility with previous version). - Possibility to use finite pressure relaxation on UEq model for more than two phases. - Possibility to initialize an unstructured simulation with the result of a previous simulation performed on a similar mesh and/or a different number of CPUs. This feature is particularly useful to fasten steady state convergence on a fine mesh using coarse mesh results. - PUEq phase-change model (handled through PTMu relaxation) does not require mass fraction threshold anymore to trigger mass transfer. - Add Moving Reference Frame coupling of a rotating region with a static one. - Renamed boundary condition names (break compatibility with previous version). - Immersed boundaries can be added in a Cartesian mesh domain (physicalEntity = -1).

Minor points: - Option to record boundary quantities such as pressure forces and shear stress (useful for aerodynamic coefficient computation). - Possibility to display cells' reference length on XML output with unstructured mesh. - Add a tutorial on mesh mapping and low-Mach preconditionning options. - Add scripts related to droplet shock-induced cavitation. - Improve variable name style for Gnuplot output. - Increase code coverage by nonreg. - Updated AMR refinement criteria to match the different modelling. - Always a little more source code translation from French to English. - A wall is now considered as a symmetry BC if viscosity is ignored. - Update of documentation.

Fixes: - Fix the getTemperature() and copyPhase() methods for multiphase models. - Fix restart bug when using alphaNull on PUEq model. - Fix bug on simulation progress when using iteration control mode. - Fix a MPI data type that caused a crash during MPI_Allreduce on some compilers. - Minor fix for Euler-Korteweg model. - Correction on UEq BC for volume fraction flux, using sM instead of uStar to respect transport equation. - Update non-regression scripts to make them compatible with OS X. - Correction for phase change with a second-order method.

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

Publications: hal-01781830, hal-03649359

Contact: Kevin Schmidmayer

Participants: Kevin Schmidmayer, Fabien Petitpas, Joris Cazé, Sébastien Le Martelot, Eric Daniel, Benedikt Dorschner, Solene Schropff, Adedotun Ade-Onojobi, Fatima Gadiri

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 [109, 112]. Although this approach, has been successfully applied to various configurations (for instance [71]), 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 [128, 79, 129, 130, 131]. 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.

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 [110, 111, 58], the interaction mechanisms between dynamic and thermal fluctuations are complex and very anisotropic due to buoyancy effects [18], 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 [83]. 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 [99, 101, 100], 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 [102].

A new 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.

8.1.3 HTLES: an original hybrid RANS/LES model

Participants: Rémi Manceau, Puneeth Bikkanahally, Martin David.

External collaborators: 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 [86] 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 [88, 87, 57, 86]. 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 [73, 56].

8.1.4 Towards embedded LES

Participants: Rémi Manceau, Pascal Bruel, Puneeth Bikkanahally, Martin David.

External collaborators: 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 [116, 54]. 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. 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 [13].

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).

The long-term objective pursued here is to extend the approach of temporal large-eddy simulation to turbulent reactive flows. As a first step towards this goal, 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 [14]. After considering in [60] 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 and disseminated in [31]. Applications to combustion were presented in [36], and the use of combined AMR and high order was discussed in [35].

8.2.2 Development of high order numerical schemes for axisymmetric turbulent flows

Participants: Anthony Bosco, Jonathan Jung, Vincent Perrier.

External collaborators: Marina Olazabal-Loumé (CEA), Céline Baranger (CEA).

Part of the Asturies project aims at deriving high order numerical schemes for turbulent compressible flows in axisymmetric geometry. This year, the work was focused on the three following points:

1. The derivation of conservative formulation for general advection-diffusion conservation laws in axisymmetric coordinates. This led to a new high order discontinuous Galerkin method for axisymmetric coordinates, which is optimal order and in which the source term are unambiguously discretized and defined. This work was disseminated in [12] and in an extended version [65].

2. The wall boundary conditions for a large family of turbulent models including the $k-\varepsilon$ models can be seen as a double boundary condition on one of the variables (for example homogeneous Neumann *and* homogeneous Dirichlet on k) and no boundary condition on ε. A simplified version of this model with these boundary conditions can be seen as a mixed formulation for a bi-Laplacian equation, on which these types of boundary conditions are natural. Inspired by theses ideas, a new discontinuous Galerkin method was developed for this type of boundary condition on the simplified linear model, and was disseminated in [64].

8.2.3 Application of high order schemes to coastal flood assessment

Participants: Vincent Perrier.

In line with our long term collaboration with BRGM wthin the development of the plugin Uhaina of our library AeroSol, an article was published in which high order methods for shallow water equations are assessed with high order discontinuous Galerkin methods, in operational conditions of coastal flood assessment [15].

8.3 Compressible and multiphase flows

8.3.1 Low-Mach-number schemes

Participants: Jonathan Jung, Vincent Perrier, Ibtissem Lannabi, Esteban Coiffier.

External collaborators: Thomas Galié (CEA), Michael Ndjinga (CEA).

Collocated finite volume schemes (Godunov, Roe, Osher, HLL, HLLC, Rusanov, etc.) are not accurate at low Mach number unless a simplicial mesh and a numerical flux conserving exactly contacts (e.g., Godunov, Roe, Osher, HLLC) are used [93]. This result has been extended to high order for the discontinuous Galerkin method on simplicial mesh in [19].

On a quadrangular mesh, many fixes are available in the literature to obtain an accurate scheme at low Mach numbers. These fixes modify the numerical flux and can be regrouped into two large families: fixes consisting of centering the pressure gradient [123] and those which are variants of the Roe-Turkel fixes [94]. Unfortunately, theses flux fixes are not accurate for high order acoustics computation. In [77] an other flux fix was proposed, allowing to tackle this problem for isentropic Euler system. This fix was extended to full Euler system in [16].

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 [62]. 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 the subject of a talk in a conference [43].

The previous study on the oscillating modes revealed that the fix developed in [16], 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 [103], 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 [20], we propose to enrich the approximation space for the velocity, then the Hodge-Helmholtz context developed for triangular meshes in [84] 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.

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 at the 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 [19]. We were also able to prove that the low order approximation space proposed on quadrangular meshes in [20] can be easily extended to higher order. The proof of these properties and the derivation of the quadrangular approximation space were published in [21].

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 is still submitted [40].

We also looked at numerical schemes based on staggered discretizations (velocity at faces and pressure at cells), initially developed for incompressible fluid mechanics [97], 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. This work was the subject of a talk in a conference [30].

We also had the opportunity to disseminate our work on low Mach number flows and div/curl preserving schemes for hyperbolic systems at different conferences [27, 28, 29, 32, 41, 42], included invited ones [25].

8.3.2 Multi-scale multiphase flows

Participants: Vincent Perrier, Kevin Schmidmayer.

As far as multiphase models are concerned, based on the ideas of [85], we have revisited the derivation of Baer-and-Nunziato models [69]. 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 [80]. In [120], 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, 91], which does not necessarily ensure the same phase entropy dissipation properties. This work was disseminated this year in the conference [24]. 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. Anju Albi Lilly was hired on this project and starts her Phd contract in January 2025.

8.3.3 Shock-induced cavitation within a droplet

Participants: Kevin Schmidmayer.

External collaborators: L. Biasiori-Poulanges (ETH Zürich, Switzerland).

In [53] and [55], 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.

8.3.4 Modelling of visco-elastic solids in multiphase flows

Participants: Kevin Schmidmayer.

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

As a work in progress, an extension of the model of diffuse solid–fluid interfaces [119, 89] is proposed to deal with arbitrary complex materials such as porous materials in presence of plasticity and damage. 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 [95]) and damage (Mazars [115]) models.

8.3.5 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 [11], 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.

8.4 Analysis and simulation of turbulent flows and heat transfer

8.4.1 Effusion cooling

Participants: Rémi Manceau, Pascal Bruel, Martin David.

External collaborators: 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 [122, 114]. 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, Alexis Ferré.

External collaborators: 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 [90, 61].

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 A. Ferré".
- CEA: “Collaboration contract for the PhD thesis of E. Coiffier".
- IFPEN: “Collaboration contract for the PhD thesis of Romaric Simo-Tamou".
- CEA DAM: “Prestation contract for the introduction of two-velocity models into ECOGEN".
- CEA DAM: “Collaboration contract for the Master 2 apprenticeship of L. Martin de Fourchambault, on the implementation of a multi-velocity model into ECOGEN".
- 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: “CFD and experimental study of a thermocline-type thermal storage for an optimized design and data entry of component scale models in the framework of a multi-scale approach”, PhD student Alexis Ferré.
- 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 National initiatives

10.1.1 ANR LAGOON

Participants: Vincent Perrier, Abderrahman Benkhalifa.

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.

10.1.2 ASTURIES

Participants: Rémi Manceau, Vincent Perrier, Jonathan Jung, Pascal Bruel, Anthony Bosco, Romaric Simo Tamou, Martin David.

Call: ISite E2S UPPA "Exploring new topics and facing new scientific challenges for Energy and Environment Solutions"

Dates: 2020-2024

Partners: CEA CESTA ; IFPEN

In the context of internal turbulent flows, relevant to aeronautic and the automotive propulsion and energy production sectors, ASTURIES aims at developing an innovative CFD methodology. The next generation of industrial CFD tools will be based on the only approach compatible with admissible CPU costs in a foreseeable future, hybrid RANS/LES. However, state-of-the-art hybrid RANS/LES methods suffer from a severe limitation: their results are strongly user-dependant, since the local level of description of the turbulent flow is determined by the mesh designed by the user.

In order to lift this technological barrier, an *agile* methodology will be developed: the scale of description of turbulence will be locally and automatically adapted during the computation based on local physical criteria independent of the grid step, and the mesh will be automatically refined in accordance. Such an innovative approach requires the use of advanced near-wall turbulence closures, as well as high-order numerical methods for complex geometries, since low-dissipative discretization is necessary in LES regions. Morevover, the identification of relevant physical RANS-to-LES switchover criteria and the refined validation of the method will strongly benefit from dedicated experiments.

The objectives of the project thus consisted in:

- Proposing a robust and efficient implementation of elliptic relaxation/blending turbulence models in the context of high-order Discontinuous Galerkine methods.
- Develop local physical criteria in order to get rid of the (explicit or implicit) dependence on the grid step of the transition from RANS to LES.
- Develop an automatic remeshing strategy which ensures consistency with the self-adaptation of the model.
- Validate the global methodology based on the 3 preceding points for configurations representative of industrial internal turbulent flows.

This project ended in November 2024 and led to numerous results and publications described in section 8.

10.1.3 ANR MSBUB

Participants: Kevin Schmidmayer, Adedotun Ade-Onojobi.

Call: ANR young researcher project within the section CE51 – Sciences de l'ingénierie et des procédés.
Dates: 2024-2029
Partners: Aix-Marseille Université ; ETH Zürich ; European Synchrotron Radiation Facility
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. 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.

10.2 Regional initiatives

10.2.1 MODEM

Participants: Vincent Perrier, Kevin Schmidmayer.

Call: AAP Région Nouvelle-Aquitaine.
Dates: 2024-2027
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, Martin David, Mahitosh Mehta, Ibtissem Lannabi, Anthony Bosco, Puneeth Bikkanahally Muni Reddy.

11.1 Promoting scientific activities

11.1.1 Scientific events: organisation

Member of the organizing committees

- Member of the scientific committee of the International Symposium on Turbulence, Heat and Mass Transfer since 2006 [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 of Computational Physics [Vincent Perrier (1)]
- Computer Methods in Applied Mechanics and Engineering [Vincent Perrier (1)]
- Mathematics of Computation [Vincent Perrier (1)]
- Computer & Fluids [K. Schmidmayer (1), R. Manceau (2)]
- Flow Turb. Combust. [R. Manceau (3)]
- Phys. Fluids [R. Manceau (2)]
- Nuclear Eng. Design [R. Manceau (1)]
- Int. J. Heat Fluid Flow [R. Manceau (1)]
- Int. J. CFD [R. Manceau (1)]
- J. Fluid Mech. [R. Manceau (1)]
- SMAI Journal of Computational Mathematics [J. Jung (1)]

11.1.3 Invited talks

[23] R. Manceau et al. 'Generation of turbulent structures for CELES (continuous embedded LES)'. in: A turbulence day in celebration of the career of Jean-Paul Bonnet. Poitiers, France, 21st June 2024. URL: https://inria.hal.science/hal-04620701

[22] R. Manceau. 'Modélisation hybride RANS/LES : du formalisme aux applications'. In: Atelier sur la représentation des fines échelles océaniques dans les simulations numériques. Plouzané, France, 9th Oct. 2024. URL: https://inria.hal.science/hal-04733173

[24] 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

[25] V. Perrier. 'How to preserve an implicit divergence or curl constraint in a hyperbolic system with the discontinuous Galerkin method'. In: ProHyp 2024 - 3rd International Workshop on Perspectives on Multiphase Fluid Dynamics, Continuum Mechanics and Hyperbolic Balance Laws. Trento, Italy, 22nd Apr. 2024. URL: https://inria.hal.science/hal-04883908

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.
- Member of the Software and Source Codes College of the Committee for Open Science from the Ministère de l'enseignement supérieur et de la recherche [Kevin Schmidmayer].
- 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.
- Rémi Manceau coordinates the 4-year E2S-UPPA project ASTURIES, which involves CEA and IFPEN.

11.1.5 Scientific expertise

Vincent Perrier participated to the evaluation of the ONERA "Projet de Recherche Fédérateur" JEROBOAM on industrialization of high order methods.

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]
- Member of the scienfic board of INRIA (Until October 2024) [Vincent Perrier]
- Member of the scienfic board of INSMI (CNRS Mathématiques), from January to October 2024 [Vincent Perrier]

11.2 Teaching - Supervision - Juries

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 [51].
- E3 [R. Manceau]: Advanced physics–Turbulence modelling for CFD, 16h/year, ENSGTI, France [52].
- M1 [K. Schmidmayer]: Introduction to Python, 16h/year, Master MSID, Pau, France.
- L2 [K. Schmidmayer]: Documentary-Communication Project, 4h/year, Master CMI, Pau, France.
- E3 (M. David): Modelling and simulation in fluid mechanics (lectures: 4h/year; classes: 16h/year). ENSGTI.
- M2 (M. David): Modelling of energy systems CFD for industry using fluent (lectures: 10.5h/year; classes: 30h/year). UPPA, Master SIMOS.
- L1 (I. Lannabi): Descriptive statistics (22.5h/year, labs), Mathematics & MIASHS. University of Pau (UPPA).has been a member of
- L1 (I. Lannabi) Mathematical Algorithms 1 and Python (49.5h/year, labs) Mathematics, University of Pau (UPPA).
- L2 (A. Bosco) Scientific computing (18h/year labs), computer science, UPPA.
- L1 MIASHS/L2 Natural sciences (A. Bosco) Descriptive statistics (22.5h/year labs), UPPA.

11.2.2 Supervision

- Defended in Sept. 2024: Alexis Ferré, "CFD and experimental study of a thermocline-type thermal storage for an optimized design and data entry of component scale models in the framework of a multi-scale approach", CEA LITEN, Rémi Manceau.
- Defended in Nov. 2024: Ibtissem Lannabi , "Discontinuous Galerkin methods for low Mach flows in fluid mechanics", EDENE project (H2020 Marie-Sklodowska-Curie COFUND), Jonathan Jung and Vincent Perrier.
- Defended in Dec. 2024: Anthony Bosco, "Development of Fast, Robust and Accurate numerical methods for turbulence models on Complex Meshes" CEA/E2S-UPPA, E2S-UPPA Asturies project, Vincent Perrier and Jonathan Jung.

- PhD in progress: 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.
- PhD in progress: 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: Adedotun Ade-Onojobi, “Numerical modelling of cavitation bubbles interacting with biomaterials”, Inria, Vincent Perrier & Kevin Schmidmayer.

11.2.3 Juries

- Reviewer of the PhD thesis of L. Villanueva, ENSMA Poitiers [R. Manceau]
- Jury member of the PhD thesis of A. Stoffels, univ. Poitiers [R. Manceau]
- Jury member of the PhD thesis of A. Loison, École Polytechnique [V. Perrier]
- Jury member of the PhD thesis of V. Courtin, ONERA Meudon / Université Paris-Saclay [V. Perrier]

11.3 Popularization

11.3.1 Participation in Live events

[49] K. Schmidmayer. ‘Mécanique des fluides numérique’. In: Rencontres scientifiques, Lycée Louis Barthou. Pau, France, 11th Apr. 2024. URL: https://hal.science/hal-04543297

[47] R. Manceau and K. Schmidmayer. ‘Chaud devant ! Les maths et la physique en action dans les réseaux de chaleur urbains’. In: Fête de la science, Lycée Jules Supervielle. Oloron-Sainte-Marie, France, 7th Oct. 2024. URL: https://hal.science/hal-04726586

[48] K. Schmidmayer. ‘Lithotripsie’. In: Nuit européenne des chercheurs. Pau, France, 27th Sept. 2024. URL: https://hal.science/hal-04718662

[46] R. Manceau. ‘Le stockage d'énergie dans les réseaux de chaleur urbains : comment ne pas réinventer l'eau tiède ?’ In: Nuit européenne des chercheurs. Pau, France, 27th Sept. 2024. URL: https://inria.hal.science/hal-04733188

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] G. Bokman, L. Biasiori-Poulanges, B. Lukić, K. Schmidmayer, C. Bourquard, E. Baumann, A. Rack, B. Olson and O. Supponen. 'Impulse-driven release of gas-encapsulated drops'. In: *Journal of Fluid Mechanics* 1001 (19th Dec. 2024), A51. DOI: 10.1017/jfm.2024.1124. URL: https://hal.science/hal-04848523 (cit. on p. 20).

[12] A. Bosco and V. Perrier. 'Discontinuous Galerkin methods for axisymmetric flows'. In: *Computers and Fluids* 270 (15th Feb. 2024), p. 106139. DOI: 10.1016/j.compfluid.2023.106139. URL: https://inria.hal.science/hal-04356856 (cit. on p. 17).

[13] 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. 16).

[14] 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. 16).

[15] A. G. Filippini, L. Arpaia, V. Perrier, R. Pedreros, P. Bonneton, D. Lannes, F. Marche, S. de Brye, S. Delmas, S. Lecacheux, F. Boulahya and M. Ricchiuto. 'An operational discontinuous Galerkin shallow water model for coastal flood assessment'. In: *Ocean Modelling* 192 (Dec. 2024), p. 102447. DOI: 10.1016/j.ocemod.2024.102447. URL: https://brgm.hal.science/hal-04761792 (cit. on p. 17).

[16] 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. 18).

[17] M. Grioni, S. Elaskar, P. Bruel and A. Mirasso. 'Numerical Simulation of Turbulent Flows using the SST-SAS Model'. In: *WSEAS Transactions on Fluid Mechanics* 19 (29th Jan. 2024), pp. 24–39. DOI: 10.37394/232013.2024.19.3. URL: https://inria.hal.science/hal-04428219.

[18] 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. 14).

[19] 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. 18).

[20] 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 p. 18).

[21] 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* (2025). URL: https://inria.hal.science/hal-04564069. In press (cit. on p. 18).

Invited conferences

[22] R. Manceau. 'Modélisation hybride RANS/LES : du formalisme aux applications'. In: Atelier sur la représentation des fines échelles océaniques dans les simulations numériques. Plouzané, France, 9th Oct. 2024. URL: https://inria.hal.science/hal-04733173 (cit. on p. 26).

[23] R. Manceau, M. Mehta, M. David and P. Bikkanahally. 'Generation of turbulent structures for CELES (continuous embedded LES)'. In: A turbulence day in celebration of the career of Jean-Paul Bonnet. Poitiers, France, 21st June 2024. URL: https://inria.hal.science/hal-04620701 (cit. on p. 26).

[24] 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 pp. 19, 26).

[25] V. Perrier. 'How to preserve an implicit divergence or curl constraint in a hyperbolic system with the discontinuous Galerkin method'. In: ProHyp 2024 - 3rd International Workshop on Perspectives on Multiphase Fluid Dynamics, Continuum Mechanics and Hyperbolic Balance Laws. Trento, Italy, 22nd Apr. 2024. URL: https://inria.hal.science/hal-04883908 (cit. on pp. 19, 26).

International peer-reviewed conferences

[26] S. Elaskar, P. Bruel and M. Frias. 'Markov Process Applied to the Study of Turbulent Premixed Flames'. In: *2024 IEEE Biennial Congress of Argentina (ARGENCON)*. ARGENCON 2024 - IEEE Biennial Congress of Argentina. San Nicolás de los Arroyos, Argentina: IEEE, 2024, pp. 1–7. DOI: 10.1109/ARGENCON62399.2024.10735851. URL: https://inria.hal.science/hal-04778417.

[27] J. Jung, I. Lannabi and V. Perrier. 'Unraveling the origin and identification of oscillatory modes in numerical low Mach number flow solutions'. In: ECCOMAS 2024 - The 9th European Congress on Computational Methods in Applied Sciences and Engineering. Lisbon, Portugal, 3rd June 2024. URL: https://inria.hal.science/hal-04884331 (cit. on p. 19).

[28] J. Jung and V. Perrier. ‘A finite volume scheme accurate at low Mach number on quadrangular mesh by space velocity enrichment’. In: ECCOMAS 2024 - 9th European Congress on Computational Methods in Applied Sciences and Engineering. Lisboa, Portugal, 3rd June 2024. URL: https://hal.science/hal-04836529 (cit. on p. 19).

[29] J. Jung and V. Perrier. ‘Discontinuous Galerkin methods for hyperbolic system that preserve exactly a curl or a divergence constraint’. In: ECCOMAS 2024 - The 9th European Congress on Computational Methods in Applied Sciences and Engineering. Lisbon, Portugal, 3rd June 2024. URL: https://inria.hal.science/hal-04884111 (cit. on p. 19).

[30] V. Perrier, E. Coiffier, J. Jung and M. Ndjinga. ‘Schéma numérique décalé pour la simulation d’écoulements à bas nombre de Mach’. In: CANUM 2024 - 46ème Congrès national d’Analyse Numérique. Le Bois-Plage-En-Ré, France, 2024. URL: https://cea.hal.science/cea-04841863 (cit. on p. 19).

[31] R. Simo Tamou, V. Perrier, Q. H. Tran, J. Bohbot and J. Coatleven. ‘Spectral analysis for high-order flux-reconstruction methods’. In: *9th European Congress on Computational Methods in Applied Sciences and Engineering*. ECCOMAS 2024 - 9th European Congress on Computational Methods in Applied Sciences and Engineering. Lisbon (Portugal), Portugal, 3rd June 2024. DOI: 10.23967/eccomas.2024.032. URL: https://hal.science/hal-04900924 (cit. on p. 17).

Conferences without proceedings

[32] V. Perrier. ‘How to preserve a divergence or a curl constraint in a hyperbolic system with the discontinuous Galerkin method’. In: High-Order NOnlinear numerical Methods for evolutionary PDEs: theory and applications (HONOM 2024). Chania (Crète), Greece, 8th Sept. 2024. URL: https://inria.hal.science/hal-04884418 (cit. on p. 19).

[33] A. Sieber, G. Shakya, G. T. Bokman, M. Belau, M. Schlötter, A. Kühl, B. Lukić, K. Schmidmayer and O. Supponen. ‘Discovering the contribution of cavitation damage in kidney stone ablation through X-ray high-speed imaging and microtomography’. In: 12th International Cavitation Symposium, CAV2024. Chania Crete, Greece, 2nd June 2024. URL: https://hal.science/hal-04700333 (cit. on p. 9).

[34] A. Sieber, G. Shakya, B. Lukić, G. T. Bokman, M. Belau, M. Schlötter, A. Kühl, K. Schmidmayer and O. Supponen. ‘Investigating cavitation activity and damage evolution in kidney stone fragmentation through X-ray high-speed imaging and microtomography’. In: 1st European Fluid Dynamics Conference, EFDC1. Aachen, Germany, 16th Sept. 2024. URL: https://hal.science/hal-04700345 (cit. on p. 9).

[35] R. Simo-Tamou, J. Bohbot, J. Coatléven, V. Perrier and Q. H. Tran. ‘High-order Flux Reconstruction Scheme development coupled with AMR in CONVERGE’. In: CONVERGE CFD Conference 2024. Madison (US-WI), United States, Oct. 2024. URL: https://hal.science/hal-04900941 (cit. on p. 17).

[36] R. Simo-Tamou, J. Bohbot, J. Coatléven, V. Perrier and Q. H. Tran. ‘High-order Flux Reconstruction schemes for combustion’. In: Journée de la Combustion Turbulente. Rouen, France, Mar. 2024. URL: https://hal.science/hal-04901139 (cit. on p. 17).

Doctoral dissertations and habilitation theses

[37] A. Bosco. ‘Development of fast, accurate and robust numerical methods for turbulent flows on unstructured meshes’. Université de Pau et des Pays de l’Adour, 5th Dec. 2024. URL: https://hal.science/tel-04903738.

[38] A. Ferré. ‘CFD and experimental study of a thermocline thermal storage’. Université de Pau et des Pays de l’Adour, 26th Sept. 2024. URL: https://theses.hal.science/tel-04820213.

[39] I. Lannabi. ‘Analysis of the problem of parasitic oscillations in Finite Volume methods for low Mach number flows in fluid mechanics’. Université de Pau et des Pays de l’Adour, 25th Nov. 2024. URL: https://hal.science/tel-04903787.

Reports & preprints

[40] V. Perrier. *Development of discontinuous Galerkin methods for hyperbolic systems that preserve a curl or a divergence constraint*. 6th May 2024. URL: https://inria.hal.science/hal-04564886 (cit. on p. 18).

Other scientific publications

[41] J. Jung and V. Perrier. 'A numerical scheme accurate at low Mach number based on space velocity enrichment'. In: 36ème séminaire CEA/SMAI-GAMNI de mécanique des fluides numérique. Paris, France, 29th Jan. 2024. URL: https://hal.science/hal-04503360 (cit. on p. 19).

[42] J. Jung and V. Perrier. 'Conservation of first order involutions in hyperbolic systems with the discontinuous Galerkin method'. In: Kickoff meeting of the ERC Synergy Grant "NEMESIS: NEw generation MEthods for numerical SImulationS. Montpellier, France, 19th June 2024. URL: https://inria.hal.science/hal-04884390 (cit. on p. 19).

[43] I. Lannabi, J. Jung and V. Perrier. 'Analyzing the issue of oscillating modes at low Mach numbers'. In: CEA-SMAI/GAMNI : la mécanique des fluides numériques. Paris, France, 29th Jan. 2024. URL: https://inria.hal.science/hal-04885344 (cit. on p. 18).

[44] R. Manceau. *The active approach of hybrid RANS/LES modeling for continuous embedded LES*. 21st Nov. 2024. URL: https://inria.hal.science/hal-04803416.

Scientific popularization

[45] I. Lannabi. *Filles, maths et informatique : une équation lumineuse*. 10th Dec. 2024. URL: https://inria.hal.science/hal-04885366.

[46] R. Manceau. 'Le stockage d'énergie dans les réseaux de chaleur urbains : comment ne pas réinventer l'eau tiède ?' In: Nuit européenne des chercheurs. Pau, France, 27th Sept. 2024. URL: https://inria.hal.science/hal-04733188 (cit. on p. 28).

[47] R. Manceau and K. Schmidmayer. 'Chaud devant ! Les maths et la physique en action dans les réseaux de chaleur urbains'. In: Fête de la science, Lycée Jules Supervielle. Oloron-Sainte-Marie, France, 7th Oct. 2024. URL: https://hal.science/hal-04726586 (cit. on p. 28).

[48] K. Schmidmayer. 'Lithotripsie'. In: Nuit européenne des chercheurs. Pau, France, 27th Sept. 2024. URL: https://hal.science/hal-04718662 (cit. on p. 28).

[49] K. Schmidmayer. 'Mécanique des fluides numérique'. In: Rencontres scientifiques, Lycée Louis Barthou. Pau, France, 11th Apr. 2024. URL: https://hal.science/hal-04543297 (cit. on p. 28).

Educational activities

[50] R. Manceau. 'Codes de calcul industriels pour la simulation des écoulements turbulents'. Master. France, 5th Feb. 2024. URL: https://inria.hal.science/hal-03207435.

[51] R. Manceau. 'Industrial codes for CFD'. Master. France, 5th Feb. 2024. URL: https://inria.hal.science/hal-03207431 (cit. on p. 27).

[52] R. Manceau. 'Modelisation de la turbulence pour la CFD'. Master. France, 15th Oct. 2024. URL: https://inria.hal.science/hal-03207437 (cit. on p. 27).

12.3 Cited publications

[53] 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 p. 19).

[54] 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. 4, 16).

[55] 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. 19).

[56] 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. 15).

[57] 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. 15).

[58] 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. 14).

[59] 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. 4).

[60] 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. 16).

[61] 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. 21).

[62] 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. 18).

[63] 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. 7).

[64] A. Bosco, V. Perrier and J. Jung. 'Variational Formulation of Wall Boundary Conditions of RANS Models in a Discontinuous Galerkin Framework'. In: CFC 2023 - 22nd Computational Fluids Conference. Cannes, France, 25th Apr. 2023. URL: https://hal.science/hal-04095353 (cit. on p. 17).

[65] A. Bosco and V. Perrier. *Discontinuous Galerkin methods for axisymmetric flows: extended version*. RR-9524. INRIA, 10th Oct. 2023, p. 49. URL: https://inria.hal.science/hal-04236220 (cit. on p. 17).

[66] 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. 4).

[67] 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. 4).

[68] 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. 4).

[69] 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. 19).

[70] 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. 6).

[71] 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. 14).

[72] 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. 6).

[73] 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. 4, 15).

[74] 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. 4).

[75] C. E. Brennen. 'Cavitation in medicine'. In: *Interface Focus* 5.5 (2015), p. 20150022 (cit. on p. 6).

[76] 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. 5).

[77] 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 (Feb. 2019), pp. 723–759. DOI: 10.1016/j.jcp.2018.11.020. URL: https://hal.inria.fr/hal-01953424 (cit. on p. 18).

[78] 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. 4).

[79] 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. 14).

[80] 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. 19).

[81] 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. 4).

[82] 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. 5).

[83] 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. 15).

[84] 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. 18).

[85] D. A. Drew and S. L. Passman. *Theory of multicomponent fluids*. Vol. 135. Springer Science & Business Media, 2006 (cit. on p. 19).

[86] 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. 15).

[87] 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. 15).

[88] 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. 4, 15).

[89] 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. 20).

[90] 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. 21).

[91] 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. 19).

[92] M. Germano. 'Properties of the hybrid RANS/LES filter'. In: *Theor. Comput. Fluid Dyn.* 17.4 (2004), pp. 225–231 (cit. on p. 4).

[93] H. Guillard. 'On the behavior of upwind schemes in the low Mach number limit. IV: P0 approximation on triangular and tetrahedral cells'. In: *Computers & fluids* 38.10 (2009), pp. 1969–1972 (cit. on p. 18).

[94] 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 pp. 5, 18).

[95] 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. 20).

[96] 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. 5).

[97] 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. 18).

[98] 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. 4).

[99] 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. 15).

[100] 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. 15).

[101] 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. 15).

[102] 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. 15).

[103] 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. 18).

[104] 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. 5).

[105] 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. 5).

[106] 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. 6).

[107] 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. 4).

[108] 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. 6).

[109] 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. 4, 14).

[110] 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. 14).

[111] 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. 6, 14).

[112] 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. 4, 14).

[113] 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. 7).

[114] 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. 21).

[115] 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. 20).

[116] 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. 16).

[117] 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. 5).

[118] 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. 4).

[119] 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. 20).

[120] 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. 6, 19).

[121] 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. 5).

[122] 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. 6, 21).

[123] 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 pp. 5, 18).

[124] 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. 5).

[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. 19).

[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. 6).

[127] 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. 4).

[128] 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. 14).

[129] 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. 14).

[130] 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. 14).

[131] 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. 14).

[132] 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. 4).