RESEARCH CENTRE

Inria Center
at the University of Bordeaux

IN PARTNERSHIP WITH:

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

2022
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

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, Researcher, HDR]
- Kevin Schmidmayer [INRIA, Researcher, from Apr 2022]

Faculty Member

- Jonathan Jung [UPPA, Associate Professor]

Post-Doctoral Fellows

- Martin David [UPPA, from Oct 2022]
- Sangeeth Simon [INRIA, until Sep 2022]

PhD Students

- Puneeth Bikkanahally Muni Reddy [UPPA]
- Anthony Bosco [UPPA]
- Esteban Coiffier [CEA]
- Alexis Ferre [CEA]
- Ibtissem Lannabi [UPPA]
- Mahitosh Mehta [UPPA]
- Romaric Simo Tamou [IFPEN]

Administrative Assistant

- Sylvie Embolla [INRIA]

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 the field of transportation, energy production and medicine. In order to progress in this direction, many building blocks have to be assembled, which motivates a variety of research topics described in the following sections and divided into four main research axes. The topics addressed, ranging from advanced physical modelling to high-order numerical discretization, require the multi-disciplinary skills that constitute the CAGIRE project-team:

- Turbulence modelling
- High-order numerical methods and efficient algorithms
- 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 near-wall 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, [113, 84, 52, 117]), most of them being participants to the European ERCOFTAC SIG-15 group of which we are an active member. In France, we collaborate with most of the teams, mainly in the industry (EDF, Dassault, PSA) or applied

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

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, [94, 97]), 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 [67] and PITM [64]); IMF Toulouse in collaboration with the ECUADOR team of the Inria center of Sophia-Antipolis (OES [60, 103]) and CAGIRE (HTLES [92, 50] [12, 58]). The originality of our work lies in the concern to provide, through temporal filtering, a formally consistent link 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 [78, 48], one of only two methods capable of providing such a consistent framework.

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 [90] 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 type of numerical problems are known: if convective time scales are considered, semi-implicit time integration is often preferred to explicit ones, because the acoustic

[3]https://cordis.europa.eu/project/id/635962
[4]https://www.gas-turbine-lab.mit.edu/
[5]http://web.stanford.edu/group/ihmegroup/cgi-bin/MatthiasIhme/
[6]https://exahype.eu/

CFL is very restrictive compared with the convective one in the low Mach number limit [68]. 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 [108] or are variant of the Roe-Turkel fix [80]. 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 [102].

Understanding and controlling complex and physically rich flows, such as unsteady multiphase compressible flows, are 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 [109]. Or in biomedical applications such as in lithotripsy (treatment for kidney stones) [106] or, recently, histotripsy (non-invasive treatment for cancers) [89] 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 [83], and also mostly in France at EDF R&D by J.M. Hérard or also by Bresch and Hillairet [62]. This low interest in this type of challenging modeling and mathematical analysis was noticed in the review paper [112] 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 [105].

- 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 [61]. 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 [57]. 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 [91, 93, 55, 96], 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 (which is in itself an originality within Inria), 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 [107] 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 are the three axes we have been developing 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.
- The flow around airfoils: the modelling of the turbulent boundary layer has been for almost a century a key issue in the aeronautics industry. However, even the more advanced RANS models face difficulties in predicting the influence of pressure gradients on the development of the boundary layer. A main issue is the reliability of the modelling hypotheses, which is crucial for less conservative design. One of the technological barriers is the prediction of the flow in regimes close to the edge of the flight domain (stall, buffeting, unsteady loads) when the boundary layer is slowed down by an adverse pressure gradient. This was the subject of the CIFRE PhD thesis of Gustave Sporschill in collaboration with Dassault Aviation.
- 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 a starting collaboration with SAFRAN HE.
- 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. Within the SEIGLE project, the team is involved in the simulation of the interaction of a droplet (representing the ablated body) and a hypersonic flow. In the Asturies project, the aim is to study the improvement on the shock/turbulence interaction by using advanced RANS models (second-moment closure).

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. The PhD thesis (CIFRE EDF) of G. Mangeon, was dedicated to the development of relevant RANS models for these industrial applications [100]. The collaboration with EDF is pursued within the ANR project MONACO_2025 and via a new CIFRE PhD thesis under discussion.

- 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 is pursued within the ANR project MONACO_2025.

- 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 is the focus of the PhD thesis of Alexis Ferré, co-supervised by R. Manceau and S. Serra (LaTeP) started in November 2020.

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. This technical barrier must be lifted, since the ambition of the PSA group is to reach a *full digital design of their vehicles in the 2025 horizon*, 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 described in section 10.3.1, in the framework of which S. Jameel was hired as a post-doc until July 2021 and S.K. Jena from January 2022.

- 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 will be 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 are 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.
- Phacoemulsification is the procedure commonly used to emulsify and remove the natural optical lens during cataract surgery. It uses an ultrasonically vibrating probe placed in close proximity to the tissue. The cavitation induced at the tip of this probe creates the desired destructive or cleaning effect. A perfusion and vacuum system is built into the probe in order to remove the emulsified tissue. The advantage of the phacoemulsification tool is that it can be inserted through a very small incision in the side of the eye and the old lens removed with minimal invasion. The new artificial lens is then inserted in folded form through the same incision and unfolded in place. More than a million such procedures take place each year. However, the World Health Organization estimates that 17 million more people in the world presently suffer from cataracts. While problems with the procedure are rare, the main concerns are collateral damage to surrounding tissue and the possibility of cavitation damage to the material of the tool itself which might result in metal debris being left behind in the eye.

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

Kevin Schmidmayer was hired on an ISFP position (permanent researcher position). He joined the CAGIRE team in April 2022. His arrival in the team strengthen and extend the research themes in the direction of the advanced modelling and simulation of multiphase compressible flows, with medical and industrial applications.

A permanent Inria engineer position shared between the CAGIRE and CARDAMOM teams has been opened to manage the software development of the Aerosol finite element library. Luca Cirrottola, a former non permanent engineer on the MMG meshing library joined the AeroSol development team in February 2022.

The EB-RSM RANS model, developed by CAGIRE, is now avalaible in the standard version of OpenFOAM[7]. After the open-source code Code Saturne, the commercial code StarCCM+, the codes Cedre of ONERA and Aether of Dassault Aviation, our favorite RANS model is now available in this widespread open-source software.

7 New software and platforms

7.1 New software

7.1.1 AeroSol

Keyword: Finite element modelling

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: The main highlight of 2022 concerning the AeroSol library was the hiring of Luca Cirrottola as an INRIA permanent engineer.

In 2022, the development of the library was focused on the following points

[7] https://www.openfoam.com/documentation/guides/latest/api/classFoam_1_1RASModels_1_1EBRSM.html

* Development environment - Work on the packaging with Guix. - Work on continuous integration by using Plafrim as gitlab runner, with dependencies handled by Guix (and Modules for legacy). - Fix of an old memory leak on PaMPA and integration in the packaging. - Beginning of work for merging the branch master, the branch used for Uhaina/Lagoon project, and the branch including turbulence models and axi models. - the development of a new library DM2 has started. It aims at replacing PaMPA.

* General numerical feature of the library - Postprocessing on wall and lines, including computation of Cp and Cf - New finite elements for quads and hexa, based on Gauss-Lobatto or Gauss-Lagrange elements were added.

* Work on SBM methods - Shifted boundary method for Neumann and Dirichlet boundary conditions, - development of high order derivative in some of the finite element classes

* Low Mach number flows: - Low Mach number filtering was extended on quads. Implementation of dicrete semi-norms div and grad, extension of some fixes to full Euler with arbitrary EOS.

* RANS turbulent flow computations: - Inlet/Outlet boundary conditions for Euler/Navier-Stokes systems allowing to get a stationary solution - HLLC numerical flux, exact jacobian with frozen wave speeds and extension to the RANS equations coupled to transport equations (turbulence models) - test case: laminar and turbulent flat plate, development and documentation of axisymmetric test case, - Fix of Spalart-Almaras model, negative fix handle - Beginning of implementation of coupled elliptic problem with Neumann and Dirichlet boundary conditions on one variable.

* Flux-reconstruction - Implementation of Flux-Reconstruction methods on Cartesian meshes

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

Partner: BRGM

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.

News of the Year: This year, the development of the DM2 library began. We focused on - the mesh reading: the parallel mesh reader of AeroSol was reused and improved. It can

* read a GMSH file with lines, triangles, quads, prisms, tetrahedra or pyramids, each cell or face being straight or curved.

* the mesh reading can be performed in parallel

* the mesh distribution when reading in parallel can rely on a geometrical prepartitioner

- from the mesh reading, a C++ structure graph is built. When read in parallel, it is parallel consistent. Some work was done on the graph structure (c++ containers, adjacency list) for ensuring interoperability with C structures.

" from the c++ structure, a graph c-structure compatible with scotch or metis can be built. A parallel mesh redistribution of this graph can be performed based on these software

Contact: Vincent Perrier

Participants: 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

8 New results

8.1 Turbulence modelling

8.1.1 Improvement of the EB-RSM RANS model

Participants: Rémi Manceau.

External collaborators: Gustave Sporschill (formerly CAGIRE, now Dassault), F. Billard (Dassault), M. Mallet (Dassault), A. Colombié (ONERA), F. Chedevergne (ONERA), E. Laroche (ONERA), 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 [94, 97]. Although this approach, has been successfully applied to various configurations (for instance [56]), 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 [114, 65, 115, 116], [15].

8.1.2 Extension of RANS turbulence models to mixed and natural convection

Participants: Rémi Manceau, Puneeth Bikkanahally, Sofen K. Jena.

External collaborators: S.M. Saad Jameel (formerly Cagire, now Plastic Omnium), V. Herbert (PSA-Stellantis), F. Dehoux (formerly Cagire, now Framatome), S. Benhamadouche (EDF).

In the mixed and natural convection regimes, as presented in two invited lectures [95, 96], the interaction mechanisms between dynamic and thermal fluctuations are complex and very anisotropic due to buoyancy effects, 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 [69]. 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 thesis [85, 87, 86], which can be easily implemented into any industrial and/or commercial CFD code. 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 [21].

8.1.3 HTLES: an original hybrid RANS/LES model

Participants: Rémi Manceau, Puneeth Bikkanahally, Mahitosh Mehta, Martin David.

External collaborators: Vladimir Duffal (formerly Cagire, now EDF), B. de Laage de Meux (EDF), H. Afailal (formerly CAGIRE, now IFPEN), Ch. Angelberger (IFPEN), A. Velghe (IFPEN).

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 [72] 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 [12] [73], i.e., the strong erroneous sensitivity of the results to the near-wall mesh. A significant result is that the study of wall pressure fluctuations and their spectra on periodic hills showed that the HTLES approach could reproduce these spectra as well as LES, down to a lower cut-off frequency than in LES due to the coarser mesh and the presence of the RANS zone [72], which suggests encouraging prospects for the prediction of mechanical and thermal fatigue. In the framework of the ANR project Monaco_2025, the thesis of P. Bikkanahally is devoted to the extension of the HTLES approach 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 [99, 58]. Moreover, the thesis of H. Afailal, in collaboration with IFPEN, was dedicated to the development of the HTLES for the non-reactive internal aerodynamics of spark ignition engines. The aim was to adapt this approach to non-stationary, cyclic flows with moving walls, for which the main challenge was to provide a reliable evaluation of the mean turbulent energy, which is a crucial parameter for the control of the transition from RANS to LES, and is obtained by explicitly applying a differential temporal filter during the simulation to separate the time-dependent mean and turbulent components of the flow [49].

8.1.4 Towards embedded LES

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

External collaborators: Fabien Dupuy (GD-Tech), Olivier Jegouzo (GD-Tech), Fabien Renard (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 [26]. The development of such a CELES approach is the main purpose of the just started 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 is the subject of the post-doc of M. David, in the framework of the Asturies project.

8.2 High-order numerical methods and efficient algorithms

8.2.1 Improvement of scalability through task-based programming

Participants: Vincent Perrier, Jonathan Jung, Sangeeth Simon.

External collaborators: M. Haefele (LMAP), Storm project-team, Hiepacs project-team.

Task based programming has emerged over about a decade as an alternative to classical imperative parallel programming based on MPI or on coarse grain OpenMP, since it provides more flexibility for addressing heterogeneous architectures. Task based programming has began entering the OpenMP standard since the 4.0 version, but still remains in specific libraries such as OMPss [74], Parsec [59] or StarPU [51] for their most advanced features. In [75], we have developed a two dimensional mock-up code for the first order finite volume approximation of the Euler system on structured meshes based on StarPU, yielding interesting results; for example, for a mesh partitioned into N parts, the code may scale with more than N cores; another example is the possibility to temporarily allocate a resource to IO and to use it again for computation once the IO is finished. From 2019 to 2022, S. Simon did his postdoc under the co-supervision of J. Jung, M. Haefele and V. Perrier for extending the mock-up code to second order finite volume, and to the three dimensional discontinuous Galerkin method for the compressible Navier-Stokes system. Within this project, we are also developing roof-line models for analyzing the different algorithms and try to explain their behaviour with respect to the architecture addressed. Currently, the 1D Galerkin method is implemented and the extension to multi-D is in progress. The work done was published in the conferences [35, 36].

8.3 Compressible and multiphase flows

8.3.1 Low-Mach-number schemes

Participants: Jonathan Jung, Vincent Perrier, Ibtissem Lannabi.

Unfortunately, the only fix we found in our previous work [63] that is accurate for high order acoustics computation is not Galilean invariant. This led us to try to tackle the problem in a different way than the numerical flux modification. We raised more fundamental questions on the connection between the low Mach number spurious mode responsible for a low accuracy and the long time behaviour of the wave system. In [88], we proved that on some finite domain configurations, the long time limit of the wave system exists, and that a numerical flux is low Mach number accurate if and only if its low Mach number acoustic development has a consistent long time behaviour. The spurious mode on the velocity at low Mach number can therefore be identified as the non-divergence-free part of the long time limit of the asymptotic acoustic system. Once this spurious mode is sharply identified, it can be filtered. This result is published in [14]. Still based on this filtering method, a spurious mode was identified in the velocity obtained by the low Mach number fix which consists in centering the pressure. This spurious mode may jeopardize the convergence of the velocity. These results were communicated in [38, 37]. We also had the opportunity to disseminate our previous work on low Mach number flows at different conferences [23], included invited ones [22, 24].

8.3.2 Development of high order numerical schemes for axisymmetric turbulent flows

Participants: Anthony Bosco, Jonathan Jung, Vincent Perrier.

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 [17, 19].

2. The derivation of new boundary conditions for Euler and Navier Stokes equations, which allows to reach a steady state. This was validated on a laminar flat plate.

3. The debugging of a Spalart-Almaras model which was implemented in AeroSol but had never been validated. This led to our first RANS computation with the AeroSol library. This was disseminated in [18].

8.3.3 Artificial compressibility method for Mach zero combustion

Participants: Pascal Bruel.

External collaborators: C. Cristaldo (Universidade Federal do Pampa, Brazil), M. Donini (INPE, Brazil), F. Fachini (INPE, Brazil).

As a first step towards the simulation of low Mach reacting flows, an efficient methodology to simulate variable density flows in the Mach zero limit, either inert or reacting was developed. The approach combined a finite volume framework on fully staggered grids with the artificial compressibility method and a dual-time stepping. The resulting code proved to be versatile enough to cope with excellent accuracy with flow configurations ranging from unsteady cylinder wakes to unsteady laminar diffusion flames [70].

8.3.4 Multi-scale multiphase flows

Participants: Vincent Perrier, Kevin Schmidmayer.

As far as multiphase models are concerned, based on the ideas of [71], we have revisited the derivation of Baer-and-Nunziato models [53]. 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 [111] or by ensuring mathematical properties [66]. In [105], 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 [111, 77], which does not necessarily ensure the same phase entropy dissipation properties. This work was disseminated in the conferences [27, 29]. V. Perrier was also invited in [28] for talking about this topic.

8.3.5 Shock-induced cavitation within a droplet

Participants: Kevin Schmidmayer.

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

In [11], 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.

8.3.6 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 [104, 76] 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 [81]) and damage (Mazars [101]) models.

8.4 Analysis and simulation of turbulent flows and heat transfer

8.4.1 Effusion cooling

Participants: Rémi Manceau, Pascal Bruel.

External collaborators: Ph. Reulet (ONERA), E. Laroche (ONERA), D. Donjat (ONERA), F. Mastrippolito (formerly CAGIRE, now SAFRAN HE).

As regards wall cooling by effusion (multiple 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 [107], [25].

8.4.2 Security of reservoirs and pipelines

Participants: Pascal Bruel.

External collaborators: S. Elaskar (University of Cordoba, Argentina), J. Saldia (University of Cordoba, Argentina), L. Gutiérrez Marcantoni (University of Cordoba, Argentina), A. Beketaeva (Institute of Mathematics and Mathematical Modelling, Almaty, Kazakhstan), A. Naimanova (Institute of Mathematics and Mathematical Modelling, Almaty, Kazakhstan).

In the framework of the cooperation with our international partners in Kazakhstan and Argentina, simulations of turbulent flows in different configurations were performed. The flow configurations ranged from the injection of a sonic jet in a supersonic crossflow [54], the interaction of an atmospheric boundary layer with aerial reservoir(s) [110] or of a channel flow with cylinders in tandem [79] to the simulation of a blast wave of Sedov-like type [82]. The objectives of these simulations were twofold: 1) Assessing the predictive capabilities of conventional RANS approaches by comparisons with experimental data in order to establish the margin of progression that could be subsequently brought about by the recourse to the more elaborated turbulence models developed by our team and 2) Providing a knowledge of the sensitivity of the different flow topologies and characteristics to the variation of the relevant parameters describing the different configurations at hand. In parallel, our test facility MAVERIC was upgraded in order to accompany the forthcoming validation activities in the framework of the ASTURIES project and the partnership with our Argentinian colleagues.

8.4.3 Thermocline energy storage

Participants: Rémi Manceau, Alexis Ferré.

External collaborators: S. Serra (LaTEP, UPPA), J. Pouvreau (CEA), A. Bruch (CEA).

A collaboration started at the end of 2020 with the CEA LITEN in Grenoble and the LaTEP of UPPA on thermocline energy storage. An experimental facility is being developed at the CEA and RANS simulations are underway 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 [20].

9 Bilateral contracts and grants with industry

Participants: Rémi Manceau, Vincent Perrier, Jonathan Jung, Pascal Bruel, Gustave Sporschill, Anthony Bosco, Mahitosh Mehta, Romaric Simo Tamou, Alexis Ferré, Esteban Coiffier.

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".

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: “Analyse et simulation numériques de discrétisations décalées en thermohydraulique diphasique”, PhD student Esteban Coiffier.

10 Partnerships and cooperations

10.1 International initiatives

10.1.1 Participation in other International Programs

ECOS-Sud A17A07 project with UNC (Córdoba, Argentine). 2022 was the last year of this project devoted to the experimental and numerical study of the wind around aerial fuel tanks. [13] [16].

10.2 International research visitors

10.2.1 Visits of international scientists

Other international visits to the team

Sergio Elaskar

Status: researcher

Institution of origin: National University of Córdoba

Country: Argentine

Dates: October 31 - November 14

Context of the visit: Ecos Sud project

Mobility program/type of mobility: research stay

10.2.2 Visits to international teams

Research stays abroad

Pascal Bruel

Visited institution: National University of Córdoba

Country: Argentine

Dates: November 29 - December 15

Context of the visit: Ecos Sud project

Mobility program/type of mobility: reserch stay

10.3 National initiatives

10.3.1 ANR MONACO_2025

Participants: Rémi Manceau.

The ambition of the MONACO_2025 project, coordinated by Rémi Manceau, is to join the efforts made in *two different industrial sectors* in order to tackle the industrial simulation of transient, turbulent flows affected by buoyancy effects. It brings together two academic partners, the project-team Cagire hosted by the university of Pau, and the institute Pprime of the CNRS/ENSMA/university of Poitiers (PPRIME), and R&D departments of two industrial partners, the PSA group and the EDF group, who are major players of the automobile and energy production sectors, respectively.

- The main **scientific objective** of the project is to make a breakthrough in *the unresolved issue* of the modelling of turbulence/buoyancy interactions in transient situations, within the continuous hybrid RANS/LES paradigm, which consists in preserving a computational cost compatible with industrial needs by relying on statistical approaches where a fine-grained description of the turbulent dynamics is not necessary. The transient cavity flow experiments acquired during MONACO_2025 will provide the partners and the scientific community with *an unrivalled source of knowledge* of the physical mechanisms that must be accounted for in turbulence models.
- The main **industrial objective** is *to make available computational methodologies* to address dimensioning, reliability and security issues in buoyancy-affected transient flows. It is to be emphasized that such problems are *not tackled using CFD at present in the industry.* At the end of MONACO_2025, a panel of methodologies, ranging from simple URANS to sophisticated hybrid models based on improved RANS models, was evaluated in transient situations, against the dedicated cavity flow experiments and a real car underhood configuration. This final benchmark exercise will form *a decision-making tool* for the industrial partners, and will thus pave the way

towards high-performance design of low-emission vehicles and highly secure power plants. In particular, the project is in line with the *Full Digital 2025 ambition*, e.g., the declared ambition of the PSA group to migrate, within the next decade, to a design cycle of new vehicles nearly entirely based on CAE (computer aided engineering), without recourse to expensive full-scale experiments.

10.3.2 ANR LAGOON

Participants: Vincent Perrier.

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

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

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

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

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

10.3.3 ASTURIES

Participants: Rémi Manceau, Vincent Perrier, Jonathan Jung, Pascal Bruel, Anthony Bosco, Mahitosh Mehta, 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 consist 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.

The development of such a methodology, based on hybrid RANS/LES modelling, with low-dissipative and robust numerical methods, independant of the initial design of a grid by the user, compatible with unstructured meshes for complex industrial geometries, in the context of HPC, is thus the ambitious, but reachable, objective of the project.

10.4 Regional initiatives

10.4.1 HPC scalable ecosystem

Participants: Jonathan Jung, Vincent Perrier, Sangeeth Simon.

HPC scalable ecosystem was a 3-year program funded by Région Nouvelle-Aquitaine (call 2018), Airbus, CEA-CESTA, University of Bordeaux, INRA, ISAE-ENSMA and Inria. Sangeeth Simon was hired as a post-doc with the objective of extending the prototype code developed in [75] to high order (discontinuous Galerkin) and non-reactive diffusive flows in 3D. The same basis will be developed in collaboration with Pprime for WENO based methods for reactive flows.

10.4.2 CELTIC

Participants: Rémi Manceau, Pascal Bruel, Puneeth Bikkanahally.

CELTIC (*Continuous Embedded LES for Turbulent flows in Industrial Configurations*) is a projet accepted in the call ESR 2022 of the New-Aquitaine Region, in partnership with the local SME GDTech.

The project aims at proposing a new model of the forcing term allowing enrichment at the diffuse interfaces between RANS and LES in continuous hybrid RANS/LES models for the simulation of turbulent flows. This will make possible the development of an innovative simulation methodology, continuous embedded LES (CELES), which consists in restricting the fine-grained model (LES) to regions where it is necessary, surrounded by the rest of the domain using a statistical approach (RANS). The combination of the fact that CELES will be based on hybrid RANS/Continuous LES approaches, easy to use in industrial applications, and on a seeding by volume forcing applicable in all situations will make it a particularly attractive method for the industry, which will bring to our industrial partner GDTech and its regional customers (for example SAFRAN HE) a real value compared to the numerical simulation methods available today.

11 Dissemination

Participants: Rémi Manceau, Jonathan Jung, Vincent Perrier, Kevin Schmidmayer, Pascal Bruel, Martin David, Mahitosh Mehta, Ibtissem Lannabi, Anthony Bosco.

11.1 Promoting scientific activities

11.1.1 Scientific events: organisation

General chair, scientific chair

- Rémi Manceau organized the 17th ERCOFTAC SIG15 / MONACO_2025 workshop[8]. This workshop, organized under the auspices of the SIG15 of the european ERCOFTAC network, which is also the final workshop of the MONACO_2025 project, was held in Pau 19-20 January 2023.

Member of the organizing committees

- Jung Jonathan was a member of the organizing committee of the workshop "New trends for complex flows" conference hold in Paris from September 19 to 21, 2022.

11.1.2 Scientific events: selection

Chair of conference program committees

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

11.1.3 Journal

Member of the editorial boards

- Advisory Board of International Journal of Heat and Fluid Flow [Rémi Manceau]
- Advisory Board of Flow, Turbulence and Combustion [Rémi Manceau]

Reviewer - reviewing activities

- Aerosp. Sci. Technol. (2) [Pascal Bruel]
- Comput. Fluids (3) [Pascal Bruel]
- Int. J. Heat Fluid Flow (2) [Rémi Manceau]
- J. Marine Sci. Eng. (1) [Rémi Manceau]
- J. Comp. Phys. (5) [Rémi Manceau (1), Vincent Perrier (1), Kevin Schmidmayer (2), Jonathan Jung (1)]
- Computers and Mathematics with applications [Vincent Perrier]
- Scientific Reports (1) [Kevin Schmidmayer]

[8] https://project.inria.fr/monaco2025ercoftacsig15workshop/

11.1.4 Invited talks

[28] V. Perrier. “Stochastic derivation of models and numerical methods for multiphase compressible flows”. In: 34e Séminaire sur la mécanique des fluides numérique. Paris, France, 24th Jan. 2022.
[25] F. Mastrippolito, R. Manceau and P. Bruel. “Modélisation des effets de giration pour les jets en écoulement transverse”. In: Journée thématique SFT : Aérothermique des systèmes propulsifs pour l’aéronautique. Paris, France, 20th Oct. 2022.
[24] J. Jung, V. Perrier and I. Lannabi. “On low Mach number fixes for Roe-type schemes”. In: Rencontres de la thermohydraulique numérique du CEA/STMF. Saint-Rémy-lès-Chevreuse, France, 28th June 2022.
[22] Jonathan Jung, Vincent Perrier "Relations between the low Mach number problem and the long time limit of the wave system". In Rencontres de la thermohydraulique numérique du CEA/STMFSaint-Rémy-lès-Chevreuse, France, 27th June, 2022
[32] K. Schmidmayer. “Modeling and simulation of bubble dynamics in the diffuse-interface framework”. In: Workshop on Cavitation, bubble dynamics and mass transfer. Strasbourg, France, 31st May 2022.
[33] K. Schmidmayer. “Modeling and simulation of bubble dynamics in the diffuse-interface framework”. In: ETH Zürich seminar. Zürich, Switzerland, 22th August 2022.
[30] K. Schmidmayer. “Droplet shock-induced cavitation using a multiphase modelling approach”. In: Sorbonne University Seminar, Pierre and Marie Curie Campus, Institut Jean le Rond ∂’Alembert. Paris, France, 8th December 2022.

11.1.5 Leadership within the scientific community

- R. Manceau has been a member of the Standing committee of the Special Interest Group *Turbulence modelling* (SIG-15) of ERCOFTAC since 2005, together with 9 other committee members (S. Jakirlić [chairman], F. Menter, S. Wallin, D. von Terzi , B. Launder, K. Hanjalić, W. Rodi, M. Leschziner, D. Laurence). The main activities of the group is to organize international workshops and thematic sessions in international congresses.

- Rémi Manceau coordinates the ANR Project MONACO_2025, a 4-year project started in 2018. The partners are: the institute PPrime, PSA Group and EDF.

- Rémi Manceau coordinates the 4-year E2S-UPPA project ASTURIES, which involves CEA and IFPEN.

- Vincent Perrier coordinates the ANR Project LAGOON, a 4-year project started in 2022. The partners are: the CARDAMOM project-team of Inria Bordeaux and the BRGM.

11.1.6 Research administration

- Co-responsible for the organisation of the LMAP seminar of Mathematics and their Applications [Jonathan Jung].

- 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].

- Elected member of the Inria evaluation committee and member of the board [Vincent Perrier]. [9]

- 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]

[9] https://www.inria.fr/en/inria-evaluation-committee

11.2 Teaching - Supervision - Juries

11.2.1 Teaching

- L1 [J. Jung]: Research and innovation, 1.5h/year, Université de Pau et des Pays de l'Adour, Pau, France.
- L2 [J. Jung]: Numerical analysis for vectorial problems, 42.75h/year, Mathematics, Université de Pau et des Pays de l'Adour, Pau, France.
- M1 [J. Jung]: Tools for scientific computing, 48h75/year, MMS, Université de Pau et des Pays de l'Adour, Pau, France.
- M2 [J. Jung, V. Perrier]: Finite volume scheme for hyperbolic systems, 24h/year, MMS, Université de Pau et des Pays de l'Adour, Pau, France.
- M1 [J. Jung]: Supervised personal work, 5 h/year, MMS, Université de Pau et des Pays de l'Adour, Pau, France.
- M2: [V. Perrier], Finite volume scheme for hyperbolic systems, Master MMS, Pau. 24h/year.
- 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 [47].
- E3 [R. Manceau]: Advanced physics–Turbulence modelling for CFD, 16h/year, ENSGTI, France [98].
- M1 [K. Schmidmayer]: Basics of heat transfer (in English), 30h/year, International Master's SIMOS, ENSGTI, Pau, France.
- M1 [K. Schmidmayer]: Heat exchangers (in English), 12h/year, International Master's SIMOS, ENSGTI, Pau, France.
- E2 [K. Schmidmayer]: Coupled heat transfers, 12h/year, ENSGTI, Pau, France.
- E3 (M. David): Advanced physics (classes: 6h/year, labs: 5h/year) ENSGTI
- E1 (M. David): Modelling and simulation in fluid mechanics (lectures: 4h/year; classes: 16h/year). ENSGTI.
- E3 (M. Mehta):Modelling of energy systems (40.5 h/year). ENSGTI
- L1 (I. Lannabi): Descriptive statistics (22.5h/year, labs), Mathematics & MIASHS. University of Pau (UPPA).
- 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

- PhD in progress: Puneeth Bikkanahally Muni Reddy, "Modelling turbulent flows in natural convection regimes using hybrid RANS-LES approaches", UPPA, ANR project Monaco_2025, Rémi Manceau.
- PhD in progress: Mahitosh Mehta, "Development of an agile methodology for hybrid RANS-LES computations of turbulent flows", UPPA, E2S-UPPA Asturies project, Rémi Manceau

- 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: 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: 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.
- PhD in progress: Ibtissem Lannabi , “Discontinuous Galerkin methods for low Mach flows in fluid mechanics”, EDENE project (H2020 Marie-Sklodowska-Curie COFUND), Jonathan Jung and Vincent Perrier.
- PhD in progress: Esteban Coiffier, "Analyse et simulation numériques de discrétisations décalées en thermohydraulique diphasique", CEA-Saclay, Vincent Perrier & Jonathan Jung.

11.2.3 Juries

- Reviewer of the PhD thesis of J. Janin, Aix-Marseille university [R. Manceau]
- Reviewer of the PhD thesis of J. Cazé, Aix-Marseille university [V. Perrier]
- President of the jury for the PhD thesis of E. Rondeaux, university Paris-Saclay [R. Manceau]
- Member of the defense committee of the PhD thesis of A. Weppe, ENSMA [R. Manceau]
- Member of the defense committee of the PhD thesis of M. Carlsson, Chalmers Technical University, Sweden [R. Manceau]
- Member of the defense committee of the PhD thesis of S. Michel, Bordeaux University, France [V. Perrier]
- Reviewer and president of the defense committee of the HDR of S. Abide, Perpignan University [P. Bruel]

11.3 Popularization

11.3.1 Interventions

[39] J. Jung. Les mathématiques appliquées au service de la sureté nucléaire: Nuit européenne des chercheurs, château de Pau, 30th Sept. 2022.
Intervention about "scientists facing war" following the movie "Adventures of a Mathematician", adapted from the autobiography of Stan Ulam at Cinéma de Mourenx (Vincent Perrier).
[40] J. Jung. Quelques applications de la recherche en mathématiques appliquées. 4th May 2022.
[41] J. Jung. Quelques applications de la recherche en mathématiques appliquées. 6th May 2022.
[42] I. Lannabi and J. Cresson. Village des sciences. 15th Oct. 2022.
[43] I. Lannabi and I. Greff. Journée Maths et filles. 8th Dec. 2022.
[44] R. Manceau. Prévoir l’imprévisible. Nuit Européenne des chercheur.e.s, Château de Pau. Pau, France, 30th Sept. 2022.
[45] K. Schmidmayer. ’Les bulles de cavitation et leurs interventions inattendues en médecine’. In: Nuit Européenne des chercheur.e.s, Château de Pau. Pau, France, 30th Sept. 2022.

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: *Int. J. Heat Fluid Fl.* 51 (2015), pp. 195–220. DOI: 10.1016/j.ijheatfluidflow.2014.09.002. URL: http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.09.002.

[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] 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.

[12] 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://hal.inria.fr/hal-03206747.

[13] M. Grioni, P. Bruel, S. A. Elaskar and A. E. Mirasso. 'An application of the scale-adapted simulation to the unsteady flow across a tube bundle'. In: *International Journal of Heat and Fluid Flow* 96 (2022), p. 109007. DOI: 10.1016/j.ijheatfluidflow.2022.109007. URL: https://hal.archives-ouvertes.fr/hal-03707804.

[14] 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.archives-ouvertes.fr/hal-03736193.

[15] 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://hal.inria.fr/hal-03597506.

National peer-reviewed Conferences

[16] M. Grioni, S. Elaskar, P. Bruel and A. Mirasso. 'Comparación de algoritmos de acoplamiento velocidad-presión para problemás no estacionarios del flujo alrededor de un cilindro circular'. In: 2022 IEEE Biennial Congress of Argentina (ARGENCON). San Juan, Argentina: IEEE, 7th Sept. 2022. DOI: 10.1109/ARGENCON55245.2022.9939984. URL: https://hal.inria.fr/hal-03851685.

Conferences without proceedings

[17] A. Bosco and V. Perrier. 'Discontinuous Galerkin method for the simulation of axisymmetric compressible flows'. In: HONOM 2022 - European Workshop on High order nonlinear numerical methods for evolutionary PDEs: Theory and Applications, Braga, France, 4th Apr. 2022. URL: https://hal.inria.fr/hal-03893160.

[18] A. Bosco and V. Perrier. 'Méthodes numérique d'ordre élevé pour la turbulence.' In: Journée du LRC Anabase avec MARGAUx. Talence, France, 5th Dec. 2022. URL: https://hal.inria.fr/hal-03893271.

[19] A. Bosco, V. Perrier and J. Jung. 'Discontinuous Galerkin method for the computation of axisymmetric flows'. In: ECCOMAS CONGRESS 2022 8th European Congress on Computational Methods in Applied Sciences and Engineering. Oslo, Norway, 5th June 2022. URL: https://hal.inria.fr/hal-03893243.

[20] 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: Congrès de la Société Française Thermique SFT. Valenciennes, France, 31st May 2022. URL: https://hal.archives-ouvertes.fr/hal-03692645.

[21] 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://hal.inria.fr/hal-03928060.

[22] J. Jung and V. Perrier. 'Relations between the low Mach number problem and the long time limit of the wave system'. In: Rencontres de la thermohydraulique numérique du CEA/STMF. Saint-Rémy-lès-Chevreuse, France, 27th June 2022. URL: https://hal.inria.fr/hal-03893258.

[23] J. Jung and V. Perrier. 'Relations between the low Mach number problem and the long time limit of the wave system'. In: HONOM 2022 - European Workshop on High order nonlinear numerical methods for evolutionary PDEs: Theory and Applications. Braga, Portugal, 4th Apr. 2022. URL: https://hal.inria.fr/hal-03893079.

[24] J. Jung, V. Perrier and I. Lannabi. 'On low Mach number fixes for Roe-type schemes'. In: Rencontres de la thermohydraulique numérique du CEA/STMF. Saint-Rémy-lès-Chevreuse, France, 28th June 2022. URL: https://hal.inria.fr/hal-03893261.

[25] F. Mastrippolito, R. Manceau and P. Bruel. 'Modélisation des effets de giration pour les jets en écoulement transverse'. In: Journée thématique SFT : Aérothermique des systèmes propulsifs pour l'aéronautique. Paris, France, 20th Oct. 2022. URL: https://hal.inria.fr/hal-03836384.

[26] 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, 26th Oct. 2022. URL: https://hal.inria.fr/hal-03832703.

[27] V. Perrier. 'Derivation of models and numerical methods for homogenized multiphase flows based on stochastic ideas'. In: ECCOMAS CONGRESS 2022 8th European Congress on Computational Methods in Applied Sciences and Engineering. Oslo, Norway, 5th June 2022. URL: https://hal.inria.fr/hal-03893230.

[28] V. Perrier. 'Stochastic derivation of models and numerical methods for multiphase compressible flows'. In: 34ème Séminaire sur la mécanique des fluides numérique. Paris, France, 24th Jan. 2022. URL: https://hal.inria.fr/hal-03893064.

[29] V. Perrier and K. Schmidmayer. 'A multiscale numerical scheme for the simulation of dispersed multiphase flows'. In: MultiMat 2022 - 10th International Conference on Numerical Methods for Multi-Material Fluid Flow. Zürich, Switzerland, 22nd Aug. 2022. URL: https://hal.archives-ouvertes.fr/hal-03752083.

[30] K. Schmidmayer. 'Droplet shock-induced cavitation using a multiphase modelling approach'. In: Sorbonne University Seminar, Pierre and Marie Curie Campus, Institut Jean le Rond d'Alembert. Paris, France, 8th Dec. 2022. URL: https://hal.science/hal-03879752.

[31] K. Schmidmayer. 'Droplet shock-induced cavitation using a multiphase modelling approach'. In: LMAP seminar. Pau, France, 5th Jan. 2023. URL: https://hal.archives-ouvertes.fr/hal-03924694.

[32] K. Schmidmayer. 'Modeling and simulation of bubble dynamics in the diffuse-interface framework'. In: Workshop on Cavitation, bubble dynamics and mass transfer. Strasbourg, France, 31st May 2022. URL: https://hal.archives-ouvertes.fr/hal-03752075.

[33] K. Schmidmayer. 'Modeling and simulation of bubble dynamics in the diffuse-interface framework'. In: ETH Zürich seminar. Zürich, Switzerland, 22nd Aug. 2022. URL: https://hal.archives-ouvertes.fr/hal-03879749.

[34] K. Schmidmayer, J. Cazé, F. Petitpas, E. Daniel and N. Favrie. 'Modelling interactions of waves with diffused interfaces'. In: MultiMat 2022 - 10th International Conference on Numerical Methods for Multi-Material Fluid Flow. Zürich, Switzerland, 22nd Aug. 2022. URL: https://hal.archives-ouvertes.fr/hal-03752096.

[35] S. Simon, V. Perrier, J. Jung and M. Haefele. 'A task-based parallelization of a finite volume code for hyperbolic conservation law'. In: 33rd parallel CFD International conference. Alba, Italy, 25th May 2022. URL: https://hal.inria.fr/hal-03893211.

[36] S. Simon, V. Perrier, J. Jung and M. Haefele. 'Design And Analysis Of Task-based Parallelization Of A Discontinuous Galerkin Euler Flow Solver On Heterogeneous Architectures'. In: ECCOMAS CONGRESS 2022 8th European Congress on Computational Methods in Applied Sciences and Engineering. Oslo, Norway, 5th June 2022. URL: https://hal.inria.fr/hal-03893249.

Other scientific publications

[37] I. Lannabi, J. Jung and V. Perrier. 'Low Mach number flows: a cure to checkerboard modes'. In: New trends in complex flows. Paris, France, 19th Sept. 2022. URL: https://hal.archives-ouvertes.fr/hal-03894133.

[38] I. Lannabi, J. Jung and V. Perrier. 'Low Mach number flows: a cure to checkerboard modes'. In: Sixteenth International Conference on Mathematics and its Applications. Jaca, Spain, 7th Sept. 2022. URL: https://hal.science/hal-03894207.

12.3 Other

Scientific popularization

[39] J. Jung. *Les mathématiques appliquées au service de la sureté nucléaire: Nuit européenne des chercheurs, château de Pau,* 30th Sept. 2022. URL: https://hal.archives-ouvertes.fr/hal-03881595.

[40] J. Jung. *Quelques applications de la recherche en mathématiques appliquées.* 4th May 2022. URL: https://hal.archives-ouvertes.fr/hal-03881588.

[41] J. Jung. *Quelques applications de la recherche en mathématiques appliquées.* 6th May 2022. URL: https://hal.archives-ouvertes.fr/hal-03881604.

[42] I. Lannabi and J. Cresson. *Village des sciences.* 15th Oct. 2022. URL: https://hal.science/hal-03894228.

[43] I. Lannabi and I. Greff. *Journée Maths et filles.* 8th Dec. 2022. URL: https://hal.archives-ouvertes.fr/hal-03894193.

[44] R. Manceau. *Prévoir l'imprévisible.* 30th Sept. 2022. URL: https://hal.inria.fr/hal-03795816.

[45] K. Schmidmayer. 'Les bulles de cavitation et leurs interventions inattendues en médecine'. In: Nuit Européenne des chercheur.e.s, Château de Pau. Pau, France, 30th Sept. 2022. URL: https://hal.archives-ouvertes.fr/hal-03794501.

Educational activities

[46] R. Manceau. 'Codes de calcul industriels pour la simulation des écoulements turbulents'. Master. France, Feb. 2022. URL: https://hal.inria.fr/hal-03207435.

[47] R. Manceau. 'Industrial codes for CFD'. Master. France, Feb. 2022. URL: https://hal.inria.fr/hal-03207431.

12.4 Cited publications

[48] 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.

[49] A. H. Afailal. 'Numerical simulation of non-reactive aerodynamics in Internal Combustion Engines using a hybrid RANS/LES approach'. Theses. Université de Pau et des Pays de l'Adour, Dec. 2020. URL: https://tel.archives-ouvertes.fr/tel-03140392.

[50] 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.

[51] C. Augonnet, S. Thibault, R. Namyst and P.-A. Wacrenier. 'StarPU: a unified platform for task scheduling on heterogeneous multicore architectures'. In: *Concurrency and Computation: Practice and Experience* 23.2 (2011), pp. 187–198.

[52] 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.

[53] 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.

[54] A. Beketaeva, P. Bruel and A. Z. Naimanova. 'Detailed Comparative Analysis of Interaction of a Supersonic Flow with a Transverse Gas Jet at High Pressure Ratios'. In: *Technical Physics / Zhurnal Tekhnicheskoi Fiziki* 64.10 (Oct. 2019), pp. 1430–1440. DOI: 10.1134/S1063784219100049. URL: https://hal.inria.fr/hal-02373910.

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

[56] 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.

[57] 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).

[58] 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.

[59] G. Bosilca, A. Bouteiller, A. Danalis, M. Faverge, T. Hérault and J. J. Dongarra. 'Parsec: Exploiting heterogeneity to enhance scalability'. In: *Computing in Science & Engineering* 15.6 (2013), pp. 36–45.

[60] 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.

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

[62] 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.

[63] 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.

[64] 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.

[65] 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.

[66] 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.

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

[68] 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.

[69] 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.

[70] M. Donini, F. Fachini, C. Cristaldo and P. Bruel. 'An Artificial Compressibility Based Approach to Simulate Inert and Reacting Flows'. In: *Journal of Fluid Flow, Heat and Mass Transfer* (2021). DOI: 10.11159/jffhmt.2021.010. URL: https://hal.inria.fr/hal-03149751.

[71] D. A. Drew and S. L. Passman. *Theory of multicomponent fluids*. Vol. 135. Springer Science & Business Media, 2006.

[72] 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.

[73] 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.

[74] A. Duran, E. Ayguadé, R. M. Badia, J. Labarta, L. Martinell, X. Martorell and J. Planas. 'Ompss: a proposal for programming heterogeneous multi-core architectures'. In: *Parallel processing letters* 21.02 (2011), pp. 173–193.

[75] M. Essadki, J. Jung, A. Larat, M. Pelletier and V. Perrier. 'A task-driven implementation of a simple numerical solver for hyperbolic conservation laws'. In: *ESAIM: Proceedings and Surveys* 63 (Jan. 2017). https://arxiv.org/abs/1701.05431, pp. 228–247. DOI: 10.1051/proc/201863228. URL: https://hal.archives-ouvertes.fr/hal-01439322.

[76] 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.

[77] 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.

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

[79] M. Grioni, S. Elaskar, A. Mirasso and P. Bruel. 'Flow interference between circular cylinders in tandem arrangement near to a plane wall'. In: *Mecánica Computacional* 37.26 (Nov. 2019), pp. 1065–1074. URL: https://hal.inria.fr/hal-02376562.

[80] 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.

[81] 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.

[82] L. Gutiérrez Marcantoni, S. Elaskar, J. Tamagno and P. Bruel. 'Simulation of blast waves using OpenFOAM'. In: *Mecánica Computacional* 37.26 (Nov. 2019), pp. 1075–1084. URL: https://hal.inria.fr/hal-02376633.

[83] 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.

[84] 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.

[85] 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.

[86] 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.

[87] 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.

[88] J. Jung and V. Perrier. 'Long time behavior of finite volume discretization of symmetrizable linear hyperbolic systems'. In: *IMA Journal of Numerical Analysis* (Dec. 2021). DOI: 10.1093/imanum/drab092. URL: https://hal.archives-ouvertes.fr/hal-03503847.

[89] 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.

[90] 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.

[91] 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.

[92] 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.

[93] 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.

[94] 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.

[95] 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.

[96] 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.

[97] 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.

[98] R. Manceau. 'Turbulence modelling for CFD'. Master. Lecture. Pau, France, Nov. 2020. URL: https://hal.inria.fr/hal-03207433.

[99] R. Manceau and P. Bikkanahally. 'Hybrid Temporal LES: from theory to applications'. In: *HiFiLeD - 2nd High Fidelity Industrial LES/DNS Symposium*. Toulouse / Virtual, France, Sept. 2021. URL: https://hal.inria.fr/hal-03366801.

[100] 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.

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

[102] 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.

[103] 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.

[104] 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.

[105] 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.

[106] 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.

[107] 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.

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

[109] 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.

[110] J. Saldía, G. Krause, S. Elaskar and P. Bruel. 'Modelizacion numerica de cargas de viento sobre un tanque de almacenamiento de combustible'. In: *Mecánica Computacional* 37.27 (Nov. 2019), pp. 1163–1175. URL: https://hal.inria.fr/hal-02376676.

[111] 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.

[112] 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.

[113] 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.

[114] 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.

[115] 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.

[116] 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.

[117] 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.