RESEARCH CENTRE

**Bordeaux - Sud-Ouest**

**IN PARTNERSHIP WITH:**

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

2021

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. – Mathematical modeling for physical sciences

A6.5.2. – Fluid mechanics

### Other research topics and application domains

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]

## Faculty Member

- Jonathan Jung [Univ de Pau et des pays de l'Adour, Associate Professor]

## Post-Doctoral Fellows

- Syed Jameel [Univ de Pau et des pays de l'Adour]
- Sangeeth Simon [Inria]

## PhD Students

- Puneeth Bikkanahally Muni Reddy [Univ de Pau et des pays de l'Adour]
- Anthony Bosco [Univ de Pau et des pays de l'Adour]
- Alexis Ferre [CEA]
- Ibtissem Lannabi [Univ de Pau et des pays de l'Adour, from Oct 2021]
- Mahitosh Mehta [Univ de Pau et des pays de l'Adour]
- Romaric Simo Tamou [IFPEN, from Oct 2021]
- Gustave Sporschill [Dassault Aviation, until Apr 2021]

## Interns and Apprentices

- Lou Guerin [Université de Paris, from Jun 2021 until Aug 2021]
- Nicolas Victorion [Univ de Pau et des pays de l'Adour, from Apr 2021 until Aug 2021]

## 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. In the near future, this panel will also be extended to medical applications, 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.

Even though it is far from being the only complex phenomenon to be taken into account and the only subject of our research, turbulence plays a central role in this project 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. However, these choices must also take into account the fact that the applications may involve many other important phenomena: for example, shocks; but also couplings of low-Mach-number aerodynamics with acoustic waves; multiphase flows; variable density; conjugate heat transfer; etc.

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 (DNS[1], LES[2], RANS[3], Hybrid RANS-LES) 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 all these 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. It will thus lead to a description of turbulence at widely varying scales in the computational domain. For example, the RANS method may cover regions where turbulence is sufficiently close to equilibrium, leaving to LES the regions where the RANS description is insufficient. 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
- Development of specific numerical schemes
- 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 to the choices made by the user (in particular the near-wall mesh). Our research work, described below, aims at answering these questions.

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

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, [86, 63, 37, 87]), 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 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, [72, 70]), 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 [49] and PITM [47]); IMF Toulouse in collaboration with the ECUADOR team of the Inria center of Sophia-Antipolis (OES [43, 79]) and CAGIRE (HTLES [69, 34] [13, 20]). 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 [57, 32], 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 [4] (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 [5], or Ihme's group[6] 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 [67] based on the Charm++ framework, and within a European project[7], based on IntelTBB, but only for hyperbolic systems whereas we wish to address the compressible Navier-Stokes system.

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

### 3.3 Development of specific numerical schemes

In this section, we are interested in two specific regimes of compressible flows: low Mach number flows and compressible multi-phase 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 CFL is very restrictive compared with the convective one in the low Mach number limit [50]. 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 [81] or are variant of the Roe-Turkel fix [60]. 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 [78].

Derivation of averaged compressible multi-phase models is currently less active than in the 2000s, and only few teams are interested in such problems. Recent advances were made at RWTH [62], and also mostly in France at EDF R&D by J.M. Hérard or also by [45]. This low interest in this type of challenging modeling and mathematical analysis was noticed in the review paper [84] as an obstacle for the improvement of numerical methods.

### 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 [68, 71, 40, 74], our strategy is to generate experimental data ourselves when possible and to 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, with our industrial partners SAFRAN and Turbomeca (now SAFRAN-Helicopter Engines). If the scientific and application themes of the team are 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 [80] 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 is the subject of the CIFRE PhD thesis of Gustave Sporschill, defended in 2021, 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.

- 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 [75]. 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 recent 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 [16].

- 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 recent PhD thesis (CIFRE PSA) of S. Jameel, supervised by R. Manceau, and also a part of the ANR project MONACO_2025 described in section 10.2.1, in the framework of which S. Jameel was hired as a post-doc until July 2021.
- 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 just-started 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.

## 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 will join the CAGIRE team in April 2022. His arrival in the team will 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 in 2021 to manage the software development of the AEROSOL finite element library. Luca Cirrottola, a former non permanent engineer on the MMG meshing library will join the AEROSOL development team in February 2022.

The ANR project LAGOON has been accepted. Coordinated by V. Perrier, it includes CAGIRE, CARDAMOM and the BRGM. It aims at improving implicit multi-grid methods and large scale IO for near shore applications.

Beyond the open-source code CODE_SATURNE, developed by EDF [35], and the commercial code STARCCM+, the EB-RSM RANS model developed by the team is now available in the codes CEDRE of ONERA and AETHER of Dassault Aviation.

The hybrid RANS/LES model HTLES, developed by the team, has been implemented in the commercial code STAR CCM+, under the name SRH (Scale-Resolving Hybrid), and has shown [76] its robustness and advantages over the most widely used method (DDES).

# 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:** In 2021, the development of the library was focused on the following points

* Development environment - Plugin : Uhaina, GeoFun, allow for plugins to register numerical schemes. - Time dependent data variables. - CI on Plafrim, intel compiler, Clang compiler

* General numerical feature of the library - Implementation of the Taylor finite element basis. - Implementation of the Gauss-Lobatto finite element basis for quads and lines - Implementation of higher order derivatives into finite element basis. - Crouzeix-Raviart, Rannacher-Turek nonconforming methods for the Laplace equation. - Dual consistent integration of source term involving gradients

* Work on SBM methods

* Low Mach number flows: - Implementation of the Euler and Waves models with porosity - initialization of a vector with a potential function. - Implementation of the Bouchut-Chalons-Guisset numerical flux. - Low Mach number filtering.

* CG implementations: - Implementation of cubature elements with mass lumping - Development of the Continuous Interior Penalty method - parallel validation of CG

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

**Contact:** Vincent Perrier

**Participants:** Benjamin Lux, Damien Genet, Mario Ricchiuto, Vincent Perrier, Héloïse Beaugendre, Subodh Madhav Joshi, Christopher Poette, Marco Lorini, Jonathan Jung, Enrique Gutierrez Alvarez, Anthony Bosco

**Partner:** BRGM

# 8 New results

## 8.1 Turbulence modelling

### 8.1.1 Improvement of the EB-RSM RANS model

**Participants:** Rémi Manceau, Gustave Sporschill.

**External collaborators:** F. Billard (Dassault), M. Mallet (Dassault), A. Colombié (ONERA), F. Chedevergne (ONERA), E. Laroche (ONERA), S. Benhamadouche (EDF), J.-F. Wald (EDF).

In order 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 [72, 70]. Although this approach, has been successfully applied to various configurations (for instance [41]), 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 [27, 10, 25].

### 8.1.2 Extension of RANS turbulence models to mixed and natural convection

**Participants:** Rémi Manceau, Puneeth Bikkanahally, S.M. Saad Jameel.

**External collaborators:** V. Herbert (PSA-Stellantis), S. Benhamadouche (EDF).

In the mixed and natural convection regimes, as presented in two invited lectures [73, 74], 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 [51]. 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 [64, 66, 65], which can be easily implemented into any industrial and/or commercial CFD code.

#### 8.1.3 HTLES: an original hybrid RANS/LES model

**Participants:** Rémi Manceau, Puneeth Bikkanahally, Mahitosh Mehta.

**External collaborators:** Vladimir Duffal (EDF), B. de Laage de Meux (EDF), H. Afailal (formerly CAGIRE/IFPEN, now Framatome), 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 [53] 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 [13, 17], 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 [53], 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 [23, 20]. Finally, 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 [33].

### 8.2 High-order numerical methods and efficient algorithms

#### 8.2.1 Improvement of scalability through task-based programming

**Participants:** Vincent Perrier, Jonathan Jung.

**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 [54], Parsec [42] or StarPU [36] for their most advanced features. In [55], 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. Since 2019, S. Simon started 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.

## 8.3 Development of specific numerical schemes

### 8.3.1 Low-Mach-number schemes

**Participants:** Pascal Bruel, Jonathan Jung, Vincent Perrier, Ibtissem Lannabi.

**External collaborators:** E. Dick (Ghent University), Y. Moguen (SIAME, UPPA), S. Dellacherie (Hydro-Québec), P. Omnes (CEA, LAGA).

In [77], the last developments of a pressure-correction algorithm for compressible fluid flow regimes were presented. It is well-suited to simulate flows at all levels of Mach number with smooth and discontinuous flow parameter changes, by providing a precise representation of convective transport and acoustic propagation. A co-located finite volume space discretization was used with the AUSM flux splitting. It was demonstrated that two ingredients are essential for obtaining good-quality solutions: the presence of an inertia term in the face velocity expression and a velocity difference diffusive term in the face pressure expression, with a correct Mach number scaling to recover the hydrodynamic and acoustic low Mach number limits. To meet these two requirements, a new flux scheme, named MIAU, for Momentum Interpolation with Advection Upstream splitting was proposed.

In [11], a numerical scheme and a low Mach number fix for a system with a non-conservative source term due to porosity variation was proposed and tested. The problem was understood on the linear wave equation with porosity and based on the linear study, a scheme for the nonlinear and non-conservative case has been proposed.

In [46], the behaviour of acoustic waves in low Mach number flows was investigated. It was found that classical low Mach number fixes fail at propagating acoustic waves at high order: either the scheme damps acoustic waves because it is not asymptotically consistent with low Mach number acoustics, or a loss of convergence order is observed for DG schemes at second order. A fix was proposed, which allows to recover the optimal order of convergence for low Mach number acoustics.

Unfortunately, the only fix we found in [46] 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 [14], 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, which we did in an article currently in review. From an inaccurate solution, an accurate solution can be obtained by solving a Laplace equation, see [26, Part II, Sec. 2.4]. Last, we proved that a known result on finite volume schemes, namely the accuracy of the Roe scheme on triangular meshes [59] holds also for discontinuous Galerkin methods [26, Part II, Sec. 2.5].

### 8.3.2 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 [12].

#### 8.3.3 Multi-phase flows

**Participants:** Vincent Perrier.

As far as multi-phase models are concerned, based on the ideas of [52], we have revisited the derivation of Baer-and-Nunziato models [38]. 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 [83] or by ensuring mathematical properties [48]. In [15], 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 phasic entropies. Under some hypothesis 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 [83, 56], which does not necessarily ensure the same phasic entropy dissipation properties.

### 8.4 Analysis and simulation of turbulent flows and heat transfer

#### 8.4.1 Modelling of solar receivers

**Participants:** Rémi Manceau.

**External collaborators:** S. Serra (LaTEP, UPPA), E. Franquet (formerly LaTEP, UPPA, now Univ. Côte d'Azur).

In collaboration with the LaTEP laboratory of Pau, we have extended the validation of the EB-RSM model to flows encountered in the solar receivers of concentrated solar power plants with very large temperature differences between two walls, such that variations of the molecular viscosity can lead to relaminarization [85]. This work, which demonstrated the ability of the EB-RSM model to reproduce these effects [44], allowed a broad parametric study to determine physical criteria to guide the design of future solar receivers [16].

#### 8.4.2 Effusion cooling

**Participants:** Rémi Manceau, Pascal Bruel.

**External collaborators:** Ph. Reulet (ONERA), E. Laroche (ONERA), D. Donjat (ONERA), F. Mastrippolito (formerly CAGIRE, now GDTech France).

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 [80].

#### 8.4.3 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 [39], the interaction of an atmospheric boundary layer with aerial reservoir(s) [82] or of a channel flow with cylinders in tandem [58] to the simulation of a blast wave of Sedov-like type [61]. 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.

#### 8.4.4 Thermocline energy storage

**Participants:** Rémi Manceau, Alexis Ferré.

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

Finally, 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.

# 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.

## 9.1 Bilateral contracts with industry

- Dassault Aviation: "Improvement of the turbulence models", contract associated to the PhD thesis of Gustave Sporschill.
- 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".

## 9.2 Bilateral grants with industry

- Dassault Aviation (Cifre PhD grant): "Improvement of the turbulence models. Application to the prediction of aerodynamic flows", PhD student Gustave Sporschill.
- 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.

# 10 Partnerships and cooperations

## 10.1 European initiatives

### 10.1.1 FP7 & H2020 projects

**Participants:** Pascal Bruel, Rémi Manceau.

- Topic: MG-1.2-2015 - Enhancing resource efficiency of aviation
- Project acronym: SOPRANO
- Project title: Soot Processes and Radiation in Aeronautical inNOvative combustors
- Duration: 01/09/2016 - 28/02/2021
- Coordinator: SAFRAN
- Other partners:

  - France: CNRS, CERFACS, INSA Rouen, SAFRAN SA, Snecma SAS, Turbomeca SA.
  - Germany: DLR, GE-DE Gmbh, KIT, MTU, RRD,
  - Italy: GE AVIO SRL, University of Florence
  - United Kingdom: Rolls Royce PLC, Imperial College of Science, Technology and Medecine, Loughborough University.

- Abstract: For decades, most of the aviation research activities have been focused on the reduction of noise and NOx and CO2 emissions. However, emissions from aircraft gas turbine engines of non-volatile PM, consisting primarily of soot particles, are of international concern today. Despite the lack of knowledge toward soot formation processes and characterization in terms of mass and size, engine manufacturers have now to deal with both gas and particles emissions. Furthermore, the understanding of heat transfer, that is also influenced by soot radiation, is an important matter for the improvement of the combustor's durability, as the key point when dealing with low-emissions combustor architectures is to adjust the air flow split between the injection system and the combustor's walls. The SOPRANO initiative consequently aims at providing new elements of knowledge, analysis and improved design tools, opening the way to:
  - Alternative designs of combustion systems for future aircrafts that will enter into service after 2025 capable of simultaneously reducing gaseous pollutants and particles,
  - Improved liner lifetime assessment methods. Therefore, the SOPRANO project will deliver more accurate experimental and numerical methodologies for predicting the soot emissions in academic or semi-technical combustion systems. This will contribute to enhance the understanding of soot particles formation and their impact on heat transfer through radiation. In parallel, the durability of cooling liner materials, related to the wall air flow rate, will be addressed by heat transfer measurements and predictions. Finally, the expected contribution of SOPRANO is to apply these developments in order to determine the main promising concepts, in the framework of current low-NOx technologies, able to control the emitted soot particles in terms of mass and size over a large range of operating conditions without compromising combustor's liner durability and performance toward NOx emissions.
- In the SOPRANO project, our objective is to complement the experimental (ONERA) and LES (CERFACS) work by RANS computations of the flow around a multiperforated plate. Franck Mastrippolito, the post-doc recruited from mid-january 2019 to mid-january 2020, performed simulations aimed at reproducing the experiment of ONERA Toulouse carried out in the same workpackage. The configuration is that of an effusion plate with a gyration angle of 90 degrees and the turbulence model is EBRSM.

## 10.2 National initiatives

### 10.2.1 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 model based on improved RANS models, will be 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 Regional initiatives

### 10.3.1 SEIGLE

**Participants:** Jonathan Jung, Vincent Perrier.

SEIGLE means "Simulation et Expérimentation pour l'Interaction de Gouttes Liquides avec un Ecoulement fortement compressible". It is a 3-year program which started in October 2017 and is funded by Région Nouvelle-Aquitaine, ISAE-ENSMA, CESTA and Inria. The interest of understanding aerodynamic mechanisms and liquid drop atomization is explained by the field of applications where they play a key role, specially in the new propulsion technologies through detonation in the aerospace as well as in the securities field. The SEIGLE project was articulated around a triptych experimentation, modeling and simulation. An experimental database will be constituted. It will rely on a newly installed facility (Pprime), similar to a supersonic gust wind tunnel/ hypersonic from a gaseous detonation tube at high pressure. This will allow to test modeling approaches (Pprime / CEA) and numerical simulation (Inria / CEA) with high order schemes for multiphasic compressible flows, suitable for processing shock waves in two-phase media.

### 10.3.2 HPC scalable ecosystem

**Participants:** Jonathan Jung, Vincent Perrier, Sangeeth Simon.

HPC scalable ecosystem is 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 [55] 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.

### 10.3.3 ASTURIES

**Participants:** Rémi Manceau, Vincent Perrier, Jonathan Jung, Pascal Bruel, Anthony Bosco, Mahitosh Mehta, Romaric Simo Tamou.

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.

# 11 Dissemination

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

## 11.1 Promoting scientific activities

**Member of the conference program committees**

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

### 11.1.1 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**

- Int. J. Heat Fluid Flow (2) [Rémi Manceau]
- Phys. Fluids [Rémi Manceau]
- Flow, Turbulence and Combustion [Rémi Manceau]
- Nuclear Enginering and Design [Rémi Manceau]
- Continuum Mechanics and Thermodynamics [Jonathan Jung]
- Aerospace Science and Technology (2) [Pascal Bruel]
- Revista Facultad de Ingenieria de la Universidad de Antoquia (2) [Pascal Bruel]

### 11.1.2 Invited talks

[23] R. Manceau and P. Bikkanahally. 'Hybrid Temporal LES: from theory to applications'. In: HiFiLeD - 2nd High Fidelity Industrial LES/DNS Symposium. Toulouse / Virtual, France, 22nd Sept. 2021.

[24] V. Perrier. 'Stochastic derivation of Baer-and-Nunziato models: homogenization of two-phase hyperbolic terms and discussions on other cases'. In: Third workshop on compressible multiphase flows Strasbourg. Strasbourg, France, 21st June 2021.

[21] J. Jung. 'Méthode de volumes finis pour la mécanique des fluides compressibles et problèmes de précision à bas nombre de Mach'. In: Journées d'inauguration de la fédération MARGAUx. La Rochelle, France, 28th June 2021.

### 11.1.3 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.

### 11.1.4 Research administration

- Co-responsible for the organisation of the LMAP seminar of Mathematics and their Applications [Jonathan Jung].
- Member of the LMAP council [Jonathan Jung, Pascal Bruel].
- Member of the IPRA research federation scientific 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]. [8]
- Member of the CT3-Num committee of Pau University, in charge of managing the computing resources and projects at Pau University [Vincent Perrier].

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

## 11.2 Teaching - Supervision - Juries

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

### 11.2.1 Responsabilities in teaching

- In charge of the L2 of the Mathematics-Computer Science BSc [Jonathan Jung]
- In charge of the L2 of the cursus Master in Engineering program Mathematics and Computer Science [Jonathan Jung]
- In charge of the L3 of the cursus Master in Engineering program Mathematics and Computer Science [Jonathan Jung]
- In charge of the M1 of the cursus Master in Engineering program Mathematics and Computer Science [Jonathan Jung]
- In charge of the M2 of the cursus Master in Engineering program Mathematics and Computer Science [Jonathan Jung]

### 11.2.2 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 [29].
- E3 [R. Manceau]: Advanced physics–Turbulence modelling for CFD, 16h/year, ENSGTI, France [31].

### 11.2.3 Supervision

- PhD defended in 2021: Gustave Sporschill, "Improved Reynolds-Stress Modeling for Adverse-Pressure-Gradient Turbulent Boundary Layers in Industrial Aeronautical Flow", Dassault Aviation, Rémi Manceau.
- 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.

#### 11.2.4 Juries

- Referee: Mohamed Yacine Ben Ali, PhD thesis, University Rennes 1 [Rémi Manceau]
- Referee: Yann Marchenay, PhD thesis, University of Toulouse [Rémi Manceau]
- President of the jury: Martin Thomas, PhD thesis, University of Toulouse [Pascal Bruel]
- Referee: Emmanuel Laroche, HDR, University of Toulouse [Pascal Bruel]
- Member: Mariovane Donini, PhD thesis, INPE (Brazil) [Pascal Bruel]

### 11.3 Popularization

#### 11.3.1 Interventions

Vincent Perrier animated a debate around the release of the movie "Adventures Of A Mathematician".

## 12 Scientific production

### 12.1 Major publications

[1] 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.

[2] 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.

[3] 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.

[4] 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.

[5] 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.

[6] 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.

[7] 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.

[8] 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.

[9] 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

[10] 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 (9th Nov. 2021), p. 115109. DOI: 10.1063/5.0064009. URL: https://hal-univ-pau.archives-ouvertes.fr/hal-03423068.

[11] S. Dellacherie, J. Jung and P. Omnes. 'Construction of a low Mach finite volume scheme for the isentropic Euler system with porosity'. In: *ESAIM: Mathematical Modelling and Numerical Analysis* 55.3 (8th June 2021), pp. 1199–1237. DOI: 10.1051/m2an/2021016. URL: https://hal.archives-ouvertes.fr/hal-03197310.

[12] 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.

[13] 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.

[14] J. Jung and V. Perrier. 'Long time behavior of finite volume discretization of symmetrizable linear hyperbolic systems'. In: *IMA Journal of Numerical Analysis* (23rd Dec. 2021). DOI: 10.1093/imanum/drab092. URL: https://hal.archives-ouvertes.fr/hal-03503847.

[15] 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.

[16] S. Serra, E. Franquet, V. Boutrouche and R. Manceau. 'Asymmetric reverse transition phenomenon in internal turbulent channel flows due to temperature gradients'. In: *International Journal of Thermal Sciences* 159.106463 (2021). DOI: 10.1016/j.ijthermalsci.2020.106463. URL: https://hal.archives-ouvertes.fr/hal-02923557.

#### International peer-reviewed conferences


[17] 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, 15th Sept. 2021. URL: https://hal.inria.fr/hal-03366796.

[18] 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, 12th Mar. 2021. URL: https://hal.inria.fr/hal-03207390.


#### National peer-reviewed Conferences


[19] J. P. Saldía, S. Elaskar, L. F. Gutiérrez Marcantoni and P. Bruel. ‘Implementation and validation of a second-moment RANS turbulence model in OpenFOAM ®’. In: MECOM 2021 - 37° Congreso Argentino de Mecánica Computacional. Resistencia, Argentina, 1st Nov. 2021. URL: https://hal.inria.fr/hal-03536915.


#### Conferences without proceedings


[20] 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, 2021. URL: https://hal.inria.fr/hal-03114394.

[21] J. Jung. ‘Méthode de volumes finis pour la mécanique des fluides compressibles et problèmes de précision à bas nombre de Mach’. In: Journées d’inauguration de la fédération MARGAUx. La Rochelle, France, 28th June 2021. URL: https://hal.inria.fr/hal-03507227.

[22] J. Jung and V. Perrier. ‘Une méthode de filtrage pour les écoulements à faible nombre de Mach’. In: 10ème biennale française des mathématiques appliquées et industrielles. La Grande Motte, France, 21st June 2021. URL: https://hal.inria.fr/hal-03275216.

[23] R. Manceau and P. Bikkanahally. ‘Hybrid Temporal LES: from theory to applications’. In: HiFiLeD - 2nd High Fidelity Industrial LES/DNS Symposium. Toulouse / Virtual, France, 22nd Sept. 2021. URL: https://hal.inria.fr/hal-03366801.

[24] V. Perrier. ‘Stochastic derivation of Baer-and-Nunziato models: homogenization of two-phase hyperbolic terms and discussions on other cases’. In: Third workshop on compressible multiphase flows Strasbourg. Strasbourg, France, 21st June 2021. URL: https://hal.inria.fr/hal-03275203.

[25] 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, 11th Jan. 2021. URL: https://hal-univ-pau.archives-ouvertes.fr/hal-03154441.


#### Doctoral dissertations and habilitation theses


[26] V. Perrier. ‘Contributions to the high order approximation of compressible flows: multiphase flows, low Mach number flows’. Université de Pau et des Pays de l’Adour (UPPA), Pau, FRA., 29th June 2021. URL: https://hal.archives-ouvertes.fr/tel-03283701.

[27] G. Sporschill. ‘Improved Reynolds-Stress Modeling for Adverse-Pressure-Gradient Turbulent Boundary Layers in Industrial Aeronautical Flow’. Université de Pau et des Pays de l’Adour, 4th June 2021. URL: https://tel.archives-ouvertes.fr/tel-03440794.

## 12.3 Other

**Educational activities**

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

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

[30] R. Manceau. 'Modelisation de la turbulence pour la CFD'. Master. France, Oct. 2021. URL: https://hal.inria.fr/hal-03207437.

[31] R. Manceau. 'Turbulence modelling for CFD'. Master. France, Oct. 2021. URL: https://hal.inria.fr/hal-03207433.

## 12.4 Cited publications

[32] 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.

[33] 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.

[34] 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.

[35] F. Archambeau, N. Méchitoua and M. Sakiz. 'Code Saturne: A Finite Volume Code for the Computation of Turbulent Incompressible flows - Industrial Applications'. In: *Int. J. on Finite Volume, Electronical edition: http://averoes.math.univ-paris13.fr/html* ISSN 1634.0655 (2004).

[36] 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.

[37] 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.

[38] 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.

[39] 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.

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

[41] 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.

[42] 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.

[43] 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.

[44] V. Boutrouche, E. Franquet, S. Serra and R. Manceau. 'Influence of the turbulence model for channel flows with strong transverse temperature gradients'. In: *International Journal of Heat and Fluid Flow* 70 (Apr. 2018), pp. 79–103. DOI: 10.1016/j.ijheatfluidflow.2018.01.007. URL: https://hal.inria.fr/hal-01944199.

[45] 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.

[46] 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.

[47] 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.

[48] 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.

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

[50] 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.

[51] 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.

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

[53] 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.

[54] 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.

[55] 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.

[56] 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.

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

[58] 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.

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

[60] 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.

[61] 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.

[62] 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.

[63] 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.

[64] 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.

[65] 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.

[66] 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.

[67] 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.

[68] 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.

[69] 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.

[70] 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: hal-01092931.

[71] 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.

[72] 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.

[73] 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.

[74] 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.

[75] 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.

[76] M. Mays, S. Laizet and S. Lardeau. 'Performance of Various Turbulence Models for Simulating Sub-critical High-Reynolds Number Flows over a Smooth Cylinder'. In: *AIAA aviation 2021 forum*. 2021, p. 2762.

[77] Y. Moguen, P. Bruel and E. Dick. 'A combined momentum-interpolation and advection upstream splitting pressure-correction algorithm for simulation of convective and acoustic transport at all levels of Mach number'. In: *Journal of Computational Physics* 384 (May 2019), pp. 16–41. DOI: 10.1016/j.jcp.2019.01.029. URL: https://hal.inria.fr/hal-02064848.

[78] 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.

[79] 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.

[80] 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.

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

[82] 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.

[83] 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.

[84] 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.

[85] S. Serra, A. Toutant and F. Bataille. 'Thermal large eddy simulation in a very simplified geometry of a solar receiver'. In: *Heat Transf. Eng.* 33.6 (2012), pp. 505–524.

[86] 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.

[87] 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.