



JOB OFFER – PhD
Development of a reactive supersonic wall-law for reusable rocket engine

OFFER INFORMATION

Reference: AAM-2026-DAV-02

Location: 42 Avenue Gaspard Coriolis – 31057 Toulouse

Team: AAM

Contact person:

- Guillaume Daviller, guillaume.daviller@cerfacs.fr
- Franck Nicoud, franck.nicoud@umontpellier.fr
- Simon Blanchard, simon.blanchard@cnes.fr

Period: 3 years – from october 2026

Salary: CNES PhD

WARNING: Applications will be accepted from February 2 to March 13, 2026, from French and international students via the CNES website (<https://recrutement.cnes.fr/fr/annonces>).

Key words: CFD, combustion, compressible flow, wall law

CERFACS

Cerfacs is a private research, development, transfer and training center for modeling, simulation and high-performance computing. Cerfacs designs, develops and proposes innovative software methods and solutions to meet the needs of its partners in the aeronautics, space, climate, environment and energy sectors. Cerfacs trains students, researchers and engineers in simulation and high-performance computing.

Cerfacs works closely with its seven partners: [Airbus](#), [Cnes](#), [EDF](#), [Météo France](#), [Onera](#), [Safran](#) et [TotalEnergies](#).



HOSTING TEAM - AAM

The Advanced Aerodynamic & Multiphysics (AAM) team is dedicated to developing cutting-edge numerical methods, physical modeling, and High-Performance Computing (HPC) techniques for new Computational Fluid Dynamics (CFD) solvers. The work focuses on fluid dynamics simulations for aircraft, rockets, and turbomachinery, in close collaboration with Cerfacs partners.

CONTEXT

Improving the design of reusable rocket engines is a significant challenge in today's competitive environment for launching lowcost payloads. The jet flows from multiple engine bays in a reusable launcher interact in complex ways, creating intricate flow phenomena, especially within recirculation zones, turbulent mixing, and intense thermochemical gradients, particularly in boundary layers. Nevertheless, predicting wall heat flux within the framework of Large Eddy Simulation remains prohibitively expensive and presents considerable technical challenges. An accurate modeling of the boundary layer is therefore essential for predicting heat transfer in reactive flows that could compromise material integrity and flight stability. Those effects are challenging to model in the case of methane combustion, while this fuel will be massively used in various European engines, such as the Prometheus. Ultimately, more accurate predictions of nozzle thermal fluxes at the walls will allow for better estimation of an



engine's lifespan for reuse. This doctoral project aims to develop a multiphysics numerical approach for simulating a reactive boundary layer in the supersonic regime by coupling compressible Navier-Stokes equations with chemical kinetic models. The goal is to accurately capture the complex interactions between gas dynamics, chemical reactions, and thermal effects near the walls. In this context, CNES is collaborating closely with Cerfacs [1] as part of the Aérothermodynamique des Tuyères et Arrière-Corps (ATAC) research group.

MISSION

The primary objective of this doctoral project is to develop a high-fidelity wall boundary condition capable of modeling phenomena within a reactive boundary layer in a supersonic regime. This model will incorporate the interaction between air and burnt gases resulting from methane/oxygen combustion [2], robust discretization schemes to capture pronounced temperature gradients, and the coupling of chemistry, thermal effects, and compressible gas dynamics [3,4]. Special attention will be given to studying heat transfer mechanisms in the boundary layer, particularly in relation to rapid chemical reactions and aerodynamics. Initially, the model will be based on the assumption of thermochemical equilibrium, before integrating the effects of adverse pressure gradients. Depending on the results, several chemical approaches might be investigated. Finally, the study will focus on a configuration representative of the Prometheus engine, such as the interaction between hot supersonic jets from the gas generators and the nozzle's external wall. The wall model will be derived and validated using Direct Numerical Simulations, Wall-Resolved Large-Eddy Simulations, and, if available, with experimental data from supersonic wind tunnels or instrumented flight tests.

ORGANIZATION OF RESEARCH WORK (36 months)

Task T1. Literature Review: A bibliographic study will be carried out to assimilate existing results and tools on reactive boundary layers in supersonic regimes, wall models, and multiphysics numerical approaches. This will situate the project within the state of the art, identify scientific bottlenecks, and define priority research directions.

Task T2. Dimensional Analysis: A study of characteristic quantities (Mach number, Reynolds number, boundary layer thickness, chemical time scale, etc.) will be conducted to rank the dominant phenomena. This analysis provides the basis for simplifying or adapting models and ensures the consistency of the assumptions made.

Task T3. Supersonic Wall Model, Simulation & Validation: This task is dedicated to developing a wall boundary condition adapted to supersonic regimes. The model will be tested through numerical simulations and validated by comparison with reference data (DNS, wall-resolved LES, or experimental results, if available). The objective is to faithfully reproduce thermal and dynamic transfers near the walls.

Task T4. Incorporation of Chemical Reactions, Simulation & Validation: In a second phase, chemical kinetics (e.g., methane/oxygen) will be integrated into the model to better capture the effects of reactions on the boundary layer. Simulations will be compared with high-fidelity results to validate the relevance of chemical schemes and their impact on wall heat fluxes.

Task T5. Consideration of an Adverse Pressure Gradient, Simulation & Validation: The effects of an unfavorable pressure gradient on boundary layer stability and heat transfer will also be studied. Simulations will assess the behavior of the developed wall law. Validation will rely on representative configurations and reference data.

Task T6. Definition of a Representative Engine Configuration: This task, carried out in close collaboration with CNES, will enable the selection and description of a geometry and operating conditions representative of the Prometheus engine. This step will ensure the industrial relevance of the project and link theoretical developments to practical applications.



Task T7. Simulation & Analysis: Application of the developed model to the realistic configuration selected in Task T6. Results will be analyzed to characterize heat fluxes, gas–wall interactions, and the impact of chemical reactions.

Task T8. Dissemination, Publications, and Writing: Throughout this thesis, the results will be presented at international conferences and published in scientific journals, as well as shared with the ATAC group (approximately twice a year), which brings together French experts in nozzle flow for space applications. In addition, a significant amount of time will be devoted to writing the thesis manuscript.

The following schedule details the planning of these tasks.

PLANIFICATION

| | Année 1 | | | | Année 2 | | | | Année 3 | | | |
|----------|---------|----|----|----|---------|----|----|----|---------|----|----|----|
| | Q1 | Q2 | Q3 | Q4 | Q1 | Q2 | Q3 | Q4 | Q1 | Q2 | Q3 | Q4 |
| Tâche T1 | | | | | | | | | | | | |
| Tâche T2 | | | | | | | | | | | | |
| Tâche T3 | | | | | | | | | | | | |
| Tâche T4 | | | | | | | | | | | | |
| Tâche T5 | | | | | | | | | | | | |
| Tâche T6 | | | | | | | | | | | | |
| Tâche T7 | | | | | | | | | | | | |
| Tâche T8 | | | | | | | | | | | | |

FORMATIONS

Cerfacs is a high-level training center that offers numerous courses in these fields of expertise, certified QUALIOPI. Please feel free to consult the program of in-person courses at <https://cerfacs.fr/event/?category=formation> or online courses at <https://cerfacs.fr/online-training/>.

DESIRED PROFILE

The candidate, holding a Master’s degree and/or an engineering diploma, should ideally have completed training in combustion, compressible fluid mechanics, and turbulence. A thorough knowledge and/or project-based experience in unsteady computational fluid dynamics (CFD) is required. In addition, skills in signal processing would be appreciated, as well as some experience with Fortran, C++, and Python codes. Oral and written proficiency in English is mandatory. Please send a cover letter and a complete CV.

For this position, the required qualities are as follows:

- Initiative and self-motivation
- Ability to communicate within a project team (expressing requirements, presenting work, etc.)
- Ability to work independently with computational codes (use of development tools, compilers, programming languages, etc.)
- The candidate will be required to present their work in written and oral English, in accordance with the standards expected in an international research laboratory.



WHAT WE OFFER AT CERFACS

- Broad access to technology, a rich interpersonal environment, and in-house skills recognized nationally and internationally.
- An inclusive and equitable work environment.
- A structure accessible to people with disabilities.
- A complementary health insurance scheme offering excellent health care coverage in addition to social security, with the possibility of enrolling family members (spouse and children).
- 6 weeks' annual leave (with the possibility of 22 extra days' leave per year linked to your choice of a 39-hour rather than 35-hour working week).
- Flexible working arrangements, with the possibility of working from home up to two days a week.
- A sustainable mobility package enables employers to pay up to a maximum of 500 euros a year to cover the home-to-work travel costs of staff who cycle to work.

HOW TO APPLY?

To apply, please send your CV and cover letter to guillaume.daviller@cerfacs.fr, franck.nicoud@umontpellier.fr, and simon.blanchard@cnes.fr. Applications will be accepted from February 2 to March 13, 2026, from French and international students via the CNES website(<https://recrutement.cnes.fr/fr/annonces>).

See you soon at CERFACS!

REFERENCES

- [1] G. Daviller, J. Dombard, G. Staffelbach, J. Herpe & D. Saucereau. "Prediction of Flow Separation and Side-loads in Rocket Nozzle Using Large-eddy Simulation", *Int. J. Comp. Fluid Dyn.*, 2020.
- [2] S. Blanchard. "Multi-physics large-eddy simulation of methane oxy-combustion in liquid rocket engines", PhD Thesis – INP Toulouse, 2021.
- [3] O. Cabrit and F. Nicoud. "Direct simulations for wall modeling of multicomponent reacting compressible turbulent flows", *Phy. Fluids*, 2009.
- [4] M. Cizeron, N. Odier, F. Duchaine, L. Gicquel, and F. Nicoud. "Implementation of a TBLE-based wall model with pressure gradient in a massively parallel LES solver". *ASME Turbo Expo*, 2024.