Jérôme Dombard

2 Bis rue Emile Paul Heuillet, 31500 Toulouse, France.
Phone: +33 6 95 86 17 15 • jerome.dombard@gmail.com

Lead engineer in CFD - aerodynamics and aerothermal flows.

SUMMARY OF QUALIFICATIONS

• Seven-year experience in carrying out high-fidelity simulations for the design and method departments of SAFRAN companies (Safran Helicopter Engines and Safran Aircraft Engines) on High Performance Computing clusters with the appropriate research tools of the CERFACS, i.e applies fundamentals to solve practical engineering configurations.
• Solves multi-disciplinary, multi-physics transient problems including combustion (perfect and real gases), two-phase flows, conjugate heat transfer, transonic flows in nozzles and turbomachinery (radial compressors and high-pressure turbines) on real industrial configurations with all technological details (effusion cooling, purges etc..).
• Pre-sales: negotiates with the client, writes proposals and delivers the project on time with excellence (from the CAD simplification, mesh generation, data preparation, case set-up, result analysis to the final delivery).
• Implements new models into the code and writes pre/post processing tools if needed and compares with experiments.
• Works on proof-of-concept projects to demonstrate the ability of the code to solve new problems and to gain the confidence of new customers.
• Proven ability to work cooperatively on complex problems to comply with a tight schedule and manage multiple projects in parallel.
• Communicates effectively in a multicultural environment and learns new languages.

EDUCATION

Short Course: Radial compressor design and optimisation at the Von Karman Institut for Fluid Dynamics - Brussels, Belgium — 2018

PhD in Fluid Dynamics at IMFT (Institut de Mécanique des Fluides de Toulouse) - Toulouse, France — 2011
PhD advisors: **Thierry Poinsot** and Laurent Selle ; *Dissertation title*: Direct Numerical Simulation of non-isothermal dilute sprays using the Mesoscopic Eulerian Formalism.

MS in Applied Mathematics, National Polytechnic Institute (INP) Bordeaux - France. — 2008

Master of Engineering in Mathematical and Mechanical Modelling at (ENSEIRB-MATMECA-IPB) (one of the French "Grandes Ecoles"). Specialisation in Computational Fluid Dynamics. — 2008

RELEVANT EXPERIENCE

Lead Application Engineer in turbomachinery for SAFRAN at CERFACS - Toulouse, France — 2016 - present

• Performs Large-Eddy Simulations of centrifugal transonic compressors, analyses large-scale databases and compares them with experiments (Turbo AVBP code).
• Carries out high-fidelity integrated simulations (combustion chamber and high-pressure turbine) with all technological details (effusion cooling, purges etc...) with a special focus on the transport of hot streaks within the turbine and possible interaction with the secondary air flow path.
• Generates and interprets aerothermal results from numerical simulations.
• Provides training and best practices to support customers, mainly in meshing, CAD and turbomachinery.
• Evaluated the capacity of several numerical codes in predicting the performance of a HP turbine at off design operation (focus on the aerodynamic design implications).
• Supervises a project that aims at the LES of a whole aeronautic engine (from the fan to the high-pressure turbine).

Application Engineer in two-phase flow combustion and hypersonic flows for SAFRAN at CERFACS - Toulouse, France — 2013 - 2015

- Carried out Large-Eddy Simulations of industrial combustors including two-phase flows to compare Eulerian and Lagrangian formalisms.
- Performed unsteady coupled convection, conduction and radiation Large-Eddy Simulations of an industrial burner and compared them with a lower order model tool and experiment.
- Developed a methodology to find the extinction limit in lean-combustion burners.
- Simulated coaxial injector-type gas-generators with LES (real gas combustion) and used a Helmholtz solver to study its acoustic modes.
- Performed LES of highly-separated and transonic flow in a Truncated Ideal Contour nozzle to compute non-axisymmetric side loads and compare it with experiments.

Postdoctoral scholar, Center for Turbulence Research, Stanford University - Stanford, CA, USA — 2012

- Performed non-reactive Large-Eddy Simulations in a SAFRAN gas turbine (AVBP code) within the context of uncertainty quantification.
- Generated unstructured meshes in complex geometries (Centaur Soft mesh generator).
- Wrote scripts to post-process the results.
- Animated a five-member team during the summer program (one-month workshop at CTR, Stanford).

PhD student at IMFT , Toulouse -France — 2008 - 2011

- Performed Direct Numerical Simulations (DNS) of two-phase flows in an Euler/Euler framework.
- Implemented a transport equation in a parallel code (AVBP) to take into account the thermal inertia of particles.
- Implemented a new methodology to evaluate the numerical and physical dissipation in a two-phase flow simulation.
- Collaborated with a team from CORIA laboratory (France) to implement a turbulence-injection boundary condition in their in-house code.

Intern at CIEMAT (Centro de Investigaciones Energéticas, Medioambientales y Tecnologicas) - Madrid, Spain — Summer 2008

- Performed DNS of an academic reacting flow (bluff-body flame).
- Conducted *a priori* tests for subgrid-scale turbulent combustion models.
- Implemented FPI tables in the AVBP solver.

Gap year at **Airbus** (Methods and tools department) - Madrid, Spain — 2006/10 - 2007/07

- Performed Detached-Eddy Simulations of an airfoil with an iced profile with the TAU code of the DLR.
- Generated unstructured meshes (Centaur soft mesh generator).

Intern at **Fluid Gravity Engineering** - Portsmouth, England — Summer 2006

- Performed a Direct Numerical Simulation of non-reacting hypersonic flows for a re-entry vehicle.
- Implemented a temperature slip boundary condition.

AWARD

Leopold Escande Prize (2011), Scientific Council of the National Polytechnic Institute of Toulouse (awarded to the ten best PhDs of Toulouse university)

SKILLS

Strong knowledge in CFD, especially in combustion and aerodynamics in turbomachines (centrifugal compressors and high-pressure turbines).
UNIX skills, script programming and knowledge of programming languages such as Python and Fortran 90. Notions of C++.
Proficient in CAD (Spaceclaim), meshing (Centaur soft) and post treatment tools (Paraview and Ensight).
Thrives in a challenging, fast-paced environment.
Result-oriented, self-motivated and autonomous.
Good interpersonal, leadership and communication skills.
Languages: French (mother tongue), English (advanced), Spanish (advanced), Portuguese (elementary).

PUBLICATIONS

C. Pérez-Arroyo, J. Dombard, F. Duchaine, L. Gicquel, N. Odier, G. Exilard, S. Richard, N. Buffaz, J. Démolis. Large-Eddy Simulation of an integrated high-pressure compressor and combustion chamber of a typical turbine engine architecture. GT2020-16288 *Submitted to* ASME Turbo Expo 2020.

M. Harnieh, F. Duchaine, L. Gicquel, N. Odier, J. Dombard. Loss predictions in the high-pressure film-cooled turbine vane of the Factor project by mean of wall-modeled Large Eddy Simulation. GT2020-14232 *Submitted to* ASME Turbo Expo 2020.

J. Dombard, F. Duchaine, L. Gicquel, N. Odier, K. Leroy, N. Buffaz, S. Le Guyader, J. Démolis, S. Richard, T. Grosnickel. Evaluation of the capacity of RANS/URANS/LES in predicting the aerodynamic performance of a state-of-the-art high-pressure turbine - effect of load and off-design operation. GT2020-15447 *Submitted to* ASME Turbo Expo 2020.

S. Agarwal, F. Duchaine, L. Gicquel, N. Odier, J. Dombard. Analysis of the unsteady flow field inside a fan-shaped cooling hole predicted by Large-Eddy Simulation. GT2020-14201 *Submitted to* ASME Turbo Expo 2020.

B. Martin, F. Duchaine, L. Gicquel, N. Odier, J. Dombard. Wall-resolved Large-Eddy Simulation of the LS89 cascade using an explicit local time-stepping method. GT2020-14171 *Submitted to* ASME Turbo Expo 2020.

A. Perrot, L. Gicquel, F. Duchaine, N. Odier, J. Dombard, T. Grosnickel. Unsteady Analysis of heat transfer coefficient distribution in a static ribbed channel for an established flow. GT2020-14493 *Submitted to* ASME Turbo Expo 2020.

G. Daviller, G. Staffelbach, J. Dombard, Julien Herpe and D. Saucereau. Prediction of Flow Separation and Side-Loads in Rocket Nozzle Using Large-Eddy Simulation. *Submitted to* International Journal of Computational Fluid Dynamics.

M. Thomas, J. Dombard, F. Duchaine, L. Gicquel, C. Koupper. Large-Eddy Simulation of Combustor and Complete Single-Stage High-Pressure Turbine of the Factor Test Rig. GT2019-91206. In ASME Turbo Expo 2019, Phoenix, USA.

B. Martin, M. Thomas, J. Dombard, F. Duchaine, L. Gicquel. Analysis of solid particle ingestion and dynamics in a turbomachine using Large-Eddy Simulation. GT2019-91215. In ASME Turbo Expo 2019, Phoenix, USA.

M. Harnieh , M. Thomas, R. Bizzari, L. Gicquel, F. Duchaine and J. Dombard. Assessment of a coolant injection model on cooled high-pressure vanes with LES. *accepted in* Flow, Turbulence and Combustion.

V. Brunet, E. Croner, A. Minot, J. de Laborderie, E. Lippinois, S. Richard, J-F Boussuge, J. Dombard, F. Duchaine, L. Gicquel, T. Poinsot, G. Puigt, G. Staffelbach, L. Segui, O. Vermorel, N. Villedieu, J-S Cagnone, K. Hillewaert, Michel, G. Lartigue and V. Moureau. Comparison of Various CFD Codes for LES Simulations of Turbomachinery: From Inviscid Vortex Convection to Multi-Stage Compressor. GT2018-75523. In ASME Turbo Expo 2018, Oslo, Norway.

J. Dombard, F. Duchaine, L. Gicquel, G. Staffelbach, N. Buffaz and I. Trebinjac. Large-Eddy Simulations in a Transonic Centrifugal Compressor. GT2018-77023. In ASME Turbo Expo 2018, Oslo, Norway.

F. Duchaine, J. Dombard, L.Y.M. Gicquel and C. Koupper. Integrated Large-Eddy Simulation of combustor and turbine interactions: effect of turbine stage inlet conditions. GT2017-63473. In ASME Turbo Expo 2017, Charlotte, NC, USA.

F. Duchaine, J. Dombard, L.Y.M Gicquel and C. Koupper. On the importance of inlet boundary conditions for aerothermal predictions of turbine stages with Large-Eddy Simulation. Computers and Fluids. Vol. 154-1. pp 60-73, 2017.

J. Dombard and G. Iaccarino. Sensitivity analysis to the normal grid-resolution in a turbulent channel flow using large-eddy simulations. In CTR, editor, *Annual Research Briefs 2012*, 2012.

J. Dombard, T. Poinsot, V. Moureau, N. Savary, G. Staffelbach and V. Bodoc. Experimental and numerical study of the influence of small geometrical modifications on the dynamics of swirling flows. In CTR, editor, *Proceedings of the Summer Program 2012*, 2012.

J. Dombard, B. Leveugle, L. Selle, J. Reveillon, T. Poinsot and Y. D'Angelo. Modeling heat transfer in diluted two-phase flows using the Mesoscopic Eulerian Formalism. *International Journal of Heat and Mass Transfer*, 2011.
DOI: 10.1016/j.ijheatmasstransfer.2011.10.050.