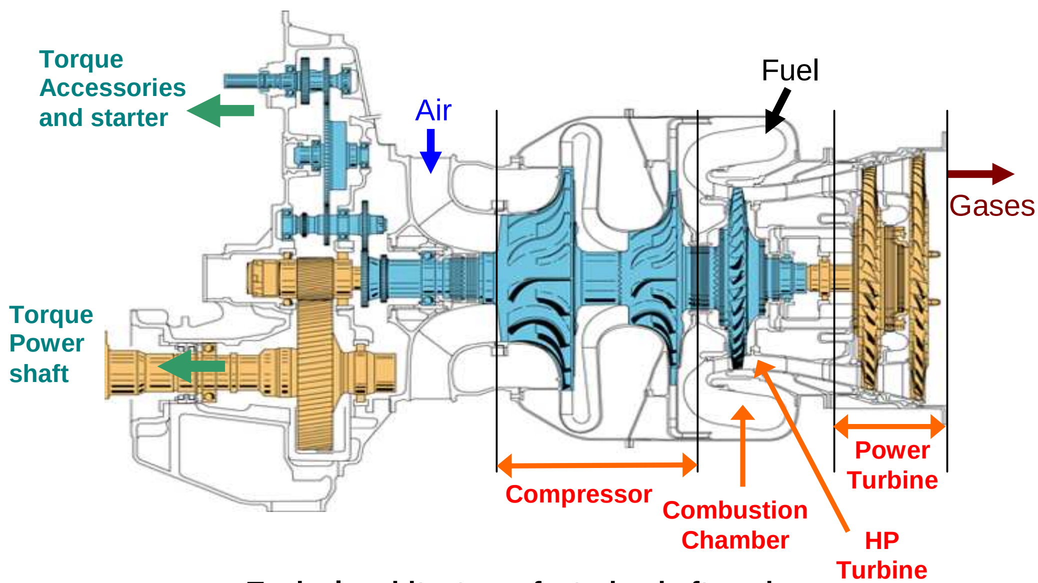
The technical challenge for green affordable turboshaft engine

Eric Seinturier
Turbomeca 64511 Bordes cedex)
Eric.seinturier@turbomeca.fr

Introduction

Because powering an aircraft has always been a difficult challenge, the aeronautic industry is constantly looking for new technologies and concepts. Since many years, numerical simulation has been an efficient tool to improve the aircraft performances.

With the environmental concerns and oil resources decrease, the performance improvement is more than ever the backbone of aeronautic research. Indeed, within a short 5-year term, the aeronautic industry must demonstrate a fuel-consumption cut by 20% and emissions reduced by 50%. For the longer term, brainstorming is already launched to replace the fossil fuels by greener alternatives and to prospect new propulsion concepts. Of course the engine manufacturers are key contributors to these objectives.



Typical architecture of a turboshaft engine

Turbomeca is developing greener turboshaft engines according to 3 main guidelines:

- **Efficiency**: Improve the engine efficiency to reduce the fuel consumption. This can be achieved by optimising the thermodynamic cycle, that is to say by increasing the overall pressure ratio. This requires more heavily loaded compressors and turbines. These rotating components must be reliable and safe so their design is highly constrained.
- **Mass**: Downsize the engine by increasing temperatures: the power of an engine is directly connected to its mass flow and its turbine inlet temperature (TIT). For one given power objective, a higher TIT allows to reduce the mass flow rate through the engine, that is to say its size and indeed, its mass.

The problem is to guaranty the life of the hot structures, mainly the high pressure turbine, just downstream the combustion chamber. The mass optimisation of all the engine components is performed by the way of simulations.

- **Emissions**: Optimise the combustion chamber in order to obtain optimum conditions to limit NOx, COx and particle emissions. To achieve that, combustion chambers are using lean fuel-air mixtures but this is generating problems of flame stability, especially during transient phases.

Of course these improvements to reduce the environmental impact of propulsion systems have a significant effect on their design. Components are globally more loaded and advanced simulations are systematically required because:

- Design margins are optimised and must be assessed accurately,
- new physical phenomenon are appearing, mainly unsteady ones
- each geometric detail has a significant impact on the whole engine behaviour
- the interactions between phenomenon of different disciplines become significant

This generates multiphysics (more degrees of freedom per integration point), unsteady (time resolution) and multiscale (more integration points) simulations requiring higher computer performance. This paper illustrates these aspects on five chosen examples detailed in the following chapters:

- Aerodynamic simulation of compressors and turbines
- Combustion chamber steady and unsteady analyses
- Aerothermal simulation in turbines
- Structural design of turbine blade shedding

Aerodynamics in compressors and turbines

High pressure ratio compressors and turbines are more sensitive to technological details as gaps, geometric radii, and secondary flow interactions. Moreover, they are subjected to violent unsteady phenomena that can generate high vibration levels. The design of such components requires to model the full compressor structure (wide domain), including the technological details (small size). The three-dimensional viscous flow equations with turbulence modelling must be solved on these huge domains, divided in several millions of control volumes, ideally with unsteady approaches.
A first illustration is given on the following multistage viscous steady analysis of a whole compressor composed of 4 axial stages and 1 impeller. The model integrates more than 10 millions points and the calculation requires 10 CPU hours on a NEC SX8 parallel machine. The CFD code used is ElsA (developed by French Onera). Figure 1 illustrates the obtained results:

- the total pressure distribution along the compressor
- the compressor map (mass flow on X-axis, pressure on Y-axis, for 3 iso-speeds) where computed results are compared with experimental data

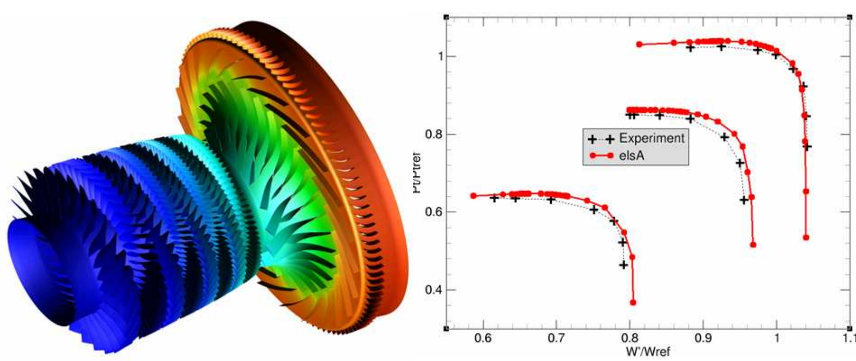


Figure 1 - total pressure (left), test-computation comparison (right)

These computations are now a standard since they provide good quality results with an affordable CPU cost. But results can be improved by accounting for small technologic details that can have a significant influence for some particular operating points. In order to enrich a model with these technologic effects, the chimera technique is used, allowing patching a basic mesh (coarse) with local chimera blocs (fine local mesh). It makes possible to mesh at low cost blade filets, tip gaps, secondary flows (bleed or reintroduction) or tip grooves and cooling holes for turbines as shown on the following Figure 2:

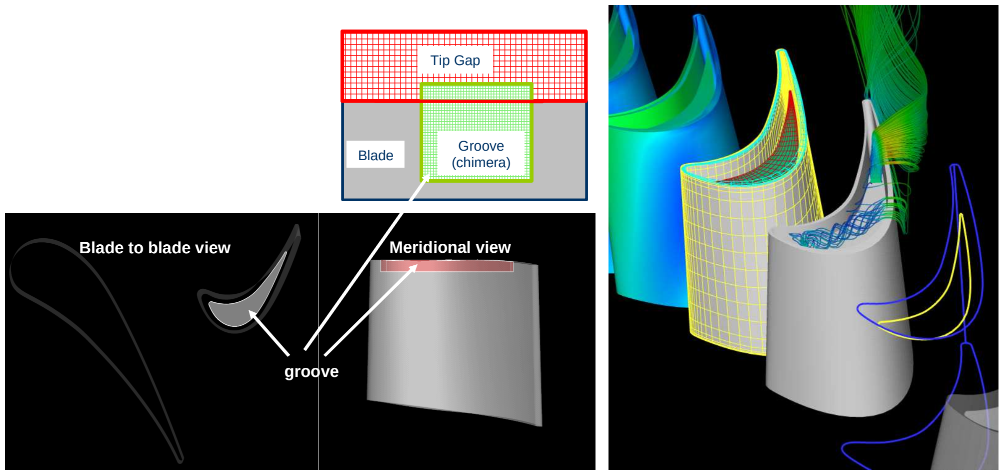


Figure 2 - tip groove modeling with chimera technique

The other progress axis for compressor concerns the unsteady simulation which remains expensive but can provide interesting results when the component is far from its design point. For instance, the points closed to the surge margin, when the compressor load is high (high pressure ratio for one given speed), are subjected to various unsteady phenomena. Moreover, unsteady simulations can be used to predict blade rows interaction (between the static blades and the rotor blades) and its effect on performance and vibrations.

Figure 3 shows the instantaneous entropy field (image of the wake of the stator) on a compressor row (stator + rotor). This 3D computation on one stator and one rotor blade row requires 50 to 100h on a NEC SX8 machine, depending on the number of iterations required for convergence. The computation of a complete multistage compressor remains very expensive to be acceptable in the design process.

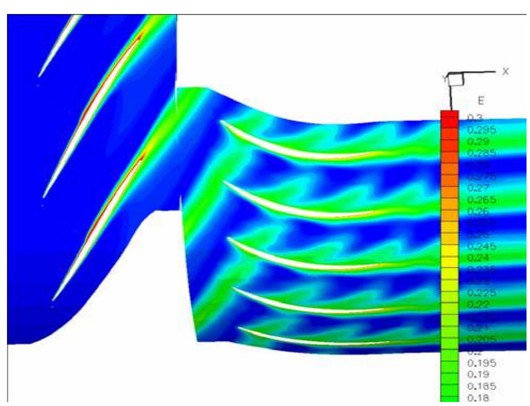


Figure 3 – 2D view Entropy instantaneous field

Combustion chamber

New combustion chamber designs must achieve a wide range of operating conditions and their modelling, especially at the limit of the flight domain or during transient phases, is a challenge for physical models as well as numerical methods.
First, the fuel spray formation must be understood, ideally with unsteady diphasic viscous analyses of the injector

- Then the aerothermal simulation must couple the main flow and the dilution flow, and then determine the flame behaviour and the temperature fields generated. Of course, the flame behaviour is strongly coupled to the temperature/pressure field in the combustion chamber.
- These temperature fields are then transported at the outlet of the combustion chamber and are used to determine the temperature fields on the downstream turbine components.
- Finally, the unsteady behaviour of the combustion chamber must be studied to determine if it remains lighted and to ensure that no unsteady phenomena appear on the engine operating range. Lean Low NOX combustor designs are much more sensible to these phenomena than conventional ones.

Steady aero-thermo-chemical analysis

As the pressure ratio and the inlet turbine temperature are increasing in order to reach more efficient thermodynamic cycles, thermal loads on the combustor are getting higher. In addition to this, lean Low NOx technologies impose to use more air for combustion and less for cooling than conventional designs. Thus the cooling of the combustor necessitates performing accurate studies and the use of advanced HPC simulations is becoming compulsory.

As an example, the advance simulation of a 3D effusion cooled combustor sector is presented on Figure 4. In order to finely handle the cooling issues of the combustor walls, all the effusion cooling holes have been included into the simulation resulting in a 13 million tetrahedral cells mesh. With this technique, it is possible to have a precise map of the thermal loads on the effusion cooled walls. This technique allows as well to better take into account the interaction between the film generated by effusion cooling holes and the main flow, and thus to predict more accurately the temperature map at the combustor exit, which is of first importance concerning the nozzle guide vane and turbine blades life.

The main characteristics of the example presented here are:

- Random Averaged Navier Stokes (RANS) model of the turbulence.
- two phase flow simulation
- physical models for flow mixing, spray spreading and evaporation, spray/flow interaction, combustion, turbulence/combustion interaction and wall convective fluxes.
- Up to 100 processors have been used in parallel for about 72 hours to perform this simulation on a 1/12 portion of the combustion chamber. This calculation was performed on the 8 200 cores French CEA-CCRT (Atomic Energy Commission – Research and Technology Computing Centre).

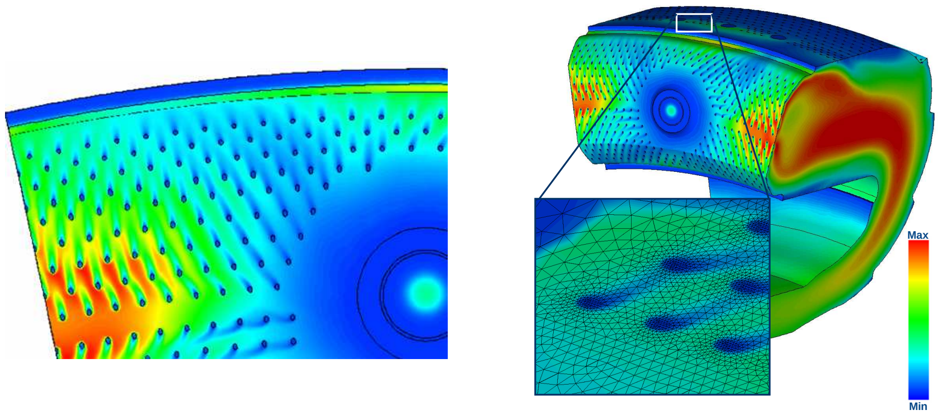


Figure 4 - Steady simulation of the combustion in a chamber – Temperature fields

This model RANS approach is not well suited to predict the very turbulent cross flow interactions between the cooling holes and the main stream. The perspective is to have more accurate results thanks to Large Eddy Simulation codes, as the one used in the following application but these techniques are still very expensive in term of CPU.

Combustion chamber unsteady simulation

The unsteady behaviour of the combustion chamber must be studied to determine if the flame is stable on the entire engine operating range. Recent studies have demonstrated the ability and accuracy of Large-Eddy Simulation (LES) to compute the reacting flow in an aeronautical combustion chamber [1, 2]. However, studies of industrial combustors are often limited to simplified geometries (e.g. with only one injector) to reduce computational cost. It has been shown [3-5] that this simplification neglects possible long-range azimuthal interactions between burners.

To assess the impact of azimuthal thermo-acoustic instabilities on an industrial turbine, a fully compressible reacting LES computation of a full helicopter combustion chamber has been performed. The combustor casing is included in the simulation. To limit the dependency of the results on the boundary conditions applied to the LES, all characteristic elements of the geometry are fully resolved. This configuration and the physical details taken into account (acoustics) make this simulation a world's first. The configuration consists of 15 identical sectors each containing one Low-Premixed Prevaporizer (LPP) injector as shown in Figure 5

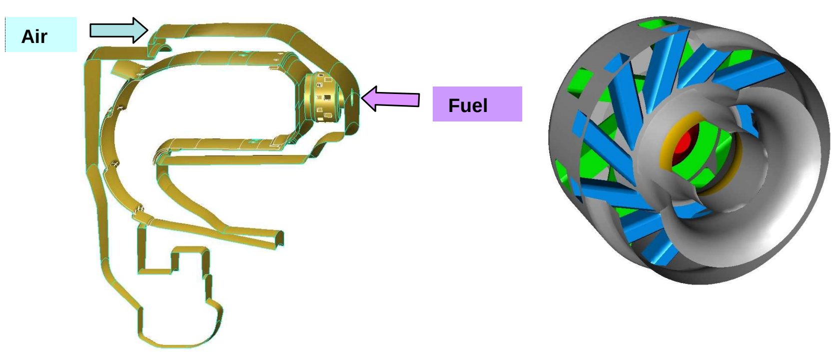
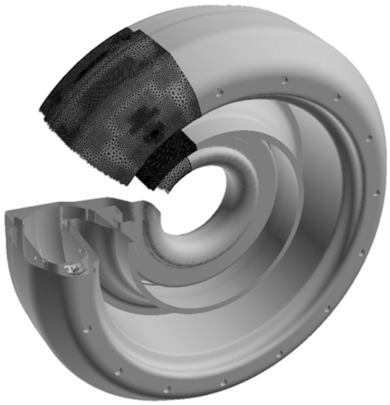


Figure 5 - Details of the Geometry: Combustor Sector (left) and swirler (right)

The full computational domain consists of more than 42 million tetrahedral cells and 8 million nodes as shown in Figure 6.



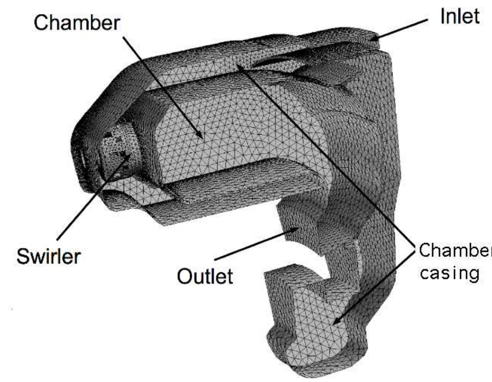


Figure 6 - Computational Mesh

The results presented are obtained with a third-order accurate scheme to integrate the LES equations, a Smagorinsky model for subgrid-scale turbulence, a dynamically thickened flame model with a simplified Kerosene chemistry scheme. The vaporization of JP10 kerosene is supposed to be instantaneous so that only gaseous flow is computed.

The simulations have been performed by the CFD team of CERFACS (Toulouse, France), with the code AVBP on the supercomputer MareNostrum located in Barcelona, Spain (10,240 CPUs running at 2.3GHz). More than 700,000 time steps have been computed at 5.4 sec/iteration on 256 processors. Over 200Gb of Data (2.5Gb per stored time step) have been recorded. 5 flow-through times can be computed in 10 days with 256 processors.

A temperature field superimposed on the full geometry is presented in Figure 7. It clearly shows the 3-D, non-axisymmetric character of the flow. Analysis of the pressure signal in the combustion chamber indicates the presence of a turning azimuthal acoustic mode at 600 Hz (Fig. 4), which is also present in the results of the industrial test-rig.

Figure 7 - Temperature Field in the combustor

Aerothermal simulations in turbines

The turbine is the component just downstream the combustion chamber, so it experiences very aggressive temperatures. The combustion gases are entering the turbine at a higher temperature than its material limit so it requires to cool down the structure with “cold air” (500℃) from the compressor.

The efficiency of greener engines is obtained by:

- a higher temperature in the combustion chamber that allows to downsize the engine for a given power
- a higher pressure ratio in order to optimise the thermodynamic cycle
- more efficient turbine components to reduce losses

As a consequence, turbine components will be subjected in the future to higher and higher temperatures. Advanced simulations are now required to design them, as illustrated here on the thermal simulation of a High Pressure Turbine Nozzle Guide Vane (HPT NGV).

The analysis to determine the temperature of the NGV must couple convection, conduction and stress analysis. Design models must account for the main flowpath (high temperature) and the secondary air system (blade cooling system). Indeed, high temperatures in the flow passage combined with internal cooling tend to increase the occurrence of local high temperature gradient generating thermo-mechanical fatigue.

Computational Fluid Dynamics is necessary to predict with accuracy local flow and heat-transfer phenomena. In the present example, a three-dimensional numerical model is set up with Fluent code. The purpose is first to analyse the aero-thermal features of NGV (hot flow passage, solid thermal conduction, internal cooling), and second, to perform a parametric analysis on different design aspects of internal cooling.

The mesh of the computational domain reaches approximately 20 millions cells. It is showed on Figure 8.

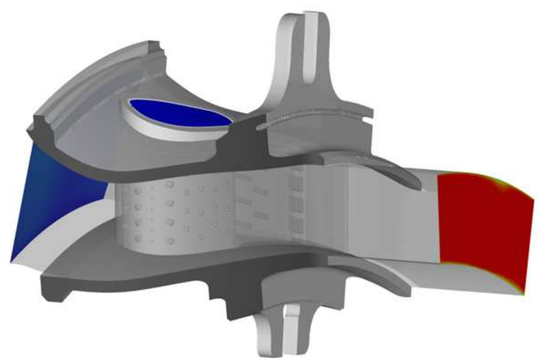


Figure 8 : Computational domain consisting of NGV flow passage and internal cooling

The simulation has required 72 hours on 12 processors has been required, on a modern 128 CPU (2 GHz) Linux cluster. Comparison with tests has shown a that the model is representative of NGV internal and external aero-thermal phenomena. It can be used to compare the effect of varying design parameters (cooling flow rate, repartition, inlet flow passage temperature boundary condition). Some results are showed on

Figure 9: the streamlines allow understanding the fluid behaviour inside and outside the NGV. This coupled convection / conduction gives finally the temperature in the structure which is a critical input for life assessment.

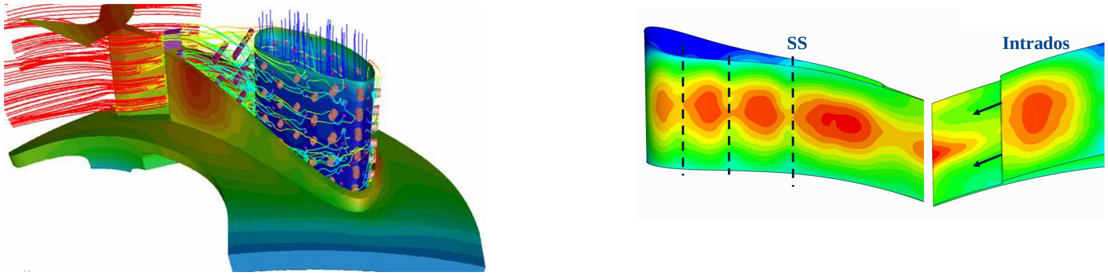


Figure 9 - streamlines (left) and temperature field on the blade surface

The trend for thermal analysis will consist in taking into account more geometric effects and more complex models for the laminar-turbulent transition which is a key driver of heat transfers.

Mass optimisation of turbine shields under impacts

The engine performance improvement and the mass reduction must be achieved guarantying the safety. International aviation regulatory bodies, such as the Federal Aviation Administration (USA) or the European Aviation Safety Agency, require that all engine debris that could occur must be contained [6].

Every rotating component is designed to resist to high load levels, covering a wide range of possible incidents. Nevertheless, in some very rare cases, a shaft overspeed could happen, generating tremendous loads on the disks. In order to avoid a disk failure (heavy debris difficult to contain) Turbomeca uses a fuse system that consist in loosing at a given speed all the blades of the power turbine. Without any blade, the disk can not accelerate anymore and is protected against burst: this is the blade shedding concept [7].

Of course, it must be demonstrated that all the blade debris are contained. This is the role of the turbine shield, which is a heavy component. Its mass optimisation is a great challenge and requires fast transient dynamics simulation. Indeed, these analyses allow reducing the number of demonstration tests which are very expensive because prototypes are significantly damage during these tests.

The phenomenon lasts only a few milliseconds but is so violent that explicit time integration is required. Moreover, during the blade shedding, significant transient loads are transmitted to the helicopter frame through the engine mounts.
As a consequence, the simulations must answer two problems:

- The prediction of the transient loads transmitted to the helicopter [8], which is driven by the rotor dynamics that defines the rotor-stator contact
- The shield optimisation to guaranty there is no perforation of the engine envelope [9] by the blade impacts.

These two problems are related temporally by the sequence of the blades release which is one of the most unknown. In order to identify this sequence, a transient computation of a free turbine is performed with the finite element explicit code, LS-Dyna. The EF model integrates the main parts of the turbine:

- The rotor of the turbine with disk,
- the 45 first stage turbine blades and 49 second stage turbine blades with their frangible section,
- the surrounding ring which defines the air path,
- the containment shield.

Figure 10 - Geometry: power turbine blade (left), power turbine module (right)

The mesh is composed of approximately 300 000 nodes and 200 000 elements.

The fragmentation modelling is then assumed by an effective plastic strain criterion, as used in the literature [11]. The resolution of dynamic equation is done thanks to the central difference scheme implemented in the finite element code.

The calculation is conducted in two steps:

- a relaxation phase which solves the centrifugal force field and allows to define the initial conditions of the second phase,
- a transient phase starting with the release of one blade

The simulation shows the domino effect subsequent to the first blade release. Analysis of the sequence obtained by integrating the whole turbine helps understanding the phenomenon and allows optimising the mass of the shields. A typical view of the result after few milliseconds is showed on Figure 11; The calculation has been performed with four processors running at 2Ghz. 80 hours were needed to simulate the first 20ms of the phenomenon.

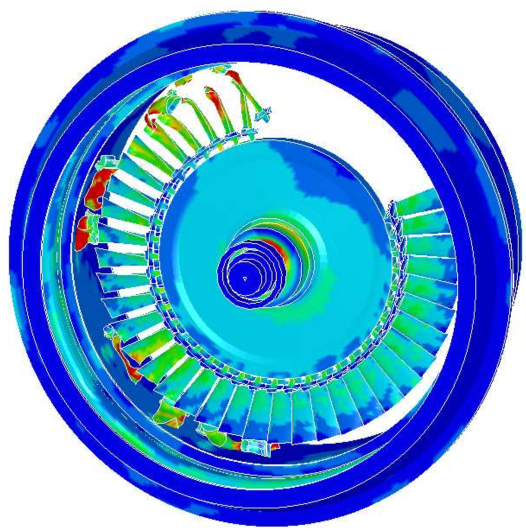


Figure 11 - Plastic deformation and release of the blades during a "Domino" sequence

Conclusions

The aeronautic community effort to reduce the environmental impact of air transport is rising new technologic challenges. Since the lead times to market and the development costs are key points for manufacturers, simulation has become the corner stone of technology prospect and development.

Various projects are ongoing around the world in France to improve the performance of simulation tools and hardware. Different initiatives are supported by European community to develop the CPU power available in Europe (http://www.hpc-europa.eu, http://www.prace-project.eu, etc ...). In France, HPC challenges are addressed in the system@tic competitiveness cluster (http://www.systematic-paris-region.org/fr/index.html) and aeronautic driven challenges in Aerospace valley cluster (http://www.aerospace-valley.com/), with MOSART project for instance

During the engine development process, models are getting enriched and the computational time is a key driver of the modelling level:

- Very simplified models for the iterative preliminary design with few hours of CPU acceptable at the maximum
- Intermediate models in the detailed design phase, with up to 24 hours of computational time are suitable
- Final assessment model, used for engine certification or physical understanding of unanticipated phenomenon with computational time between one week and one month.

To reduce computational times, codes are run on parallel machine but the scalability of simulation codes can vary much. Excellent results have been demonstrated for few thousands of processors for LES code AVBP developed by the CERFACS.

As a consequence, the new technologies required by the aeronautic are driving significant computational needs. The hardware is today able to answer to most of the CPU power need (even is more is always a benefit) but the main short term challenge is the efficiency of software's in massively parallel simulations.

References

[1] G. Boudier, L. Gicquel, T. Poinsot, D. Bissières and C. Bérat, *Comparison of LES, RANS and Experiment in an Aeronautical Gas Turbine Combustion Chamber*, In Proc. of the Combustion Institute, 31(2):3075-3982 (Elsevier,Pittsburgh, 2007).

[2] P. Moin and S.V. Apte, *Large-Eddy Simulation of Realistic Gas Turbine Combustors*, AIAA Journal, 44(4), 698-708, April 2006.

[3] G. Boudier, N. Lamarque, G. Staffelbach, L.Y.M. Gicquel, and T. Poinsot. *Thermo-acoustic stability of a helicopter gas turbine combustor using large-eddy simulations*, International Journal of Aeroacoustics, 8(1):69-94, 2009.

[4] G. Staffelbach, L.Y.M. Gicquel, G. Boudier, and T. Poinsot. *Large Eddy Simulation of self excited azimuthal modes in annular combustors*. Proc. of the Combustion Institute, 32, 2009.

[5] G. Staffelbach, L.Y.M. Gicquel, and T. Poinsot. *Highly parallel large eddy simulations of multiburner configurations in industrial gas turbines*. The Cyprus International Symposium on Complex Effects in Large Eddy Simulation, 2005.

[6] European Aviation Safety Agency, CS-E Rules for the Certification of engines, 2007

[7] F. Deheeger, M. Lemaire, M. Pendola, D. and Vallino, *Reliability analysis of a blade shedding safety system.* In Safety and Reliability of Engineering Systems and Structures, A. G., ed., Rome, Millpress, Rotterdam, June, 19-23, 2005, Schuller G.I. and Ciampoli M. Eds, p. 544.

[8] M. Herran, H. Chalons, D. Nélias and R. Ortiz, *Implementation of rotor dynamics effects into the Europlexus code for the prediction of transient dynamic loading of engine mountings due to Blade Shedding unbalance*, In Proc of ASME Turboexpo 2009, GT2009-59615

[9] M. Herran, H. Chalons, D. Nélias, R. and Ortiz, *Modelling the impact of a blade on a shield during a blade shedding* . In Proc. of the Vibrations, Choc and Bruit Congress 2008, Ecully, France.

[10] A. Rusinek, J.R. Klepaczko, R. Bernier, *Caractérisation de trois alliages et modélisation du comportement themo-visco-plastique*, Internal technical report Turbomeca - LPMM Metz, 2007

[11] K.S. Carney, J.M. Pereira, D.M. Revilock, P. Matheny, *Jet engine fan blade containment using an alternate geometry*, International Journal of Impact Engineering 2008, 1-9