USE OF ADVANCED CFD FOR THE DESIGN OF SNECMA MOTEURS TURBO-MACHINERY

N. Liamis, F. Bastin, J. Lépine
Snecma Moteurs
Centre de Villaroche, 77550 Moissy Cramayel, France

Keywords: *CFD, Turbo-machinery, Compressor, Combustion Chamber, Turbine*

Abstract

During the last two decades, CFD tools have been a significant source of improvements in the design process of turbomachinery components, leading to higher performances, cost and cycle savings as well as lower associated risks. Such methods are the backbone of compressor, combustion chamber and turbine design methodology at Snecma Moteurs and their use is a key factor in designing components with high performance level. Moreover, the first test series of components designed with this methodology generally show performances close to the objectives that were assigned.

In the late 90's new, modern and powerful methods have been developed, led by a constant progress in computation power. As a matter of fact, methods such as steady 3D Navier-Stokes multistage solvers, unsteady 3D Navier-Stokes stage solvers, coupled heat transfer and combustion solvers, and even Large Eddy Simulation are now commonly used in the design process. As an example, several thousands of 3D Navier-Stokes computations are done yearly at Snecma Moteurs for the design of low and high pressure compressor and turbine blades.

In addition to their modeling capabilities, the efficient use of such methods through the design process has been enabled by a close integration in a global methodology and an adequate exploitation environment. Their validation against experimental data, their calibration, the correlation between different levels of modeling are of critical importance to an actual improvement in design know-how.

In the present paper, the integration of different CFD methods in the design process is described. Several examples of application illustrate their practical use. Some comparisons between computational results and experimental data show their capabilities as well as their present limitations. The prospects linked to developments, currently under way, are discussed.

1 Introduction

The design of advanced turbomachinery components has to meet ever more requirements that are demanding. Higher performance must be reached within shorter design cycles and at lower cost. Moreover, ambitious objectives of weight, complexity and manufacturing cost reduction are assigned. For compressor, combustion chamber and turbine designers, this implies the capability to control the complex flow phenomena occurring in all the operating range of the component, early in the program. Besides performances, the aggressive design of advanced components also requires an early focus on all the aspects related to engine integrity.

Up to the mid 70's, most of the design and optimization process relied on an empirical approach, which meant a large number of tests. The all-experimental optimization strategy was time and cost consuming for two reasons at least. Each iteration implies all the phases from design to manufacturing, instrumentation and testing. Secondly, determining what must be improved in the design required heavy instrumentation on each component, in order to identify the "hard spots". For these reasons, Snecma Moteurs started very early to take advantage of the fast growing computer power and of the concurrent advances in CFD. The use of CFD tools in the design methodology has brought major improvements in the iterative optimization process of each component. An attempt to summarize this contribution could use the following keywords : faster response, broader range of alternative solutions, better description of flow complexity. Indeed, every computation node in a numerical simulation is also a “measurement” node, which allows an easy and comprehensive analysis of the flow prediction.

But unfortunately, CFD is far from faithfully reproducing reality yet. Even with the power of the most recent super computers, simulation capabilities still depend on models or are limited to component parts. The computation of a complete 3D component with unsteady and viscous small-scale phenomena is still out of reach for a long time. So the major challenge for both the engine component designer and the CFD method developer consists in integrating new computational methods, with their capabilities and limitations, in the design process in a fast, safe and efficient way. Every new method brings new answers, but also raises new questions. The most obvious risks in using a new, more powerful tool are either misunderstanding or overconfidence in the results. Therefore, a constant effort must be dedicated to methods of comparison, validation and calibration. This means in particular that heavily instrumented rigs, representative of real engine components must be used to produce a validation database.

At Snecma Moteurs, a strong interaction between component designers and CFD tools developers allows an early use of advanced methods in the design process. Most of the CFD developments have been undertaken in close cooperation with Research Laboratories and especially with Onera.

This paper will present some current applications of advanced CFD tools for compressor, combustion chamber and turbine design. Some comparison with experimental results will show the prediction capabilities of different methods. Finally, conclusions will be drawn on the current use of CFD tools in the design methodology.

2 CFD applications in compressor and fan design

2.1 Objectives of modern compressors and fans design

During the last 3 decades, the introduction of ever-improving CFD tools in the methodology of compressors design has led to an improvement of global performance as well as a constant reduction of design time cycle. Thanks to the implementation of potential methods in the 70’s, followed by Euler methods in the early 80’s and then by stationary Navier-Stokes methods since 90’s, the aerodynamic efficiency of fans has been improved by about 10 % in 25 years, as illustrated by figure 1. High and low pressure compressors have known the same kind of evolution with another step in the late 90’s with the introduction of steady multi-stage and unsteady rotor-stator simulation.

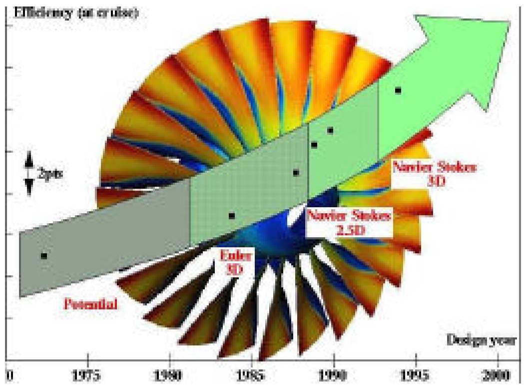


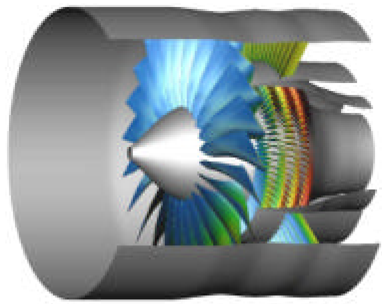
Figure 1 : Impact of CFD on the improvement of efficiency of Snecma Moteurs civil fans.

Nevertheless, this race towards higher aerodynamic efficiency is now slowing and the objectives of modern compressors and fans are slightly different than it used to be in last decades. In the case of fans, the next generation is most likely to have the same efficiency than the previous one because it is close to a theoretical optimum. Thus, the goal for designers will be to maintain this level of performance while focusing on noise reduction and cost savings. For multi-stage compressors, the main targets are a better operating margin and lower costs. In both cases, reducing cost can be translated into two main objectives. On the one hand the number of blades and stages has to be reduced, which leads to highly loaded blades and very complicated flows, and on the other hand, the design time cycle has to be shorten, which requires a fast and reliable design process. The only realistic way to achieve all these objectives is rely on an intensive and appropriate use of CFD [1].

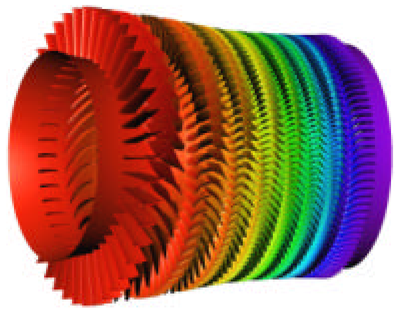
2.2 Use of CFD in compressor and fan design

Thanks to modern super-computers, simulating the flow in a whole compressor module of a civil aircraft engine is today something quite easy to achieve. A stationary solution of the flow in a multi-stage compressor can be obtained within few hours with a well optimized code used on a multi-processor computer. As an example, figure 2 displays the results of a fan-booster-OGV set and of a high pressure compressor simulations, obtained by using the multi-stage 3D Navier-Stokes solver CANARI, developed by ONERA and based on the mixing plane method. This kind of multi-stage computations is very helpful to optimize the matching of high pressure and low pressure compressors, but it can also be very hazardous if the use of CFD is not perfectly under control.

Most of the computations that are done are steady because they require a reasonable time and because not much more information is needed for the aerodynamic design of a blade. However, although they are very expensive in terms of computer time, unsteady simulations are very useful for acoustic and aero-elastic analysis. Therefore, this kind of tool is used only for final analysis, but there is no doubt that within few years it will be intensively used, being part of a whole integrated multi-disciplinary design process.



a) Flow field of a fan-OGV-booster configuration (static pressure)



b) Flow field of a HP compressor (static pressure)

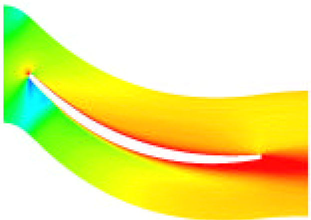
Figure 2 : 3D multistage steady Navier-Stokes compressor simulations.

In the next 2 parts, we focus on aspects of CFD that are of main importance for compressor designers.

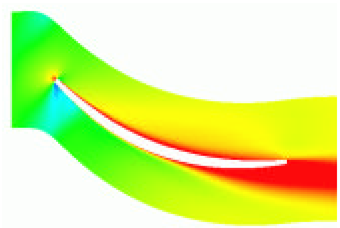
2.2.1 Mesh and turbulence model

The boundary layer effect in a compressor is of main importance, especially in the last stages of high pressure compressors, where it is of the same order of size than the annulus. At of off-design conditions (i.e. close to surge point), secondary flows, chock/boundary layer interaction and stall dominate the whole flow. For these raisons, mesh refinement and turbulence models have a strong impact on the computation results. A bad choice in these parameters can result to an over-estimation of the surge margin and a poor operability. This is illustrated by figure 3, displaying for the same stator at 10% span four computations with four different turbulence models. This simulation is close to the surge point and a large stall occurs, which is well correlated by measurements. The results have been obtained with a structured grid of about 1 million points using similar codes : CANARI from ONERA and Turbo3D from

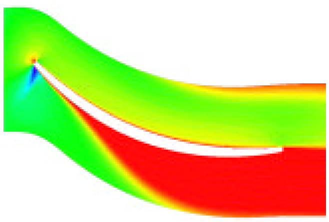
UPMC/LEMFI. In the case of algebraic (CANARI/Michel) and 1 transport equation (CANARI/Spalart-Allmaras) models, the stall is not predicted because the model is unable to predict the boundary layer thickening, whereas in the case of 2 transport equations models (CANARI/k-l and Turbo3D/k-ε) the prediction is very satisfactory.



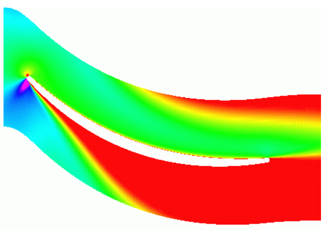
a) Algebraic model (Michel)



b) Spalart-Allmaras model



c) k-ε model



d) k-l model

Figure 3 : Importance of the turbulence model in off-design computations.

The same computation with a 2 transport equations model of turbulence on a lighter mesh (400 000 points) is also unable to predict the stall. In fact, it appears that the parameter that has to be controlled is the size of the first mesh cell. A good rule is generally to have a y+ on hub, casing and blade of about 1. However, some models require thinner mesh as, for example, k-ε model that gives better results with a value of y+ about 0.3. Although this rule is easy to apply for structured grids, the control of the mesh in the boundary layer becomes a tricky problem for unstructured grid.

For some cases, 2 transport equations turbulence models are not precise enough for an accurate flow analysis. For example, to carry out an acoustic analysis from a computed flow, it is needed to take into account the non-isotropic character of the turbulence, which is not possible with such turbulence models. A good way to overcome this difficulty is to lead a full resolution of the Reynolds stress tensor. This is the solution chosen by LEMFI [2], [3] for the code Turbo3D, used at Snecma Moteurs for computing flows with a Reynolds Stress Model (RSM).

2.2.2 Technology modeling

A great challenge for the compressor simulation is to take into account the technology impact on the flow. Indeed, tip clearance, bleeding, hub or shroud misalignment can have a major effect on the flow and the compressor performance [4], [5], [6]. The small size and the complex geometrical definition of these technological elements make them difficult to implement, whereas their influence can be of first order.

Tip clearance and bleeding can be analyzed with quite simple modeling, requiring no major modification of the grid topology and of the code. Therefore, they are commonly taken into account and their impact is well understood.

Other technological effects require either grid topology modifications by adding blocks, either an implementation of a more complex methodology. Therefore, to tackle with these kind of configurations, it is a good compromise to compute the compressor once, including all the technology elements, in order to extract rules or models applicable to simpler simulations. One application of this kind, is presented on figure 4, where the cavity between the fixed and rotating parts of the hub has been computed. With an analysis of this result, it is possible to understand the phenomena taking place in this complex flow area and then to implement the cavity effect in a lighter way, using only boundary conditions.

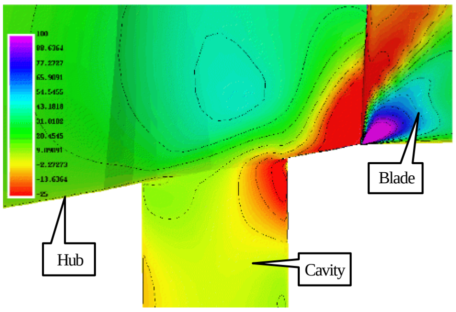


Figure 4 : Modeling a cavity in front of rotor.

3 CFD applications on combustion chamber design

3.1 The 3D Navier-Stokes platform for combustion: N3S-Natur

To the aerodynamic challenges that have been the target of CFD for a long time, such as turbulence, mixing, swirling flows or cross-flows interactions, the design of a combustor adds the complexity of specific physical phenomena : essentially two-phase flow physics and combustion. The CFD available experience for these last aspects is more recent. Therefore, combustion design generally relies more on experimental investigations and empiric rules of design.

Snecma Moteurs has decided very early to invest into CFD, by developing a strong cooperation with Onera, CNRS (Laboratories EM2C, CORIA, LCSR, LCD…) and Cerfacs. It was decided to build a 3D Navier-Stokes solver targeted on combustion applications. A few constraints were highlighted. The computational grid had to be easily generated, and was to cope with the general geometrical complexity and diversity of combustion systems (single annular, double annular, axially staged combustors, reheat systems…). The reheat system, and to some extent the main combustion chamber are regulated downstream by a sonic throat (in the first case the main nozzle, in the second case the high pressure turbine distributor), which could be represented. Industrial turnaround times were finally expected on easily accessible computer resources.

These objectives led to the development of N3S-Natur, which is a compressible, multi-species 3D Navier-Stokes solver, using unstructured tetrahedral meshes, coupled with a variety of turbulence, two-phase flow and combustion models. The code is parallelized and runs on platforms ranging from vector or scalar supercomputers to PC clusters. The code was co-developed by the french electric power company EDF, the car manufacturer Renault, and Snecma Moteurs, with scientific input from Inria, CNRS/LMFA, and Simulog. This association, which allows to reduce the investment costs for each partner, also turned out to be an efficient way of technical "cross-fertilizing".

The core industrial application for N3S-Natur is the definition of the air distribution in the dilution zone to optimize the temperature field at the exit of the combustor, and therefore ensure the safety of the high pressure turbine. Dilution air control is an essential factor for avoiding excessively hot spots seen by the turbine blades. The detail of the modeling level for theses simulations may be found in [7] or [8]. The basic components are a lagrangian solver for the liquid fuel break-up and vaporization, a k-ε type turbulence model, and a presumed pdf (probability density function) turbulent combustion model. The combustion model is suited to non-premixed combustion, and based on infinitely fast chemistry, where the reaction rate source term is controlled by mixing, with a characteristic time provided by the turbulence model. It is limited by chemical equilibrium, which is precomputed and tabulated using the detailed (complex) chemistry of kerosene oxidation.

3.2 N3S-Natur: current and future applications

Grid construction is essential for combustion chamber simulation. Snecma Moteurs was previously relying on a code using structured meshes. Mesh construction is actually unstructured, automated and coupled to a combustion chamber database, and takes between 15 minutes and two days for the most complex injection system. An example of surface mesh is shown in figure 5, where special care is taken to provide enough grid points across each air passage, without impacting the grid across the whole chamber, as in the case with structured grids.

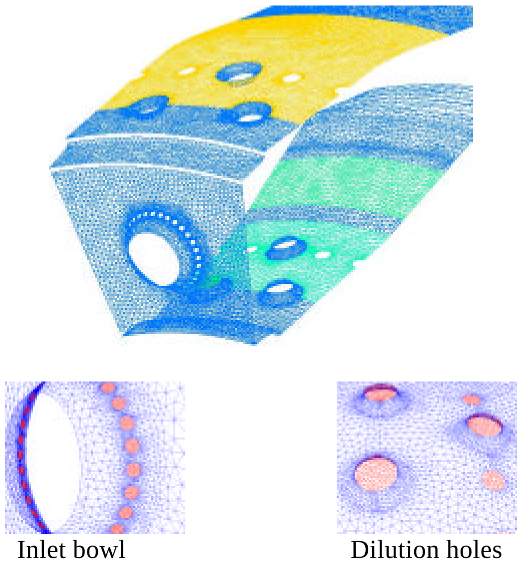


Figure 5 : Unstructured surface mesh for a single annular flame tube.

The code is continuously validated versus the experimental database of Snecma Moteurs combustors. Figure 6 shows results obtained on a single annular combustor of current generation. The exit temperature field is predicted at a level allowing optimization and design choices. It must be noticed that such results are due not only to the code, but also to the boundary condition definition methodology. Regarding the injection system, a complex mix of numerical simulation, experimental characterization, and extrapolation from test to design point conditions is still necessary.

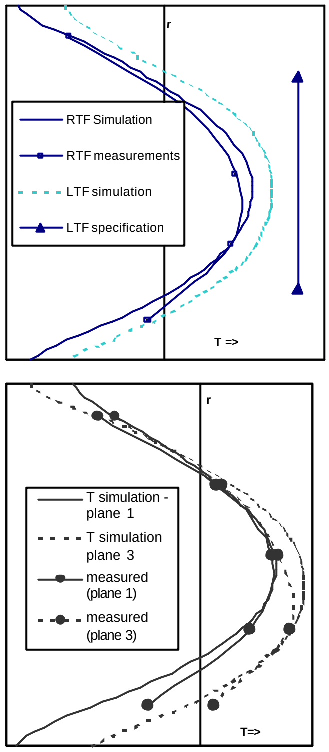
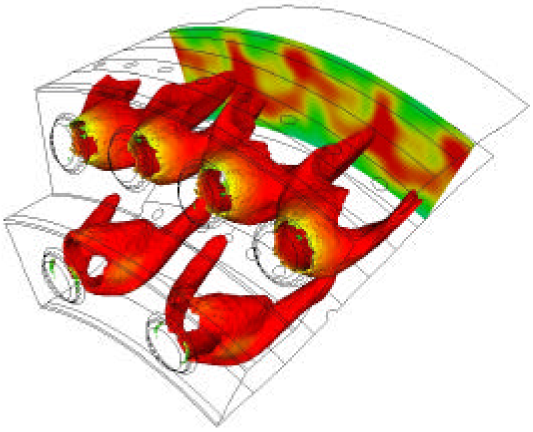


Figure 6 : Exit temperature profiles. The RTF is the radial temperature factor, where a spanwise average is performed at each radial location. The LTF is the lateral temperature profile, indicating the maximum temperature at each radial location. Plane 1 is the radial plane including the injection system axis. Plane 3 is the intermediary radial plane, between injection systems.

As mentioned above, correlations may be established for classical single annular combustors with constant cooling or injection technological level, but fail to cover the diversity of solutions developed by the combustion designer. CFD is therefore useful for concepts such as double annular or axially staged combustors, which are one of the ways to resolve contradictory emission and stability requirements, and therefore obtain low emission engines. An illustration of the use of N3S-Natur on a chamber design is given in figure 7 for an axially staged combustor. CFD is used here to optimize the exit temperature field under constraint of maintaining the efficiency (limiting over-rich regions within the chamber).



Fuel mixture fractions

Figure 7 : Simulation of an axially staged combustor.

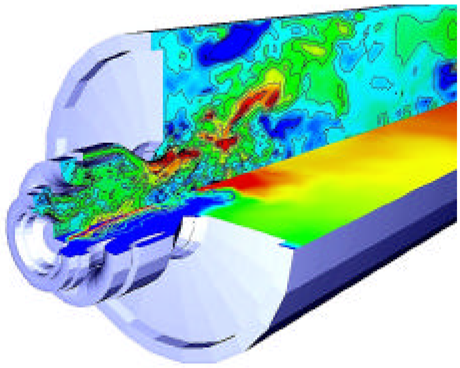
Models regarding combustion and turbulence aspects are continuously developed for N3S-Natur, but the present development plan is focused on finite rate chemistry effects. Infinitely fast chemistry, used on the present approach, is suited to the temperature prediction at high engine operating points (cruise or take-off), and to some extent of NOx production. But it is not appropriate to the simulation of CO, or to the behavior of the chamber far from those points (stability analysis). These performances are presently estimated via correlations and simplified approaches. Current developments aim to take into account finite rate chemistry effects within the Navier-Stokes simulation itself. Advanced transported pdf methods, which produced excellent result [9], are therefore incorporated in N3S-Natur. The actual computation is still not affordable because the chemistry of kerosene oxidation involves a few hundred chemical species and more of a thousand chemical reactions. Therefore, reduced chemistries must be developed, posing the problem of reducing a high-dimensionality real dynamical system to an much lower-dimensionality approached system. Techniques such as neuron networks are investigated for this aspect. A real combustion chamber computation including detailed chemistry effects, which was not even considered a few years ago, is finally considered as close perspective, with turn-around times a few times higher than present industrial simulations.

3.3 Large Eddy Simulation for combustion

Large Eddy Simulation (LES) is a full of promise technique, in which Snecma Moteurs decided to invest for combustion applications, for two main reasons. The first is the prediction of combustion instabilities, a complex coupling between combustion and pressure oscillations within the chamber. An essential component of such a process is the flame response to upstream perturbations. This response and more generally the combustion dynamics are not seen by classical RANS approaches, while LES has shows promising results. The second motive is related to combustion modeling prospects. Current modeling is applied on a statistically averaged aerodynamic field, provided by RANS simulations, while the combustion process takes place on a mixture of fuel and oxidizer that is local in space and time, and is generally different from the statistical average. LES, by its ability to capture large scale intermittency, simplify the combustion modeling problem ; the turbulence model "closure" problem is simpler.

LES being an advanced technique, the choice was made to rely on the expertise of Cerfacs Research Center. The code developed by Cerfacs and relying on the parallel library COUPL (maintained at Oxford University) is AVBP [10]. The basic philosophy was to design a code perfectly suited for compressible LES, and therefore to pay special attention to the properties of the numerical scheme, dissipation, and boundary conditions. An efficient parallelism and the use of multi-elements grids, able to represent arbitrarily complex geometries, ensure industrial turn-around times.

An interesting application of LES was done on LPP (Lean Premixed Prevaporized) injection systems. These systems present attractive properties in terms of NOx reduction, but they are also prone to combustion instabilities, especially to flashback, where the flame moves upstream into the LPP, leaving its intended stabilization location in the chamber. Successful LES of flashback were reported in [8] in 2D, but computations are now performed in 3D with a good agreement with the measured behavior of the module (figure 8).



Velocity and temperature

Figure 8 : 3D Large Eddy Simulation of a premixing injection system.

An interesting quantitative information that LES can produce is the flame response, represented by a delay time τ and a sensitivity index n between a pressure or velocity perturbation and the heat release response. Comparison of such quantities with experiments conducted by the CNRS laboratory EM2C have shown how LES is able to produce quantities which are critical to dynamic problems such as combustion instabilities, and difficult to evaluate via RANS methods. The perspective for LES is taking into account two-phase flows and complex chemistry. Work is going on and LES is expected to gain ground in the following years.

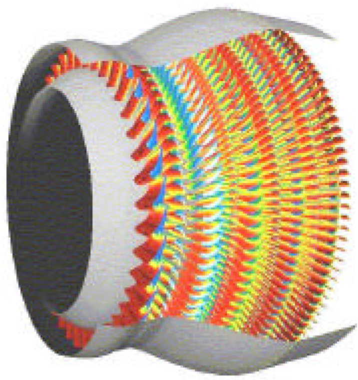
4 CFD applications on turbine design

4.1 Aerodynamic analysis

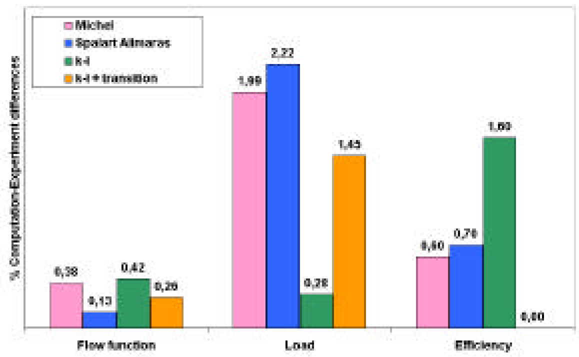
The first step of the turbine aerodynamic design consists to the definition of its operating point. Two ways are used to model its behavior : through-flow simulations or 3D multi-stage simulations. If the first method can be judged satisfactory during the pre-design phase when cycles are not well defined, the second one is preferable during the design phase for its capacity to provide more accurately the velocity triangles and the coolant injection effects.

The steady multi-stage 3D Navier-Stokes calculations are carried out with the CANARI solver, developed by Onera. They are based on a mixing plane model and all the technological effects (tip clearance, filet radii, groove) and coolant injection regions (cooling holes, trailing edge slots, platforms leakage, high pressure cavity leakage) can be taken into account [11], [12]. A result of a multi-stage dual spool configuration simulation (cooled high pressure turbine + low pressure turbine) including all technological effects is given on figure 9 a).

Turbulence and transition phenomena can modify significantly the turbine behavior. For design practices, the boundary layer of film cooled blades can be considered as fully turbulent, while for blades not cooled with film but protected by internal circuits or/and thermal barrier cooling, transition modeling can be helpful to predict blade heat transfer coefficient. In all cases the turbulence and the transition modeling have an impact on the turbine performances prediction as shown in figure 9 b) for a low pressure turbine.



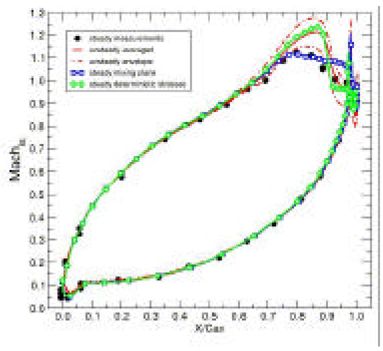
a) Flow field of a HPT + LPT configuration (static pressure)



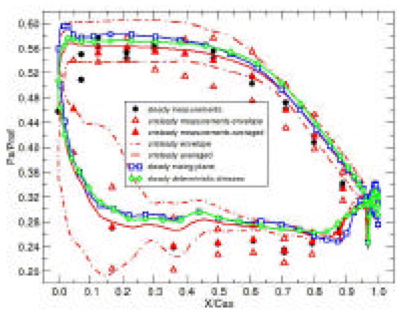
b) Influence of turbulence model and transition on LPT performances

Figure 9 : 3D multistage steady Navier-Stokes turbine simulations.

Moreover, unsteady effects due to potential interactions, turbulence, shocks and wakes on blade leading edge, as well as cold / hot spot impact on blade profile are important in turbine environment. So, even if their prediction is not yet entirely part of the design process, unsteady phenomena are today studied and quantified on turbine configurations. Two different approaches, based on the 3D Navier-Stokes solver CANARI, are used : unsteady stage calculations and steady stage calculations using the deterministic stresses model, which takes into account through modelisation, for a given row, the effects of the neighboring rows [13]. Figure 10 a) and b) presents, for an uncooled high pressure turbine stage, the results obtained by a steady stage simulation using the mixing plane model, an unsteady stage simulation and a steady stage simulation using the deterministic stresses model, compared to measurements.



a) NGV 50% span isentropic Mach number



b) Rotor blade 50% span static pressure

Figure 10 : Influence of stage modeling for high pressure turbine simulations.

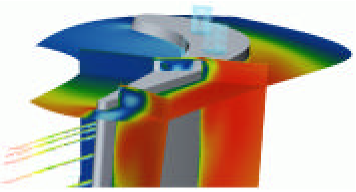
4.2 Aerothermal analysis

The aerothermal design of turbine blades is of primary importance for the high pressure turbines, where the inlet temperature, several hundreds of Kelvin higher than the maximum temperature admissible by the material, implies the use of cooling techniques. It is closely linked with the aerodynamic design and the mechanical design. Moreover, this multidisciplinary iterative design has to take into account the manufacture constrains, during each project step. Several computational procedures are used for the design and analysis of turbine aerothermal phenomena : the first one for the investigation of turbine external flows, the second one for internal flows in blade cooling cavities and the third one for the prediction of blade metal temperature field.

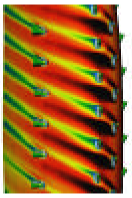
4.2.1 External aerothermal analysis

The critical environment, generated by the high inlet turbine temperature, has led to the use of cooling flows, in order to increase blade life. In particular, film cooling is now a common technique used to prevent the blades from overheating and the prediction of its performance is a major issue in view of the blade efficiency and life. For a given aerodynamical flow field, the number, the location, the pitch, the shape, the streamwise and compound injection angles of holes and trailing edge slots are important geometrical parameters for the design of a cooled blade.

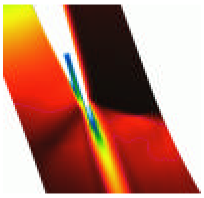
3D Navier-Stokes calculations of film cooling is a challenge, because the phenomena involved are very complex, due to the very vortical flows and to their length scales, which are very different from the common length scales of the flow around the blades. As for the aerodynamic analysis, the Navier-Stokes simulations for the film cooling prediction [14], [15] are performed with the CANARI solver, developed by Onera. Some results, concerning grooved tip, leading and trailing edge cooling, are presented on figure 11 a), b) and c).



a) blade tip cooling
(Mach number)



b) leading edge cooling
(wall heat flux)



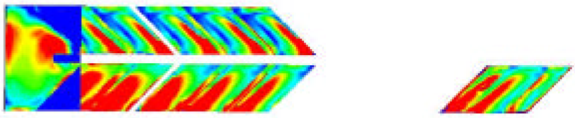
c) trailing edge cooling
(Mach number)

Figure 11 : External aerothermal analysis of turbine blade.

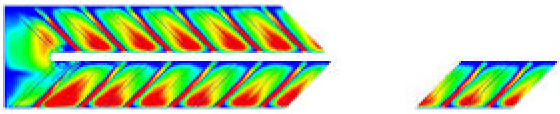
4.2.2 Internal aerothermal analysis

Modern turbine airfoils are designed with internal cooling passages, arranged with radial or serpentine patterns, through which the coolant air travels to remove heat from the airfoil walls, in order to ensure blade life. The internal cooling performances of turbine blades are highly dependant on the technologies used : ribs, pedestals, pins fins, impingement, blade internal circuits. An important part of the designer's work is to optimize the cooling flow systems encountered inside the blade, minimizing the amount of cooling mass flow and optimizing the cooling efficiency. Thus, 3D Navier-Stokes calculations are carried out to analyze, in details, flow and heat transfer on new complex technological concept [16].

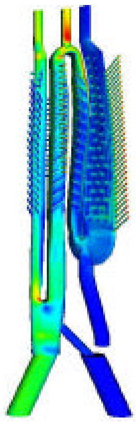
The Navier-Stokes simulations, performed with the MSD solver developed by Onera, provide results close to experimental data, as can been shown by the comparison for the U bend duct case presented on figure 12 a) and b). So, their use, for the prediction of the flow field into internal cooling circuits, is increasingly encouraged. Actually, the whole internal cooling circuit can be considered, as shown on figure 12 c) for a high pressure turbine rotor blade internal cooling circuit containing holes, ribs and pins fins.



a) experiment U bend duct case
(wall heat flux)



b) calculation U bend duct case
(wall heat flux)



c) rotor blade internal circuit
(Mach number)

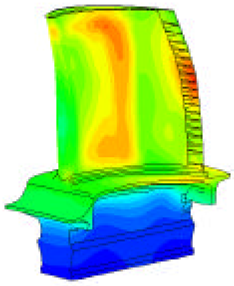
Figure 12 : Internal aerothermal analysis of turbine blade.

4.2.3 Thermal analysis

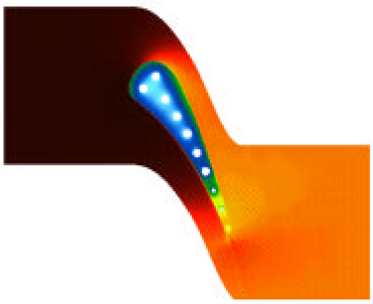
The prediction of the blade metal temperature is the final achievement of the aerothermal design. Due to the fluid / solid interaction, the global heat transfer analysis on blades is one of the most challenging works for turbine designers.

Generally, the Navier-Stokes simulations and the conduction calculations are used independently, a manual iteration model being used between the fluid and the solid media. The methodology adopted is to prescribe the couple heat transfer coefficient / reference temperature from the Navier-Stokes code to the conduction solver, and the wall temperature from the conduction solver to the Navier-Stokes code. A result of the iterative (uncoupled) thermal analysis process, provided by the commercial conduction solver ABAQUS, for a high pressure turbine rotor blade protected by thermal barrier coating, is presented on figure 13 a).

However, the manual iteration methodology is time consuming and can be faulty in general 3D flows. Moreover, results depend on the number of manual iterations done by the designers. Therefore, to prevent these problems, an automatic general coupling procedure between the Navier-Stokes solvers and the conduction solver ABAQUS has been developed [17]. A result of a conjugate heat transfer calculation, carried out on the NASA C3X cooled blade with internal cavities, is presented on the figure 13 b).



a) uncoupled heat transfer
(metal temperature)



b) conjugate heat transfer
(static and metal temperature)

Figure 13 : Thermal analysis of turbine blade.

5. Conclusion

The role of CFD in the design process of aero-engine components is fast growing. Numerical simulation significantly contributes to reaching industrial objectives such as the reduction of cycle time and cost in a project development. It is constantly demonstrating its effectiveness on several applications, allowing an earlier and more reliable component optimization.

The advances in computer technologies and the combined development of CFD techniques increasingly enlarge the field of possible applications. In this paper, a broad spectrum of examples in aero-engine component design and analysis shows the capabilities of today's CFD methods.

However, the increasing importance of CFD requires special care to turn numerical methods into integrated tools easy to handle by component designers. Validation and calibration are essential to an effective use of advanced, but still limited, simulation techniques. Well-instrumented experimental vehicles are definitely needed for this purpose.

The integration of advanced CFD tools in the design methodology of aero-engine components have marked an important milestone and have led to significant changes in the overall design process. The development of new applications makes the near future very promising. The next generation of computers will enable a new step in component optimization. However, much has to be done in numerical techniques and modeling of flow physics, in order to turn this opportunity into an industrial achievement.

Acknowledgements

The authors wish to thank Snecma Moteurs for the authorization to publish the present paper. The results previously presented are issued from several works, partly supported by the french organizations DPAC and DGA. It must be emphasized that these results have been obtained by many co-workers at Snecma Moteurs, during the past years. Their help in reviewing and shaping up the manuscript is also gratefully acknowledged.

References

[1] J.F. Escuret, D. Nicoud, P. Veysseyre, Recent advances in compressor aerodynamic design and

analysis, AGARD RTO Lecture series 211, RTO-EN-1, September 1998.

[2] G.A. Gerolymos and I. Vallet, Tip-clearance and secondary flows in a transonic compressor rotor, ASME 98-GT-366.

[3] G.A. Gerolymos and I. Vallet, Wall-normal-free Reynolds-Stress Model for compressible rotating flows applied to turbomachinery, AIAA Journal, vol. 40, 2002.

[4] S.R. Wellborn, I. Tolchinsky and T.H. Okiishi, Modeling shrouded stator cavity flows in axial-flow compressors, ASME 99-GT-75.

[5] A.A.J. Demargne and J.P. Longley, Cavity and protrusion effects in a single-stage compressor, ASME 2001-GT-433.

[6] J.D. Denton, Lessons learned from Rotor 37, presented at the 3rd International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows (ISAIF), Beijing, China, September 1-6, 1996.

[7] F. Ravet, L. Vervisch, Modelling non-premixed turbulent combustion in aeronautical engines using PDF generator, AIAA Paper 98-1027.

[8] O. Mahias, F. Bastin, F. Ravet, C. François, Lean premixed combustor emissions performance modeling using 3D CFD codes, AIAA Paper 2000-3199.

[9] J.C Larroya, C. François, M. Cazalens, L. Vervisch, Testing a new Monte Carlo method for solving pdf equation in turbulent combustion, Fifth International Conference on Technologies and Combustion for a clean environment, vol I, pp 177-183, 1999.

[10] T. Schönfeld, M.A. Rudgyard, Steady and unsteady flow simulations using the hybrid flow solver AVBP. AIAA Journal, 37(11), November 1999, pp 1378-1385.

[11] Liamis, N., Duboué, JM., CFD Analysis of High Pressure Turbines, ASME Paper 98-GT-453.

[12] Liamis, N., Brisset, C., Lacorre, F., Duboué JM., CFD Analysis of Dual Spool Turbine Configuration, AIAA Paper 99-2525.

[13] Kirtley, KR., Turner MG., Saeidi S., An Averaged Passage Closure Model for General Meshes, ASME Paper 99-GT-77.

[14] Fougères, J.-M. & Heider, R. (1994) : Three-Dimensional Navier-Stokes Prediction of Heat Transfer with Film Cooling. ASME Paper 94-GT-14.

[15] Ginibre, P., Lefebvre, M., Liamis N., Numerical Investigation of Heat Transfer on Film Cooled Turbine Blades, Heat Transfer in Gas Turbine Systems, Annals of the New York Academy of Sciences, vol 934, pp 377-384, 2001.

[16] Paté, L., Duboué, JM., CFD : A Tool to Design Jet Engine Internal Cooling Systems, AIAA Paper 98-3563.

[17] Montenay, A., Paté, L., Duboué, JM., Conjugate Heat Tranfer Analysis of an Engine Internal Cavity, ASME Paper 2000-GT-282.