TURBOMACHINERY DESIGN USED INTENSIVE CFD

H. Joubert, H. Quiniou
Snecma
Villaroche
77550 Moissy

Keywords: *Turbine, Compressor,Aerodynamic, CFD, Steady , Unsteady, Flutter*

Abstract

Computational Fluid Dynamics (CFD) tools represent a significant source of improvement in the design process of turbomachines, aiming to better performances, lower costs and associated risks.

Snecma compressor and turbine design process relies intensively on CFD methods. In the 80s, the use of 3D Euler methods was the cornerstone of commercial fan design methodology meeting high level of efficiency. In the early 90s, 3D Navier-Stokes solver and unsteady 3D Euler solvers were introduced in design process. Hundreds of 3D Navier-Stokes calculations were run before the release of a compressor or a turbine.

Numerical models are constantly improved to account for real geometry and include various technological effects (tip clearances, fillet radii, flowpath alignment, bleeds, cooling flows…) and multi stage simulations. This means to implement more complex turbulence models, transition simulation and to simulate unsteady phenomena

This paper illustrates the effort made by Snecma to adapt these methods to design process following three patches:

- *Simulation improvement*
- *CFD methods validation / calibration*
- *Adaptation to designer environment*

1. Introduction

The design of advanced aero-engine turbomachinery components has to meet ever more demanding requirements. Higher performance must be achieved within shorter design cycles and at lower cost. Ambitious objectives in the reduction of weight, complexity and manufacturing cost lead to fewer compressor and turbine stages, and therefore to increased stage loading.

For designers, this new situation implies the capability to control the very complex flow phenomena occurring in highly loaded stages, on the whole operating range of the engine, early in the program. In addition to aerodynamic performance, the aggressive design of advanced, fully 3D blades also requires an early focus on all the aspects related to engine mechanical integrity: blade flutter and forced response and thermal constraint.

Up to the end of the 70's, most of the design and optimisation process relied on an empirical approach, which meant a very large number of tests. The all-experimental optimisation strategy was very time and cost consuming for two reasons at least. Each iteration implied all the phases from design to manufacturing, instrumentation and testing. Secondly, determining what had to be improved in the design required a comprehensive instrumentation on real engine components, which was strongly limited by technological constraints. As a result, it was very difficult to identify the potential problems and even more difficult to understand them.

For these reasons, Snecma started very early to take advantage of the fast-growing computer power and of the concurrent advances in Computational Fluid Dynamics ([1], [2], [3] [4], [5]). This approach has been greatly rewarded as the introduction of CFD tools in the design methodology has brought major improvements in the iterative optimisation process of engine components (fans, compressors, turbines and nacelles).

An attempt to summarise 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.

Unfortunately, CFD is still far from faithfully reproducing reality. Even with the power of the most recent supercomputers, simulation capabilities still depend on very approximate physical models or are limited to component parts. The computation of a full multistage high-pressure compressor or high-pressure HP turbine with 3D, unsteady and viscous, cooling and leakage, small-scale phenomena are out of reach for a long time yet. As a result, 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 overoptimistic confidence in the results.

A constant effort must therefore be dedicated to the comparison, validation and calibration of methods. This means in particular that heavily instrumented rigs, representative of real engine flows be used to produce an appropriate validation database.

At Snecma, a strong interaction between designers and CFD tool developers has always allowed an early use of advanced methods in the design process: 3D Euler in 1984, Quasi-3D Navier-Stokes in 1988, 3D Navier-Stokes in 1992. The recent development of parallel vector computing and workstation network now makes possible the use of multistage 3D Navier-Stokes in the design process of multistage compressors and turbines. Most of those recent CFD developments have been undertaken in close co-operation with research laboratories such as ONERA, LEMFI (University of Paris VI) and Ecole Centrale de Lyon and Von Karman Institute.

This paper brings up the aerodynamic design and analysis process of compressors and turbines at SNECMA. Since design methodology has been presented several times, this paper will be focused on analysis tools which are described with a particular emphasis on the numerical basis of currently used CFD codes and on the combined efforts to develop homogeneous and user friendly pre and post-processors.

2. Evolution of compressor and turbine analysis tools

The efficient integration of new computational tools with increased simulation capabilities is a real challenge for the designer. To take advantage of the fast advances in CFD developments, the design methodology and process must be constantly adapted according to new simulation capabilities.

A major new step has recently been introduced in the blade analysis phase by accounting for technological effects such as radius fillets, rotor tip clearances, cooling and leakage or buttons of variable-stagger stator vanes. The current full 3D Navier-Stokes code is now capable of predicting the main consequences of these effects which are considered to be significant in the matching of high-pressure compressor stages or in the design of HP and LP turbines. From a practical point of view, the level of consistency achieved between 3D calculations and throughflow objectives is driven by the industrial know-how based on past experience. The discrepancy observed between 3D and throughflow models highlights the limitations of the classical approach which splits the real flow in two 2D flows called S1 (blade to blade) and S2 (hub to casing). The steady multi-stage 3D Navier-Stokes code introduces a major change in the design process since it is no longer necessary to prescribe aerodynamic conditions from the throughflow model at the interface between blade rows. This new multi-stage tool is now fully integrated into the design methodology even though the analysis of a new design is always initiated using single blade row calculations. As a matter of fact, it appears to be more efficient to initiate the design with rapid calculations and ultimately to check the whole compressor and turbine operation with more elaborate tools such as the multi-stage code.

The use of advanced tools allows the designer to enhance the inlet specific flow of fans and to increase the mean stage loading of compressors and turbine while ensuring adequate stability margins and life requirements.

3. Description of analysis tools

3.1 Navier-Stokes solvers

In the 70's and early 80's, because of computer power limitations, industrial users had to develop specific CFD codes to solve the various problems they had to face. CPU time and memory considerations needed a deep optimisation of software coding. This lead to a set of tools that had their own numerical scheme, pre and post processing.

In the late 80's and early 90's, outstanding development of computer technology and progress made in numerical techniques made possible a large number of new CFD applications. More and more sophisticated methods became available for industry: 2D or 3D, viscous or inviscid, steady or unsteady codes. On the other hand, growing computer capability allowed a more global optimisation of development of CFD codes regarding not only CPU time performances but also codes development and support costs which must be kept at an affordable level. All applications were supposed to take credit from the work on any of them in order to save time and money.

At Snecma, two types of Navier-Stokes solver are currently used:

ONERA's CANARI solver

LEMFI 's Turbo3D

ONERA's CANARI is a compressible, finite volume, time marching, multi-domain code solving full Navier-Stokes equations on a structured grid. The numerical core is based on a four step Runge Kutta explicit scheme combined to Jameson and Turkel second and fourth order numerical dissipation model. On top of this scheme, residual are smoothed by an implicit technique developed by Lerat [4], [5].

Turbulent closure of the Reynolds averaged Navier-Stokes equations is obtained using different turbulence models - Michel's [8] or Baldwin-Lomax [9]. algebraic models, Spallart-Almaras [10] one-equation-model, k-ε two-equations-model [11]. The use of algebraic turbulence model is a good compromise between code robustness, acceptable accuracy and computing costs and elapsed time even if the extension of such models for 3D applications requires special care.

Transition model has been recently introduced in this code . The transition location is computed using the well known Abu-Ghannan and Shaw model [12].

All boundary conditions are imposed through compatibility relations.

For HP turbine coolant flow is simulated by using an injection technique prescribing the upstream conditions of cooling flow.

Moreover the wall clock time required for multi-stage computations is reduced by the ability of CANARI to run in parallel on several processors of a Fujitsu VPP 300 supercomputer. By using this PVM approach a complete steady multistage computation of a LP turbine or a HP compressor can be get in one night.

However in spite of very successful design based on the intensive use of CANARI, higher accuracy turbulent solvers were needed to improve unsteady flow analysis. Developed by the LEMFI laboratory for Rotor/Stator simulations, the TURBO3D code solves the compressible Favre-averaged Navier-Stokes equations, with a two-equation closure of the turbulence terms proposed by Launder & Spalding and Launder & Sharma [11]. This « low Reynolds » model has demonstrated its performance for calculation of separated flow got on fan at part speed or high turbulent flow in rear stages of an high-pressure compressor.

The numerical method of the code is described in detail by Vallet [17], [18]. The mean-flow and turbulence transport equations are expressed into the Cartesian coordinate rotating frame and are discretized in space, on a structured grid, using a 3rd order upwind-biased MUSCL scheme and Van Leer flux vector splitting with Van Albada limiters. The resulting semi-discrete scheme is integrated in time using a 1st order implicit procedure. The resulting scheme is highly robust and efficient.

3.2 Pre and post processing tools

A high quality modular solver is not sufficient to obtain industrial-grade CFD tools. Fast and precise simulation tools are not much helpful if the time and manpower required to build the simulation test case and to analyse it is too important.

Due to the simplicity and repeatability of the compressor geometry, the benefits of structured multiblock meshes can be fully exploited: simple algorithms, easy coding,... As a result, SNECMA has been able to develop its own dedicated mesh generator and associated pre-processing. This mesh generator is based on few « external parameters » (less than one hundred) which can be « graphically » modified to obtain an « optimal » (in terms of quality) mesh. Then, the « batch » version generates one million mesh points for a single blade 3D Navier-Stokes grid in less than one minute on standard Silicon Graphics workstations.

Due to the simplicity and repeatability of the compressor and turbine geometry, the benefits of structured multiblock meshes can be fully exploited: simple algorithms, easy coding... As a result, Snecma has been able to develop its own dedicated mesh generator

To account for technological effect such as radius fillets- or HP turbine blade groove additional processing is performed on the standard mesh. To guarantee future demands and evolution those tools have been developed using « Object-Oriented Languages ».

Finally, the aim of pre-processing is to collect data and requirements from the mesh generator, throughflow computations, and user demands, to define the input grid and boundary conditions of the solver and the aerodynamic input. Initialisation is directly provided by throughflow data for steady computations, then altered as needed to account for boundary layers developing at endwalls. Additional inputs are taken into account for unsteady calculations, like blade vibration modes for flutter simulation for instance.

To be efficient, post-processing tools must offer several levels of investigation. A straightforward, batch tool is necessary to obtain reduced data, providing fast answers even from heavy 3D Navier-Stokes computations. This is particularly convenient in parametric studies around a configuration close to the final one. It provides spanwise distribution of standard flow quantities, or pressure distribution on blade profiles. But a deeper and interactive investigation may be necessary, with strong 3D visualisation capabilities. At Snecma, the solution consists in the combination of in-house, modular tools specific to turbomachinery applications and of 3D visualisation software from vendors.

4. Application of analysis tools to the prediction of turbomachine performance

4.1 Technological effects

For the design of multi-stage high radius ratio such as core compressor or high-pressure turbine, it is very important to account for technological effects such as tip clearance, flowpath misalignment, bleed flows or cooling flows. All these parameters affect widely stage performances (flow and efficiency) and therefore the whole loading stagewise distribution. If not well anticipated, they may

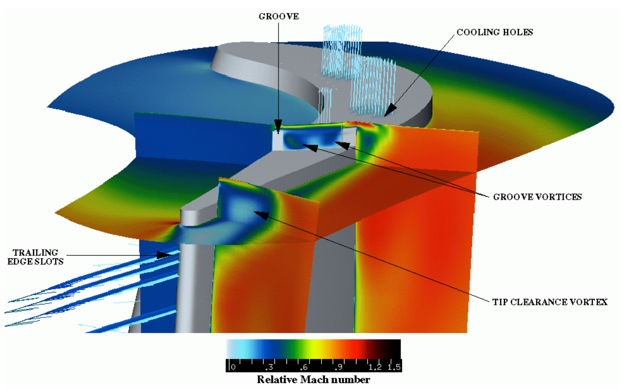

Figure 1 Tip clearance simulation on HP turbine blade

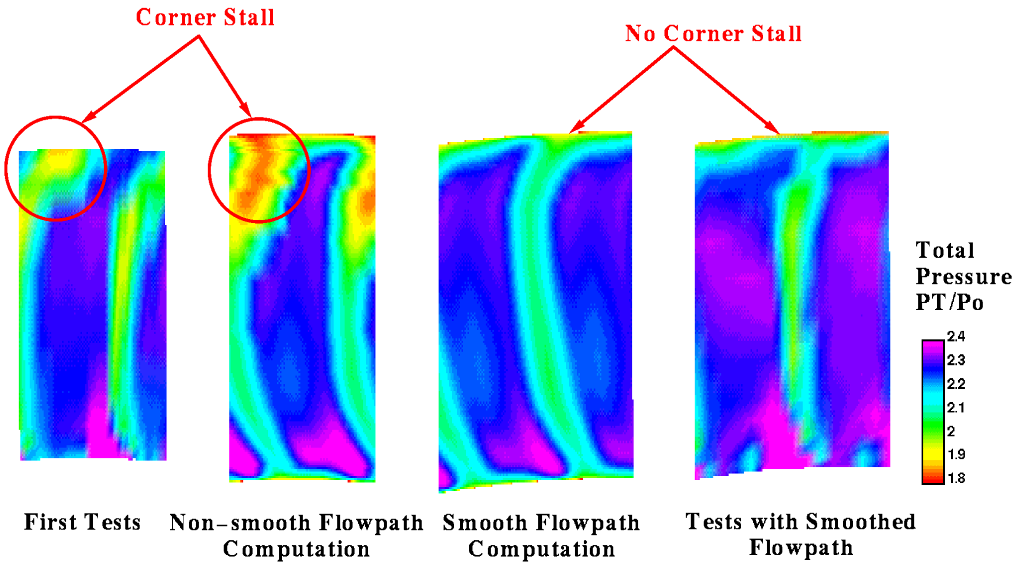

Figure 2 : Compressor variable stagger vanes - Experiment vs Computation Comparison

lead to severe stage mismatching inducing performance penalty (all the stages do not reach high efficiency at the same compressor operating point) or operability defects in the case of a compressor.

Figure 1 shows different secondary flows that are to be accounted for during a cooled turbine design. Tip clearance vortices, cooling flows injected in the turbine blade groove can impact turbine efficiency and heat fluxes coefficient . This tip clearance simulation is used to predict with a very good accuracy the efficiency sensitivity to tip clearance variation. So, with such a tool it is possible to design 3D turbine shapes less sensible to tip clearance variation. This type of simulation is also used to predict the turbine blade tip temperature with a better accuracy in order to keep the integrity of this part of turbine blade during its life.

The consequences of this tools are not limited to turbine performances (efficiency, life) but affect also compressor operating line which can shift off-design and therefore provide poorer flow or efficiency or even unacceptable stall margin jeopardising engine operability

As detailed in referenced paper [19], CANARI code is able to predict flow trends induced by flowpath discontinuity. During the tests of a single stage research compressor, flow measurements were carried out at blade and vane exit. 3D Navies-Stokes simulations (run using CANARI code) on the rotor blades were in good agreement with traverses. Downstream stator vane unexpected discrepancies were found between test data and flow simulation.

As shown on figure 2 a region of high-pressure losses can be identified behind the stator vanes on a substantial portion of the annulus toward the outer annulus. This corner flow separation could not be reproduced by 3D Navier-Stokes computations of the stator vanes.

Following the test results, a close examination of the ECL4 geometry revealed that the buttons of the variable stator vanes were slightly out of line with the flowpath, by a value of about 0.3 mm which amounts to 0.5% of vane height.

As detailed by Escuret[19], using a simple numerical approach to account for the actual geometric discontinuity

of the annulus, the « CANARI » computation then showed flow trends similar to that of the experiment. Also, a detailed analysis of the computed flow field indicated that the flow turning (in both the radial and tangential directions) due to the button blockage contributed toward strengthening the effect of secondary flows. Subsequently, after the geometry of the VSV buttons was corrected to match the flowpath, further tests confirmed much more satisfactory flow behaviour leading to a one-point improvement in overall compressor adiabatic efficiency.

4.2 Multiple blade row calculations

With the recent increase in computational power, the aerodynamic designers are no longer restricted to the analysis of an isolated blade row when using 3D Navier-Stokes computations. It is now possible to consider multiple blade row configurations to take into account the influence of adjacent blade rows [13],[20] and to predict the development of endwall losses across the compressor and turbine, and the matching between vane and blade provided that some simplifying assumptions are made so as to perform a steady computation.

Two different approaches currently used at SNECMA for the coupling of blade rows are presented in this paper:

- the **mixing plane** approach
- the **deterministic stress** approach

4.2.1 Mixing plane approach

In this simple approach, the tangential averages (in the stationary frame of reference) of flow quantities at one side of the interface plane between adjacent blade rows are used to update the numerical scheme values on the other side of the interface plane. The precise choice of the physical flow properties to be averaged and exchanged across the interface plane is not a trivial matter as there is no such average flow that would satisfy all the conservation equations at the same time. A simple solution is to use primitive flow variables, replacing the internal energy by the static pressure: ρ, ρV_x, ρV_r, ρV_t, p, k, ε. Another widely used approach is to use conservative flow variables, i.e. ρV_x, ρV_x+p, $\rho V_x V_r$, $\rho V_x V_t$, $\rho V_x H$. A third approach is to favour the continuity of the entropy flux over that of ρV_x+p. Whatever solution is used, a natural effect of the mixing plane approach is that the average of some physical flow properties are discontinuous across the interface plane between adjacent airfoils.

Validations of this methodology have been done by applying CANARI multistage computation on different single stage uncooled HP turbines. The experiments were performed in cold flow continuous test rig. The experimental configuration investigates the nominal 1% tip clearance case. Two turbines were tested: the first one has a classical design parameters, the second one is a high lift design , the blade and vane counts have been reduced to reduce weight and cost. During the tests, the total isentropic efficiency is determined by measurements mass-weighted total pressure and by measurement of the torque provided by a torquemeter. Test accuracy is equal to 0.5 point.

The nominal operating condition are summarized in the following table :

Inlet total pressure	320000 Pa
Inlet total temperature	400 K
Rotational speed	8292 RPM
Turbine work	235 J/Kg/K

In the multi-blade computation the total number of mesh points is equal to 2.8 millions, the turbulence model is the Michel's one, the flow is assumed to be full turbulent (figure 3).

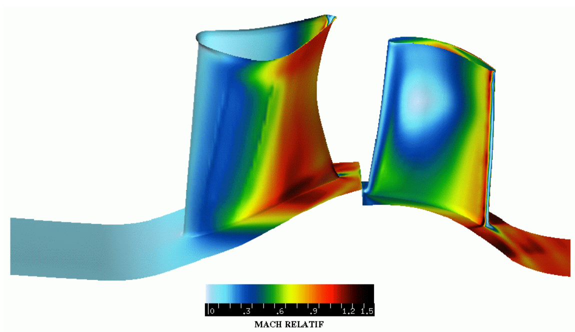

Figure 3 : Single Stage HP Turbine Calculation : Mixing Plane Approach

CFD isentropic efficiency is determined by integration of the exit total temperature and exit total pressure. Good agreement between the mixing plane multistage results and the tests results is found (figure 4). In particular the measured efficiency gap between the two tested turbines is well got by the calculation.

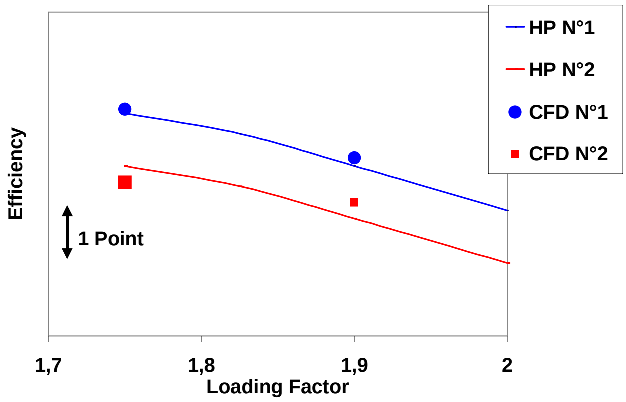

Figure 4: Single Stage HP Turbine Efficiency Comparison

Another validation has been done on a dual spool turbine configuration (figure 5). Single stage HP turbine is cooled and 4 stage LP turbine is uncooled. Full cooling methods (Liamis [13]) is used to simulate blade and vane cooling flow and the hub and casing leakage. The total number of mesh points is equal to 12 millions, the turbulence model is Michel's one and the flow is assumed to be full turbulent. A good agreement between measured exit conditions and computed flow is got: around 0.5 % for total pressure (figure 6a), around 1% for total temperature (figure 6b) and around 2° for exit swirl (figure 6c). The most important discrepancy is observed at hub and casing. This is attributed to the simulation of the leakage flow mainly in HP turbine but also in LP turbine.

4.2.2 Deterministic stress approach

Easy to implement and cost effective, mixing plane approach does not simulate accurately

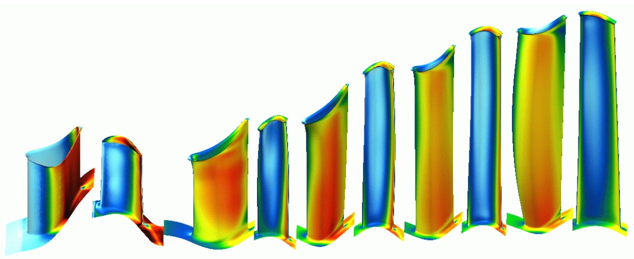

Figure 5 : HP & LP Turbine – Steady Multi-Stage Calculation - Mixing Plane Approach

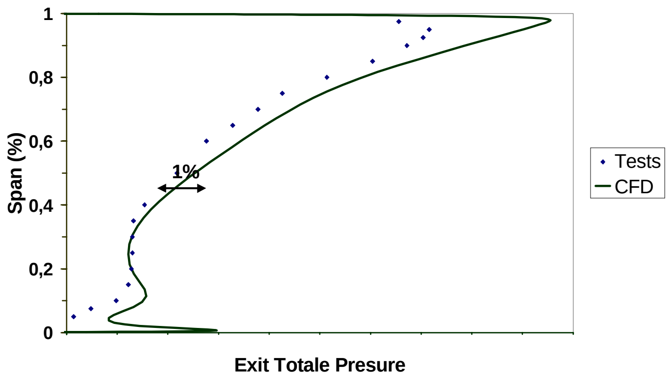

Figure 6a : HP & LP Turbine Multi-stage Computation : Exit Totale Pressure Comparison

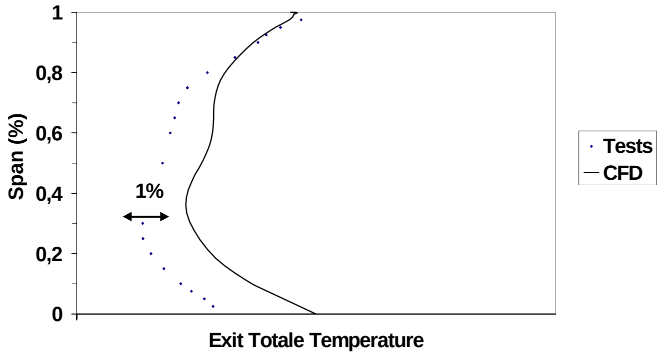

Figure 6b : HP & LP Turbine Multi-stage Computation : Exit Totale Temperature Comparison

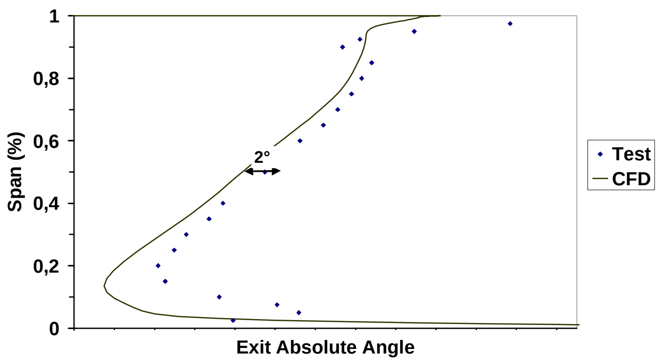

Figure 6c : HP & LP Turbine Multi-stage Computation : Exit Absolute Angle Comparison

flow disturbances coming from adjacent rows such as wakes, shocks or potential pressure fields and their unsteady interactions.

An approach first exposed by Adamczyk [21], proposes to account for the « average » contribution of the temporal and passage-to-passage flow perturbations in a steady, periodic from blade passage to blade passage, multi-stage flow model: the so-called « average-passage » flow equation system. Although the derivation of this model is mathematically rigorous, its interest is practically limited by the difficult task of estimating the new terms in the equation system, i.e. the « deterministic stresses ».

A simplified approach proposed by Rhie [22], neglects the passage-to-passage flow variations and calculates the values of the deterministic stress terms using a steady representation of blade row interaction. This approach has recently been introduced in the CANARI code under a collaborative research project with Ecole Centrale Lyon, ONERA, TURBOMECA and Snecma [23]. As illustrated on Figure 7 for the case of a transonic HP turbine, the deterministic stresses are calculated using spatial (i.e. tangential) averages on overlapping meshes where axi-symmetric bodyforces have been applied to account for the potential effect between closely coupled rows. During the computation, both the deterministic stress terms and the body forces are exchanged from one blade computational domain to the other.

Although it is more CPU expensive than the mixing plane approach, this approach presents some valuable advantages. It is a continuous interface plane approach as, by definition, the contribution of deterministic stresses restores the continuity of tangentially averaged flow properties across interface planes. Moreover, it simulates the average wake blockage and the steady mixing effects that are believed to be of primary importance for the matching of blade rows. This method has be compared to an averaged unsteady computation (figure 8). Agreement between two results is quite pretty, however, purely unsteady effects such as wake or shock wave chopping by neighbouring blade rows are clearly neglected, in many case this method is clearly an improvement of the mixing plane approach for a cost cheaper than unsteady CFD

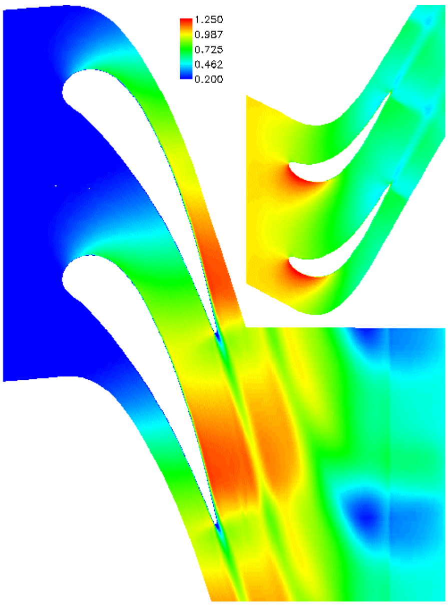

Figure 7 : HP Turbine Stage - Multi-Blade Calculation - Deterministic Stresses Approach (Rhie)
Steady state Mach number

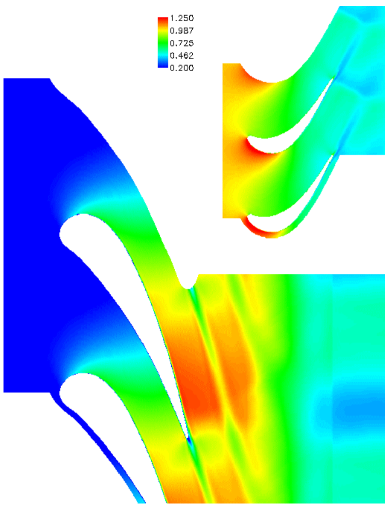

Figure 8: Unsteady multi-blade computation of an HP single stage turbines: Averaged Mach number

.5. Unsteady analysis

5.1 HP and LP Turbine Interaction

Periodic unsteadiness is inherent to flows in turbomachine, generally steady-state approach provide good accuracy to design turbine and compressor (§ 4). Unfortunately in some case unsteady computation must be done: aeromechanical phenomenon such as forced response, flutter or hotter and cooler gases segregration or rotor-stator interaction can only be predicted by using time-accurate analysis. In order to study interaction between a HP turbine and an LP turbine a quasi-3D computation has been conducted on a stream surface around midspan allowing a varying stream tube thickness (figure 9). In order to reduce the computation time a spatio-temporal periodicity method developed by Fourmaux [14] is applied. Such method allow a quick convergence without any modification of the airfoil counts or geometry. Unsteady numerical results in particular unsteady total pressure and unsteady static pressure are in good agreement with test data. Consequently, this code can be used with confidence in the design process to optimise the geometry to reduce the aerodynamic excitation on each airfoils.

Some 3D computation using CANARI solver have been carried out by ONERA [15]. In spite of interesting results, these computations are nowadays limited to single stage configuration due to CPU time limitation (figure 10).

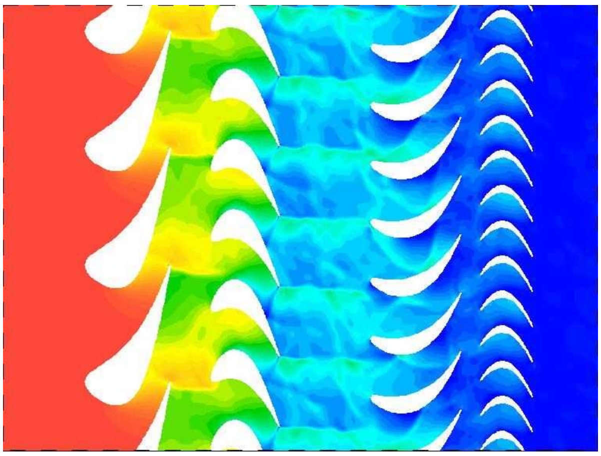

Figure 9 : Unsteady Multi-stage computation of HP & LP turbines

5.2 Blade flutter prediction

Blade flutter is a major concern for the safety of compressor operation [16]. An unstable coupling between the blade eigenmodes and the resulting flow unsteadiness induces it. This phenomenon is well known on aircraft wings and helicopter blades. The origin of this excitation can be flow separation at the leading edge on suction side when a blade operates at high incidence and low rotational speed (subsonic flutter) or shock motion when inlet flow is supersonic (supersonic flutter). Another type of flutter can come from shock motion in a choked configuration (choke flutter)

5.2.1 Supersonic flutter

To study supersonic flutter, a methodology has been set up based on mechanical analysis software and unsteady 3D Euler code. This process has been extensively brought up by Gerolymos [23] and Burgaud [24].

Vibration analysis is carried-out on a whole-bladed disk accounting for cyclic symmetry, providing airfoil motion and eigenfrequencies. The resulting blade displacement distribution for a given mode is input in an unsteady 3D Euler code. The unsteady calculation (assuming chorochronicity) is performed on a single blade passage for different wave number (figure 11). The

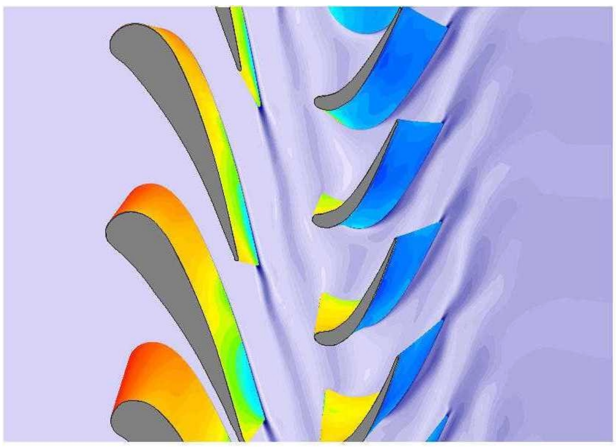

Figure 10 : 3D CANARI HP Turbine unsteady computation

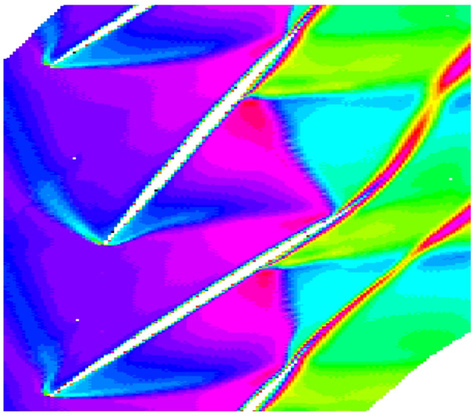

Figure 11 : Euler 3D unsteady supersonic flutter analysis Instantaneous Mach number

unsteady pressure and motion are then time-integrated over a natural period to provide mean power distribution that is summed over the whole blade to yield the aerodynamic damping parameter (figure 12).

This methodology has been validated on different research fan blades. It is a part of Snecma fan design process to anticipate any potential risk of supersonic flutter early in the project.

5.2.2 Subsonic flutter

Unlike supersonic flutter, subsonic flutter involves separation of the boundary layer on the blade suction surface near the leading edge. In consequence, it requires the use of unsteady Navier-Stokes computations with a complex flow phenomenon.

The approach recently developed by Snecma is based on a two equations k-ε turbulence model with a low Reynolds model near the wall. This approach is currently being validated against 2D cascade data representative of the tip section of a wide chord fan at part speed conditions (inlet Mach number of 0.7 and 6° positive incidence). Figure 13 shows the Mach number contours around the blade airfoil as computed by the 2D Navier-Stokes code with a k-ε turbulence model: a large flow separation initiates at the leading edge and extends over 10% of the airfoil chord on the suction surface. The real and imaginary parts of the unsteady static pressure coefficients on the airfoil suction

Figure 12 : 3D Euler unsteady supersonic flutter analysis : Mean power distribution

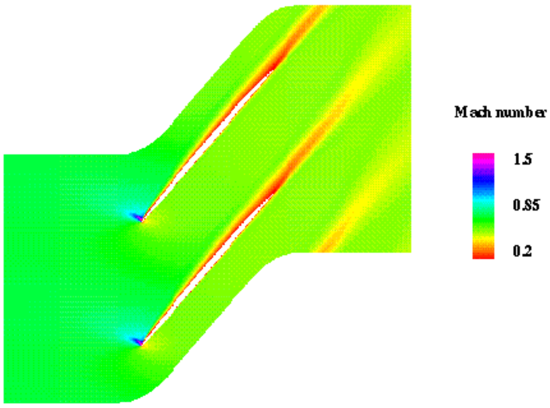

Figure 13 : 2D Navier-Stokes - k-ε turbulence model - Mach number contours

and pressure sides are computed to determine the limit of instability in term of pressure ratio and rotational speed.

The current approach taken by Snecma is to use this newly developed numerical analysis in support of the available empirical criteria as the validity of these criteria is generally limited to the range of existing configurations.

6. Conclusions

This paper has presented the latest advances in the aerodynamic design and analysis process of fans, compressors and turbines at Snecma. The main conclusions of the paper can be summarised as follows:

- The role of CFD in the design procedure is fast growing. New tools have been developed allowing the treatment of numerous difficult problems as close as possible to the reality: tip clearance, technological effects, and multistage effects, unsteady phenomena (interaction effect, forced response, aeroelastic and aerodynamic instabilities).
- Designers also dedicate a great effort to turn numerical methods into integrated tools which are easy to handle. The aim is to create a « user friendly » environment to enable the designer to focus primarily on the physical analysis of numerical results.
- Experimental investigations of research turbine and compressors with a comprehensive and high quality set of measurements are essential to produce an appropriate database for the validation and calibration of advanced numerical methods.

In spite of good agreement between numerical results and tests data, some improvements are needed to be able to design the next generation of aircraft engine:

- with less stage, less airfoils
- more efficiency
- more affordable
- design in reduce time

These challenging objectives could be reach by improving our tools in term of transition and turbulence description, unsteady flow simulation and by using aero-thermal-mechanical optimisation of compressor and turbine.

Acknowledgements

The financial support of the French SPAé (« Service des Programmes Aéronautiques ») is fully acknowledged. Snecma CFD specialists are also indebted to ONERA and Snecma's « External Research Laboratories » (LEMFI , ECL, VKI) for their contribution to CFD developments and to the testing of research compressors and turbine. must be emphasised that the results presented in this paper have been obtained by many co-workers in the Snecma Compressor and Turbine Departments. The authors also wish to thank the technical direction of Snecma for the permission to publish this paper.

References

[1] KARADIMAS G.: The Position of the Unsteady Flow Computation in the Compressor and Turbine Design and Analysis Process. AIAA 92-0015, Reno, January 1992

[2] FALCHETTI F., THOURAUD P.: Methodology for Advanced Core Compressor Design. European Propulsion Forum, Conference Proceedings 90.014, DGLR-Bericht 90.01, pp 113-121, 1990

[3] JOUBERT; H., GOUTINES, M.: Use of CFD Methods to Design Engine Nacelles. ASME 93-GT-117, Cincinnati May 1993

[4] BROCHET, J. : Aerodynamic Design of the CFM56-5C fan, The Leading Edge, GE Aircraft Engine publication, April 1993

[5] VUILLEZ, C., PETOT, B.: New Methods, New Methodology, Advanced CFD in the SNECMA Turbomachinery Design Process. AGARD Lecture Series on « Turbomachinery Design using CFD », May to June 1994

[6] LERAT, A., SIDES, J., DARUT, V. : An Implicit Finite-Volume Method for Solving the Euler Equations, Lecture notes in physics, Vol. 170, pp 343-349, 1982

[7] LERAT, A. : Implicit Methods of Second Order Accuracy for the Euler Equations, AIAA Journal, January 1985

[8] MICHEL, R., QUEMARD, C., DURANT, R. : Application d'un Schéma de Longueur de Mélange à l'Etudes des Couches Limites Turbulentes d'Equilibre, ONERA NT N°154, 1969

[9] LOMAX H., BALDWIN B.S. : Thin layer approximation and algebraic model for separated turbulent flows. AIAA Journal, n° 78-257, 1978

[10] ALLMARAS S., SPALART P. : A one-equation turbulence model for aerodynamic flows, AIAA Journal, n° 92-0439, 1992

[11] LAUNDER, B.E., SHARMA, B.I. : Application of the Energy Dissipation Model of Turbulence to the Calculation of Flows near a Spinning Disk. *Lett. Heat Mass Transfer*, Vol. 1, pp. 131-138, 1974

[12] ABU-GHANAM B., SHAW, R. : Natural Transition of Boundary Layers, The Effects of Turbulence, Pressure Gradient and Flow History. Journal of Mechanical Engineering Science Vol. 22, pp 213-228

[13] LIAMIS N., BRISSET C., LACORRE F., DUBOUE J-M.: CFD Analysis of Dual Spool Turbine Configuration AIAA 1999

[14] FOURMAUX A., DUBOIS L. : Unsteady flow computation in a double shaft turbine 14 th International symposium on Air Breathing Engine ISOABE Florence Italy 1999

[15] SGARZI O., TOUSSAINT C. : A parallel computation of a 3D unsteady flows in a full stage of a transonic turbine : The TUMULT project. 14 th International symposium on Air Breathing Engine ISOABE Florence Italy 199

[16] JOUBERT H.: Supersonic Flutter in Axial Flow Compressors. Unsteady Aerodynamic s of Turbomachines and Propellers , Cambridge UK 1984

[17] VALLET, I. : Aérodynamique Numérique 3D Instationnaire avec Fermeture Bas-Reynols au Second Ordre, thesis, University of Pierre et Marie Curie, Paris, 1995

[18] GEROLYMOS G., VALLET, I. : Implicit Computation of the 3D Compressible Navier-Stokes Equations using k-ε Turbulence Closure, AIAA Journal, Vol. 34, 1996

[19] ESCURET J.F., VEYSSEYRE Ph., VILLAIN M., SAVARESE S., BOIS G., NAVIERE H. : Effect of a Mismatch between the Buttons of Variable Stator Vanes and the Flowpath in a Highly Loaded Transonic Compressor, ASME Paper 97-GT-471, 1997

[20] LIAMIS N., DUBOUE J.M. : CFD Analysis of High Pressure Turbines, Paper accepted to ASME98, 1998

[21] ADAMCZYK, J., : Model Equation for Simulating Flows in Multistage Turbomachinery, ASME Paper N°85-GT-226, 1985

[22] RHIE, C., GLEIXNER, A., SPEAR, D., FISCHBERG, C., ZACHARIAS, R. : Development and Application of a Multistage Navier-Stokes Solver - Part I, ASME Paper 95-GT-342, 1995

[23] BARDOUX, LEBOUEF F.,DANO C. TOUSSAINT C. Characterisation of deterministic correlations for a turbine stage ASME Paper 99 GT 100 and 101 Indianalpolis 1999

[24] GEROLYMOS, G.A. : Advances in the Numerical Integration of the 3D Euler Equations in Vibrating Cascades, ASME Paper N°92-GT-170, 1992

[25] BURGAUD, F., BERTHILLIER, M. : The Contribution of Numerical Methods in the Aeroelastic Analyses of Turbomachinery Fans and Compressors, ECCOMAS, 1996