AERODYNAMIC DESIGN AND 3D NAVIER-STOKES ANALYSIS OF A HIGH SPECIFIC FLOW FAN

F. Falchetti, H. Quiniou, and L. Verdier
Department of Compressor Aerodynamics
Sneccma
Moissy Cramayel, France

ABSTRACT

Advanced research is currently being conducted at Sneccma to design a swept wide chord fan blade with a very high specific flow value of 230 kg/s/m² at the aero design point instead of 212 kg/s/m² on current turbofan engines.

This concept has the potential benefit of higher thrust than conventional engines for a given value of the fan diameter. This induces potential advantages in terms of performance (reduced nacelle drag) and installation under the wings.

These advantages are subject to the condition that we retain high fan bypass efficiency with axial Mach number close to 0.8. The intensive use of 3D Euler and 3D Navier-Stokes blade to blade computations in the aero design makes it possible to overcome this difficulty.

This paper describes the aero optimization of a swept fan blade with a specific flow of 230 kg/s/m², a corrected tip speed of 350 m/s and a bypass pressure ratio of 1.45. Applications of the most recent numerical methods are presented, including 3D Navier-Stokes blade to blade computations.

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>H</td>
<td>Fraction of blade height</td>
</tr>
<tr>
<td>M</td>
<td>Mach number</td>
</tr>
<tr>
<td>P/P</td>
<td>Total pressure ratio</td>
</tr>
<tr>
<td>P_t</td>
<td>Total pressure</td>
</tr>
<tr>
<td>T_t</td>
<td>Total temperature</td>
</tr>
<tr>
<td>U_e</td>
<td>Reduced tip speed</td>
</tr>
<tr>
<td>n_is</td>
<td>Isentropic efficiency</td>
</tr>
</tbody>
</table>

Introduction

The design of advanced turbofan engines is driven by demands for greater power, minimum size and weight, reduced fuel consumption, pollutant emission and noise level, high reliability and low operating and maintenance costs. The engineering task is to achieve continuous improvements in all the components of an engine, both for new and derivative designs. Among all components, the fan is a particularly important one for high bypass ratio engines.

Fan aerodynamic efficiency directly affects specific fuel consumption of the engine, and the flow rate capability induces the diameter of the nacelle. The reliability of the engine depends on the fan ability to withstand the consequences of foreign objects ingestion (bird, hail,...), and to operate in all flight conditions without stall or flutter problems. Reduction of fan noise also contributes to a quiet operation of the engine, thus to a better environment and comfort of airlines passengers.

Until the late 70's, improvements of fan technology were achieved by carrying out a long and expensive experimental work, including aerodynamic, structural dynamic and acoustic tests, on compressor rig and on full engine, ground tests as well as flight tests. The rapid evolution of computer technology, together with progress in computational fluid dynamics, have played a vital role in changing this situation. 3D and viscous flow computations permit the performance of the fan to be optimized before any metal is cut.
For several years, Snecma has devoted major effort to new design and analysis codes for improving their aero design methodology. Computation methods are essential tools for the fan aero design and much of today's initial testing is effectively carried out by computer. This allowed Snecma to design fans with exceptionally good performance (fig.1), including shrouded fans such as CFM56-5A, and -5C, wide chord research fans such as Snecma TS27, as well as counterrotating high speed propellers for the UDF engine. Thanks to the extensive use of computation methods, the development of these fans was also very rapid, thus reducing engine development timescales (Karadimas, 1989).

Further improvements of fan performance can be achieved by using upgraded CFD methods. Besides the benefits obtained from more accurate predictions of the complex flows characteristics, other improvements can be obtained by a better integration of the different disciplines (aero-mechanics, aero-acoustics...) that are involved in the optimization of a fan. From another point of view, the use of advanced CFD permits to explore the feasibility of new design concepts within very short timescales.

Such an advanced design concept will be presented in this paper, which deals with the aerodynamic design of a swept blades, high specific flow, wide chord fan. This work was conducted at Snecma as a part of advanced studies for future high bypass ratio engines. The major objective of this preliminary design was to prove the aerodynamic feasibility of a large increase in fan specific flow, that is to design a fan with minimum diameter for a given flow rate.

Snecma state of the art of fan performance

The power of Snecma commercial fan technology and aero design methodology was demonstrated by performance levels that were obtained by research wide chord fans (TS27) as well as CFM56 engine fans (-5A and -5C2) or by counterrotating high-speed propellers (GE36) (Nicoud, Brochet and Goutines, 1989).

The measured bypass isentropic efficiency reached excellent levels for all operating points. Moreover, constant improvements were achieved along the years, with bypass efficiency levels growing from .895 on CFM56-5A fan (1987) up to .907 on -5C2 (1990). The measured bypass efficiency of these fans is represented on Fig.1 for the full operating line versus reduced tip rotating speed.

Expected Benefits of High Specific Flow Fan

Current fan technology assumes a maximum value of the inlet specific flow at design point of 210-215 kg/m²/s. This corresponds to a mean value of inlet axial Mach number M = 0.65. Passing over this limit would permit to decrease the fan diameter for a given
mass flow. Large payoffs can be expected specially for high bypass ratio, high thrust engines as those required for tomorrow's large transport aircrafts.

Between current fans that have a small potential of specific flow increase and high speed propellers that operate at 0.85 flight Mach number with very good aerodynamic efficiency, there is a potential of 5 to 10% increase in fan specific flow. Such improvements are expected by associating the benefits of wide chord technology and both axial and tangential sweep of the blade.

Hence one may expect up to 5% saving in fan diameter for a given massflow. As far as the nacelle external shape is driven by the fan diameter, the corresponding reduction of engine diameter leads to expect weight and drag reduction, as well as easier installation on the aircraft. The acoustic of such an engine will also benefit of leading edge sweep of the blades, which is supposed to reduce the noise level.

**Specifications of the project**

This project is identified under the generic name TS31. In order to test the aerodynamic feasibility of very high specific flow fans, we chose the specifications of this preliminary design project to be very close to the physical limits. The objective was to design a fan with an inlet specific flow of 230 kg/m²/s at aero design point, taking into account a need for 1.5% increase in massflow at maximum overspeed as well as 1% endwall blockage factor. This corresponds to a maximum mean value of axial Mach number $M = 0.78$ at design point (Fig.3).

This work was conducted as a part of Snecma advanced studies for high bypass ratio engines, and we selected thermodynamic specifications from typical projects for future 40K engine. The aero design point for TS31 fan was chosen between Cruise and Max Climb operating points, Max Climb conditions being expected for 1.05 overspeed and 1.5% increase in flow rate.

The corresponding specifications for aero design point are as follows:

- Pressure ratio: 1.45
- Bypass ratio: 8.85
- Reduced tip speed: 350 m/s

**Aero Design Methodology**

A complete overview of Snecma fan design methodology was presented by Karadimas (1988). The typical calculation procedure is illustrated in Fig. 4.
A geometric method for profiles with supersonic inlet Mach number. After designing all airfoil sections, a stacking of these sections provides the complete volume of the blade. When the blade shape is defined, major aerodynamic criteria are checked, such as throat margin, throat location, diffusion level... First analyses are performed using 3D Euler (Vivian and Veuillot, 1978; Brochet, 1980) and quasi 3D Navier-Stokes (Vuillez and Veuillot 1990) computations for several operating conditions i.e. different values of throttling and rotation speed. Quasi 3D Navier-Stokes analysis of airfoil sections gives a good estimation of viscous losses and deviation. This approach needs very short cpu times on a super computer, and is used extensively during the iterative process for aero-mechanical optimization of the fan. Near the end of the optimization procedure, 3D Navier-Stokes computations are performed in order to provide a very accurate prediction of the fan performance. Results an compared with the design objectives, such as spanwise variations of inlet and outlet flow angles, pressure ratio, efficiency,... and the geometry of the blades is modified until objectives are matched.

The next sections will describe the design procedure of the TS31 fan, and present a prediction of the performance.

Throughflow calculation

The throughflow calculation is built on a definition of the flowpath that is representative of a single stage research fan to be tested on a compressor rig. It includes the fan rotor, flux splitter, outlet guide vane (OGV) as well as booster inlet guide vane (IGV). A meridian view of the flowpath is shown on fig.5, including the computation mesh that is adapted to the swept rotor blades. Both axial and tangential sweep of the blades are taken into account by the calculation, which modify the streamline distribution and radial equilibrium of the flow.

One of the main difficulty for the definition of the flowpath geometry was to keep moderate inlet Mach number on IGV and OGV despite the high value of specific flow at the rotor inlet. This was achieved by designing the hub of the fan with a very low conicity. The mean slope is 14° against 20-25° for the hub of current Snecma wide chord fans.

The analysis of OGV and IGV inlet conditions shows maximum values of inlet Mach number that are respectively M = 0.7 at OGV midspan and M = 0.84 for IGV near hub sections, which are representative of the current technology.

Geometrical characteristics of the different blades are introduced as input data of the throughflow computation, taking into account structural specifications. The blade numbers of IGV (84) and OGV (56), as well as fan/OGV axial spacing are accounting for acoustic requirements. The main characteristics of the fan blades are given in fig.6.

Throughflow calculation

The throughflow calculation is built on a definition of the flowpath that is representative of a single stage research fan to be tested on a compressor rig. It includes the fan rotor, flux splitter, outlet guide vane (OGV) as well as booster inlet guide vane (IGV). A meridian view of the flowpath is shown on fig.5, including the computation mesh that is adapted to the swept rotor blades. Both axial and tangential sweep of the blades are taken into account by the calculation, which modify the streamline distribution and radial equilibrium of the flow.

One of the main difficulty for the definition of the flowpath geometry was to keep moderate inlet Mach number on IGV and OGV despite the high value of specific flow at the rotor inlet. This was achieved by designing the hub of the fan with a very low conicity. The mean slope is 14° against 20-25° for the hub of current Snecma wide chord fans.
Blade design

The blade geometry is defined by designing seven airfoil sections. The three hub sections are designed using a quasi 3D inverse method that was developed jointly by ONERA and Snecma (Jacquotte, 1989; Nicoud, Le Bloa and Jacquotte, 1991). This method solves the equations for quasi 3D compressible potential flows, which take into account rotation speed of the rotor, together with radius and streamtube thickness variations. The airfoil section geometry is calculated from given chordwise distributions of suction side and pressure side flow Mach number together with given inlet flow angle and Mach number. This method has proved good performance when designing profiles with subsonic inlet Mach number and suction side peak Mach number up to 1.35 (Nicoud, Le Bloa and Jacquotte, 1991). It permits to limit the maximum relative Mach number on the suction side and to control the deceleration on the blade.

The four tip profiles are designed using a Snecma proprietary aero-geometric method. The control parameters of the geometry are linked to aerodynamic properties of supersonic flows such as oblique shock angles, characteristics lines, throat location, .... Considering the large sweep of the blade in the tip region that have strong interaction with the casing, two profiles were designed within the last 20% of blade height in order to get a better control of the shock position in this region. Some structural specifications and aerodynamic parameters must be specified such as

- leading edge thickness
  - including bird ingestion requirements (2.5 lbs)
- maximum thickness
- trailing edge thickness
- chord length
- suction side incidence
- downstream deviation
- throat margin

The shape of the blade was optimized in order to account for the severe structural requirements of such swept blades. Tangential sweep is responsible for larger radial displacement between 0 and 100% rotation speed. Forward axial sweep is responsible for very high static stress in the hub and leading edge region. Several aero-mechanical iterations were necessary in order to get the final volume that fulfill both structural requirements and aerodynamic objectives.

The final blade (fig.8) keeps axial sweep of the leading edge, the benefits of which are a higher flow capacity and reduction of shock strength, while the trailing edge was moved for structural reason until there was no axial sweep. The tangential sweep of the blade was controlled in order to minimize static stresses and displacements. The axial and tangential coordinates of the stacking axis are given in Fig.9. 3D structural analyses of the blades were performed for two different materials. Fig.10 shows a short summary of the results obtained with solid titanium blades or carbon-carbon composite blades.
3D Euler analysis

As 3D viscous or unsteady computations need very large computing power, thus being rather expensive methods, their use is limited to the last steps of the design procedure. However, most of the relevant information on the transonic flow properties around fan blades can be obtained by solving the 3D Euler equations for steady flows. The 3D Euler solver was developed jointly by ONERA and Snecma [4,5] and Snecma has more than ten years experience in using this code for fan and compressor design. Thanks to recent progress both in CFD techniques and in computer technology, such computations are today easy to use. With a typical 22000 points mesh as used for TS31 blade to blade computations, the solution is reached within five minutes cpu time on current supercomputers. This makes 3D Euler a cheap and efficient tool when the aeromechanical optimization of the blade requires large and numerous modifications of the geometry. In the particular case of the TS31 swept blades, a large amount of work was necessary in order to get a blade geometry that satisfies the severe structural requirements together with aerodynamic objectives. The major aerodynamic difficulty for such low tip solidity blades is to control the stability of the shock surface location and avoid unstarted operating conditions at nominal rotation speed as well as part speed. Several 3D Euler computations are run for different values of rotation speed and throttling conditions. The Euler solution do not account for viscous effects, but provides the relevant information about the flow capacity of the fan as well as starting margin along a speedline. The shock location for aero design conditions is shown in Fig.11 that represents contours of constant relative Mach number on both pressure side and suction side of the TS31 final design blade.

The rotor performance map given in Fig.12 represents the variation of flow rate and total pressure ratio along three speedlines corresponding to 97%, 100% and 105% of design rotating speed. Considering that the values must be corrected for viscous effects that are not accounted for by Euler equations, these results match the design objectives of flow capacity as well as started margin. In particular, the 3D Euler results shows 1% increase in flow rate for 5% overspeed. The computations were performed for the same 3D geometry of the blade as for design point, which do not account for structural untwist of the blade at overspeed. This deformation is expected to be responsible for 0.5% additional increase in massflow, which then matches the objective of 1.5% total increase at 5% overspeed.

<table>
<thead>
<tr>
<th>MATERIALS</th>
<th>TITANIUM</th>
<th>CARBON - CARBON COMPOSITE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass</td>
<td>15.3 kg</td>
<td>5.3 kg</td>
</tr>
<tr>
<td>Tip displacement T.E.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Radial</td>
<td>2.0mm</td>
<td>1.8mm</td>
</tr>
<tr>
<td>Tangential</td>
<td>3.6mm</td>
<td>17.2mm</td>
</tr>
<tr>
<td>Axial</td>
<td>-5.0mm</td>
<td>-9.4mm</td>
</tr>
<tr>
<td>Tip displacement T.B.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Radial</td>
<td>1.5mm</td>
<td>-2.8mm</td>
</tr>
<tr>
<td>Tangential</td>
<td>-4.2mm</td>
<td>7.1mm</td>
</tr>
<tr>
<td>Axial</td>
<td>4.0mm</td>
<td>1.0mm</td>
</tr>
<tr>
<td>Maxi Stress</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pressure side</td>
<td>613 KPa</td>
<td>320 KPa</td>
</tr>
<tr>
<td>Suction side</td>
<td>385 KPa</td>
<td>298 KPa</td>
</tr>
<tr>
<td>Permissible stress</td>
<td>900 KPa</td>
<td>1500 KPa</td>
</tr>
</tbody>
</table>
An artificial dissipation based on the work of Jameson, Schmidt and Turkel (1981) is added to ensure the stability of the scheme and to capture correctly the flow discontinuities in the inviscid flow regions. It uses a sensor combining pressure and velocity gradients to detect shock waves and contact discontinuities.

Local time stepping is used to accelerate the time integration process, as well as an implicit residual smoothing procedure proposed by Lerat (1985). This greatly increases the stability domain of the explicit scheme and thus allows much larger time steps.

As a matter of fact, reaching a $10^{-3}$ reduction of residuals and a flow rate conservation better than 0.01% requires a maximum of 3000 iterations, compared to 15000 iterations when using the explicit scheme and local time-stepping alone. Boundary conditions are imposed through compatibility equations obtained from characteristic relations.

Validation of 3D Navier-Stokes Computations

This code was calibrated on 3 different wide chord fan blades with different pressure ratio (1.5 - 1.7) different corrected tip speed (370 - 413 m/s) and different size (22" - 61"). Detailed results of this calibration were presented by Couaillet, Veysseyre and Vuillot (1991). This section presents a summary of this work that gives confidence in the results obtained for the TS31 rotor.

The research fan TS27 (Fig.13) was tested at Snecma in 1985 and extensive test results are available such as classical traverse data downstream of rotor and OGV, as well as L2F anemometry data inside the rotor blade passage. The measured performance of this rotor at nominal conditions are the following:

- Pressure ratio: 1.65
- Specific massflow: 208 kg/s/m²
- Relative tip speed: 400 m/s
- Bypass efficiency: .91

Fig.13 : Snecma Research Fan TS27
A comparison of test and 3D Navier-Stokes computation results is shown in Fig.14 and 15. The presented results were obtained using a 613000 nodes computation grid, inlet conditions derived from test results as well as downstream static pressure. A slip boundary condition is used on the casing, i.e. without accounting for tip clearance effects.

A comparison of constant relative Mach number inside the blade passage is presented in Fig.14 for a 95% of blade height airfoil section. The predicted shock location shows a good agreement with L2F anemometry measurements.

![3D Navier Stokes](image1)

Fig.14 : Contours of relative Mach number 95% of blade height section

Pichwise averaged values of downstream total pressure and total temperature versus blade height are compared in Fig.15. 3D Navier-Stokes results are exceptionally close to the measurements obtained with classical traverse probes, except in the tip region affected by clearance flows that were not taken into account by the presented computation. The momentum averaged rotor efficiency obtained from computed results is 0.936 against 0.931 from traverse measurement data. The accuracy of 3D Navier Stokes computation and the calibration made on various fan blades enable us to predict fan flow capability within 0.5%.

![3D Navier Stokes Analysis of TS31 Fan](image2)

3D Navier Stokes Analysis of TS31 Fan

The computational grid used for TS31 rotor analysis is a multidomain H-O-H mesh with a total of 600 000 nodes with 69 in the radial direction and 159x53 in the O mesh around the blade. Partial view of the grid is shown in Fig.16, together with enlarged views of the leading and trailing edges for a midspan section to illustrate the resolution in the representation of these regions.

![Leading Edge](image3)

Fig.16 : Computational grid mesh (600 000 points)

Inlet profiles of total pressure, total temperature and flow angle are considered as boundary conditions and treated via compatibility relations. Inlet total temperature and flow angle are uniform, and the total pressure gradient near the casing is representative of experimental conditions. At exit, the static pressure is fixed at mid-span, and combined with radial equilibrium conditions across the exit plane. No slip conditions are used on blade and hub surfaces. In this preliminary design computation, the tip clearance is not taken into account and a slip condition is used on the casing surface.

The converged solution presented in this paper was obtained within 3000 iterations, and less than 5 hours of CRAY Y/MP cpu-time. Contours of constant static pressure on the suction side and downstream of the blade are represented in Fig.17. The comparison with Euler results (Fig.11) confirms the adequacy of using 3D Euler in the first steps of the blade optimization methodology in order to control the shock location and stability for such transonic fan rotors.
However this flow image for both Euler and Navier Stokes calculation can be different due to tip clearance effect. Indeed the viscous phenomena induced near the blade tip can interact, more or less with the blade shock configuration and affect the efficiency and the flow rake.

The analysis of computation results predict the following characteristic of this rotor:

<table>
<thead>
<tr>
<th>3D N.S.</th>
<th>Objective</th>
</tr>
</thead>
<tbody>
<tr>
<td>Specific flow (kg/s/m²)</td>
<td>231</td>
</tr>
<tr>
<td>Pressure ratio</td>
<td>1.49</td>
</tr>
</tbody>
</table>

Accounting for the correction deduced from 3D Navier Stokes calibration and a realistic type clearance effect, a specific flow of 227 to 228 kg/s/m² can be estimated which is lower by 1% than the objective. Snecma is developing a 3D Navier Stokes method taking into account tip clearance effect (100 x 50 x 15 mesh cells) that is available in 1994 and is in the process of validation to be introduced in design procedure. On fig.18, objective and 3D Navier Stokes total pressure gradient are compared. 3D Navier Stokes computation is more throttled than the objective.

The predicted momentum averaged rotor isentropic efficiency is 0.91, which is a very good level for such a high value of inlet specific flow. A comparison with classical fan technology is presented in Fig.19, where we plotted the typical evolution of rotor efficiency along the fan operating line versus the corresponding specific massflow, together with the predicted position of TS31 at aero design point.

**Conclusions**

A preliminary design of Snecma TS31 swept wide chord fan blade demonstrates the feasibility of increasing the inlet specific flow of commercial fans with good aerodynamic performance. The design objectives are matched by associating the benefits of wide chord technology and both axial and tangential sweep of the blade. Structural requirements such as maximum local static stress and bird ingestion capacity are accounted for and satisfied. 3D Navier-Stokes analysis shows the flow capacity of this fan matches the design objective. This corresponds to an inlet specific massflow of 230 kg/s/m² at aero design point, with another 1.5% increase at 1.05 overspeed, which is very close to the physical limit before choke. This extreme flow capacity is achieved with a predicted rotor efficiency of 0.91 and sufficient started margin at design point, which is far ahead from current fan technology.
Nevertheless, some questions are not answered by the present study. The major one concerns the part speed performance of such a fan. It is specially important to insure that increasing the flow capacity of the fan will not affect the overall performance of the engine at cruise conditions. Further studies are under progress at Snecma to adress this point. It is expected that the optimal value of fan specific flow should be slightly decreased and adapted to each particular engine project. A more complete study is carried out to design a fan blade that allows to increase engine thrust using the same nacelle and outlet guide vane. For this application, specific flow objective has been decreased to 224 kg/s/m². This blade should be tested in early 1995.

Another remaining question when going to a real application on an engine is the aeroelastic behaviour of such swept blades. This problem was not addressed during the presented study, but will be a cornerstone of the next design studies before running scale tests of the Research Fan TS31.

The successful achievement of such ambitious objectives relies on prediction capability of advanced CFD methods. The use of 3D Navier-Stokes computation represents a major step in upgrading Snecma aero design methodology. We are confident this will permit Snecma to improve fan technology for future aircraft engine with the same success as those obtained on CFM56 engine fans.

Acknowledgements

The 3D Navier-Stokes solver was developped at ONERA with financial support of the French DRET. The authors also wish to thank Snecma for permission.

References


