A Methodology Proposal to Design and Analyse Counterrotating High Speed Propellers

D. NICOUJ, J. BROCHET, M. GOUTINES
SNECMA Villaroche, 77550 Moissy-Cramayel, France

ABSTRACT

Contrarotating high speed propellers are able to significantly reduce fuel consumption of high subsonic aircrafts. The achievement of this goal requires the optimization of the transonic flowfield on the blades in order to obtain high efficiency. For several years, 2D and 3D aerodynamic computational methods have been used to design high performance turbofans. A similar methodology can be developed for high speed propeller design, and this paper presents a typical application of such methods. We first present an application of the through-flow method. An outer fictitious casing is chosen in order to simulate undisturbed flow far from the propellers, and the mesh is adapted to the high swept blades. Radial distribution of loading is selected using aerodynamic criteria.

Then, a quasi geometrical method supplies the bidimensional profiles accounting for structural specifications such as chord length, maximum thickness and root attachment. Suction side incidence and downstream deviation are also specified. After the profile stacking operations, which use conformal application on the axisymetric stream surfaces, the tridimensional transonic flowfield is drawn by a 3D Euler solver on an appropriate domain. This code uses a multi-domains technique and includes the energy equation for non-constant rothalpy cases. Particular interest is focused on the Mach number distributions and on the shock strength. The final loss prediction is made by means of a shock loss model and a bidimensional boundary layer calculation based on the Euler static pressure distributions. The profile shapes are modified and the above process is repeated until the required deflection, a convenient throat margin, and sufficiently thin and well attached boundary layers are obtained. Finally, the global performances are issued from 3D Euler and boundary layer computations, completed by the calculation of secondary flow effects.

NOMENCLATURE

\begin{align*}
M & \quad \text{Mach number} \\
Pt & \quad \text{stagnation pressure} \\
p & \quad \text{static pressure} \\
Z & \quad \text{altitude} \\
\beta & \quad \text{relative angle} \\
\Delta h & \quad \text{specific enthalpy increase per blade row} \\
\omega & \quad \text{loss coefficient, percent} \left( \frac{Pt_2 - Pt_1'}{Pt_1 - p_1} \right) \times 100 \\
^1 & \quad \text{at inlet} \\
^2 & \quad \text{at outlet} \\
is & \quad \text{relative to an isentropic evolution} \\
^' & \quad \text{relative to a blade row}
\end{align*}

1. INTRODUCTION

Several recent studies on transport aircraft propulsion point out that advanced high speed propellers offer a large performance improvement (Mikkelsen et al., 1984 ; Smith, 1987 ; Sullivan, 1987). The theoretical advantages of propeller propulsion are very attractive. Highly loaded, thin and highly swept blades permit high propulsive efficiency at a flight Mach number of about 0.8. There is an interesting potential of fuel savings with single rotation propellers compared to the most efficient turbofans. But larger fuel savings (approximatively 25 percent) may be achieved by using counter-rotation propellers to recover swirl loss. This solution leads to a reduction of the propeller diameter and a smaller reaction torque.

Presented at the Gas Turbine and Aeroengine Congress and Exposition—June 4-8, 1989—Toronto, Ontario, Canada
The purpose of this paper is to demonstrate that modern numerical methods currently used in fan aerodynamic design (Karadimas, 1988) can be easily applied to flow computations around high speed propellers and that good performance might be expected by using this method.

2. DESIGN PROCEDURE

The calculation procedure is the one generally used for fan design, as described in figure 1.

![Fig.1 : Highspeed propellers calculation procedure](image)

The first step of the design process is an inverse axisymmetric throughflow computation. Such numerical methods are quick and efficient tools for optimizing some geometric and aerodynamic parameters (flowpath shape, number of blades, axial position of the bladerows, radial distribution of aerodynamic loading,...). The axisymmetric mean flow through the transonic propellers is computed by means of a streamline curvature method developed at S.N.E.C.M.A.. Some modifications were however necessary in order to calculate high speed propellers, and will be hereafter detailed. For a given set of the above parameters, the main results of this first computation are the velocity diagrams for each spanwise position and each propeller.

When the velocity diagrams are known, 2D profiles are designed for several spanwise locations. The quasi-geometric methods used for fan aerodynamic design were modified for application to high speed propellers because solidity is much lower and tip flow is not guided by a casing. The complete blade geometry is obtained by stacking these profiles on the blade axis, and the design phase is then completed.

The aerodynamic analysis of the propellers is then performed for several operating points. A good prediction of the flowfield through high speed propellers, more than classical fans, needs the use of three-dimensional flow computations. Particular interest is focused on the Mach number distributions and on the shock strength, and most of the relevant information can be obtained by simulating the inviscid, time-averaged flow through the counterrotating propellers. In practice, a 3D Euler solver is used, with the same pseudo-unsteady method as that developed by O.N.E.R.A. for various internal transonic flows (Brochet, 1980; Viviani and Veuillot, 1978). This Euler computation provides the shock losses, and the final loss prediction is obtained by adding the viscous profile losses. These are provided by a bidimensional boundary layer calculation based on the Euler static pressure distributions (Papailiou, 1974). The profile shapes are then modified, and this step repeated until the required flow properties (deflexion, throat margin,...) are obtained with satisfactory performance.

Finally, the global performances of the propellers are obtained by means of a through-flow computation in which the profile and shock losses are directly issued from 3D Euler and boundary layer analysis. This calculation takes into account secondary flows at the hub wall (Brochet and Falchetti, 1987).

The procedure is repeated with different loading distributions until the result is satisfactory.

3. SELECTED APPLICATION

Since S.N.E.C.M.A. is involved in a cooperative program with General Electric on the UDFT™/GE36 (fig.2), the formula for the presented application is the same as the one chosen by G.E.. The aerodynamic design conditions are as follows:

- Flight Mach number $M = 0.78$
- Altitude $Z = 10500$ m (35000 ft)

For that study, a 12 blade forward rotor and a 10 blade aft rotor were finally chosen.

![Fig.2 : Unducted fan engine](image)

4. DESIGN THROUGH FLOW CALCULATION

The axisymmetric mean flow in the transonic propeller is computed by means of a streamline curvature method developed at S.N.E.C.M.A.. This method can only calculate internal flows. A fictitious casing above the blades was added, whose diameter is assumed to be large enough so that the mean flow near the propellers is not disturbed. The downstream contraction ratio of the flowpath is adjusted in order to obtain a downstream static pressure close to the
freestream value. The blades are extended up to the fictitious casing by thin and non-lifting surfaces. The flowpath is shown in fig.3 with the calculation stations. The computation grid is adapted to the highly swept blades of the propellers, and is fine enough to allow a correct representation of chord- and spanwise distributions of blade thickness and aerodynamic loading.

The spanwise loading distribution is optimized according to the above criteria, and with respect to the fixed value of total shaft power. The final distribution selected for the forward and the aft rotor is shown in fig.4.

In the design phase, the losses are predicted by means of different correlations. Profile loss calculation is based on the Lieblein equivalent diffusion factor, which is commonly used for compressor design. A new model developed for low solidity blades provides the shock losses. The suction side peak Mach number is computed as a function of velocity triangles and some blade geometrical parameters. Then, the classical relationships provide the total pressure losses through a strong shock. Fig.5 shows the agreement of the loss model in through-flow calculation with the results of 3D Euler and boundary layer computation.

The main objectives of the inverse through-flow calculation are first to correctly select the spanwise blade loading, and second to optimize the hub wall shape. The following points must be taken into account.

The first one concerns the power loading near the hub wall. Since the blades are thick in this area, the relative Mach number should be decreased, especially at the leading edge of the aft rotor. The power loading is controlled in order to achieve this aim, especially to avoid choked flows in this region.

Since tip sections of an unducted propeller have no lift, the power loading in this region is decreasing to zero. The spanwise gradient of loading must however be reduced as much as possible for a limitation of 3D effects.

5. BLADE DESIGN

The blade geometry on the two rotors is defined by means of nine 2D profiles regularly spaced along the span. The profiles are designed by means of a method developed by S.N.E.C.M.A for supersonic compressor profiles. Some structural specifications and aerodynamic parameters must be specified, such as:

- Leading and trailing edge thicknesses
- Maximum thickness
- Chord length
- Suction side incidence
- Downstream deviation
- Throat margin

Since throat margin and downstream deviation are physical results instead of data, their values are the result of several iterations between blade design and 3D flow analysis. For the first calculation, the deviation is estimated with the equivalent Carter formulation, corrected with a model for 3D effects.

The next step consists of generating the blade volume. New profiles are at first obtained after a conformal application on axisymmetric stream
surfaces, before being stacked on a tridimensional axis that accounts for the blade slope in the tangential and axial directions. In order to facilitate blade manufacture, shape smoothness is needed which leads to better mechanical and aerodynamic properties. Top and side views of the forward and aft propeller blades are shown in fig.6 and fig.7.

![Fig.6 : Forward and aft rotors - top view.](image)

![Fig.7 : Forward and aft rotors - meridian view.](image)

6. 3D ANALYSIS

Solving the full 3D viscous and unsteady flow field of a counterrotating propeller, as in most configurations of multiblade row machines, is very difficult. Three-dimensional, unsteady, inviscid codes are now in use and 3D Navier-Stokes solvers are currently under development, but were not available during this study. However, most of the relevant information on the transonic flow field properties (Mach number distributions, shock strength) can be obtained by simulating the time-averaged inviscid flow through the propellers.

Three-dimensional inviscid codes have been developed using several algorithms, including the pseudo-unsteady methods in use at O.N.E.R.A. and S.N.E.C.M.A. for several years. The code used for this propeller application is a full Euler solver, including the energy equation for flows with non-uniform rothalpy. The Euler equations are written under conservative form, for a finite difference discretization with an explicit Mac Cormack scheme. Boundary conditions are expressed as compatibility relations. Since the axial flow is assumed to be subsonic, four conditions must be specified at the upstream boundary, and only one at the downstream boundary. At the upstream boundary, total pressure, total temperature, and radial and tangential components of the velocity are specified. At the downstream boundary, simple radial equilibrium is enforced. This condition is computed at each iteration, with a fixed value of the hub wall static pressure. The other boundary conditions are as follows: spatial periodicity in the pitchwise direction, no slip on solid surfaces, and non reflecting boundary condition on the upper free surface.

![Fig.8 : Computation domain for a single propeller.](image)

The time averaged Euler equations are solved in the reference frame of each bladerow, so that the computation mesh is needed for each propeller. A multidomain approach is used in order to get well adapted grids for the lower subspace including the blades and for the open space above the blades. During the solution procedure, computation variables at the boundary of those two subdomains are coupled by means of compatibility relations. For this study, 60x18x16 grids are used, as shown in fig.9 for each bladerow.

Fig.14 shows countours of relative Mach number on the pressure and suction sides of the forward and aft blade rows, and fig.16 is a graphic superposition of both grids and Mach number distributions. On the suction side of the both blades (Fig. 14), strong shocks are located along the trailing edge, terminated at about 70% of span on the first propeller and 40% on the second. The maximum local values of the relative Mach number is about 1.25 for the first propeller and 1.35 for the second. Considering the high inlet Mach number, these are remarkably good results.

In addition, there is no supersonic region in the vicinity of the nacelle / pressure side surface intersection for the first propeller and a small supersonic region for the second one. This indicates that the flow in the hub region is not choked, which meets one of the previous design objectives. More details are given in Fig.14, 15 and 16.
7. BOUNDARY LAYER AND SECONDARY FLOW COMPUTATIONS.

7.1 Boundary layer analysis.

Bidimensional boundary layer computations are performed for each designed blade section, using the 3D Euler static pressure distributions. Results are presented for the hub section of the aft propeller in Fig.10, which shows the chordwise evolution of the shape factors $H_{12}$. The boundary layer is well attached, as indicated by the low values of $H_{12}$ in the turbulent part. Viscous losses and downstream deviation are deduced from these calculations.

The spanwise boundary layer loss distributions are presented schematically for each rotor in Fig.11 in addition to Euler shock losses. The total loss coefficient is in the order of 1% for both the forward rotor and for the aft rotor. The low values of Euler losses under 70% on the first blade and 40% on the second confirm the location of the strong shocks.

Fig.12 shows the good agreement between the calculated downstream relative angle (3D Euler + boundary layer) and the objective (design through flow computation).

7.2 Secondary flows computation.

The last step of the aerodynamic analysis of the propellers consists of a prediction method for secondary flows in the hubwall region which provides the viscous layer losses on the nacelle and the actual operating incidences of the hub profiles. In this method (Brochet and Falchetti, 1987), secondary flows are thought of as a three dimensional end wall boundary layer coupled to a through-flow calculation which represents the inviscid free stream. Local
values of secondary flow effects are computed by solving a set of equations deduced from the pitchwise averaged, parabolized Navier-Stokes equations.

The longitudinal component of velocity is obtained by solving a system of integral equations (boundary layer approach), while the transverse components are yielded by the differential transport equation of vorticity ("Secondary flows" approach (Sitaram and Lakshminarayana, 1983)). Fig. 13 shows the absolute flow angle and relative Mach number at the outlet of the first blade row.

8. CONCLUSIONS

For several years, 2D and 3D aerodynamic computational methods have demonstrated high performance in the design of turbofan engines. A similar methodology was successfully developed by S.N.E.C.M.A. for high speed propellers.

The use of through-flow computations allows the optimization of power loading distributions. The profile design methods take into account blade passage effects, unlike isolated airfoil design methods. This approach, with a particular emphasis on transonic flow properties, yields choices differing from those obtained by the application of classical propeller theories.

ACKNOWLEDGEMENTS

The authors wish to thank S.N.E.C.M.A. for permission to publish the paper.
Fig. 14: relative Mach number
Fig. 15: Relative Mach number
Fig.16 : Graphic superposition of both grids and Mach number
REFERENCES

AGARD Fluid Dynamics Meeting on Aerodynamics and Acoustics of Propellers, Toronto, Canada.

Brochet, J., 1980, "Calcul Numérique d’Ecoulements Internes Tridimensionnels Transsoniques."
La Recherche Aerospatiale n° 1980-5, pp 301-315.
English translation ESA TT 673.

8 th ISABE, Cincinnati.

ASME Paper n° 88-GT-141.

NASA Contractor Report 3671.

Papailiou, K.D, 1974, "Optimisation des dispositifs decélérateurs à fortes charge fondate sur une théorie intégrale de la couche limite."
Thèse de Doctorat et Sciences Univ. de Lyon.

Smith, L.H., Jr., 1987, "Unducted Fan Aerodynamic Design."
ASME Paper n° 87-GT-233, July.

AGARD Conference-Proceeding 421.

ONERA publication n°1978-4
English translation ESA TT 561.