THREE-DIMENSIONAL NAVIER-STOKES ANALYSIS OF A TWO-STAGE GAS TURBINE

Andrea Arnone
Department of Energy Engineering
University of Florence
Florence, Italy

Erio Benvenuti
Nuovo Pignone S.p.A.
Florence, Italy

ABSTRACT

A three-dimensional Navier-Stokes solver has been extended to include multi-row capability. The coupling between the various rows is implemented by means of mixing planes. Those planes are handled by retaining the radial distortions while mass-averaging in the pitch-wise direction. The code is applied to a two-stage, heavy-duty gas turbine at design conditions. A comparison of pitch-averaged quantities between blade rows with preliminary measurements and with through flow analysis is presented along with a discussion of the flow features in terms of secondary flows. Using roughly half a million grid points, an operating condition can be examined in about an hour on a modern supercomputer.

INTRODUCTION

The design process of gas turbine blade rows is based on the efficient use of the currently available tools, such as through flow, 2D inviscid/viscous and 3D inviscid/viscous analyses. With the recent progress in turbine performance, detailed aspects like heat transfer, secondary flows, and radial mixing have become more and more important, and attempts to predict them are mostly based on three-dimensional viscous procedures. In the last few years, several codes have been developed for predicting three-dimensional viscous flow through an isolated blade row. The works of Subramanian and Bozzola (1987), Chima and Yokota (1988), Choi and Knight (1988), Davis et al. (1988), Hah (1989), Nakahashi et al. (1989), Weber and Delaney (1991), Dawes (1991), Jennions and Turner (1992), and Arnone (1993) are some examples of efforts in this direction. Along with the evolution in computer technology, memory and computing time have become cheap enough, and viscous codes based on the Navier-Stokes equation have become feasible for industrial applications and are routinely used in turbomachinery component design and optimization.

The real flow inside a turbomachine is unsteady and strongly influenced by rotor-stator interactions, wake, and clearance effects. Even if important steps are being made in time-accurate simulations (e.g. Rai, 1987, Rao and Delaney, 1990), this approach is still very expensive when applied in three dimensions. Therefore, a time-averaged approach should still be considered a basic tool for modern, every-day design.

Experience has shown that an isolated blade row study can be very useful in understanding the flow features of a multistage machine. However, the machine environment can strongly influence the row performance, and establishing the proper match between the various rows with an isolated row solver is a time-consuming process, especially for transonic flow conditions.

In the present work, the three-dimensional Navier-Stokes solver TRAF3D, developed by Arnone et al. (e.g. Arnone et al., 1993, Arnone 1993), has been extended to include multi-row capability. Communication between blade rows is achieved by using mixing planes, which are handled by keeping the radial distribution while averaging in the circumferential direction: "pitch-averaging". Using this approach, the flow quantities as well as the radial distortions are easily transferred between blade rows.

The procedure is applied to a two-stage heavy-duty gas turbine in order to have a picture of the whole turbine. An operating condition can be examined in about 25 hours on a modern supercomputer.
workstation using a total of roughly half a million points (1-1.5 hour on the Cray Y-MP).

Computed results are discussed and compared with through flow analysis in terms of pitch-averaged quantities between blade rows. Some preliminary measurements obtained on an instrumented machine are also included. However, before comparing overall performance with calculations, the effects of injected cooling air, turbine inlet, temperature circumferential distortion due to a single can combustor, and inter-blade flow leakage must be accounted for in the theoretical predictions. A study of secondary flow motion is presented in terms of simulated oil flow patterns close to solid walls.

COMPUTATIONAL PROCEDURE

Basic Single-Row Solver

The computational procedure is based on the TRAF3D code developed by Arnone et al. (1992, 1993). Details on the numerical aspects of the code can be found in the bibliography, however, the key aspects of the methodology will be listed for completeness in the following.

The unsteady Reynolds-averaged Navier-Stokes equations are discretized using a cell-centered finite-volume scheme, and a four-stage Runge-Kutta stepping method (Jameson et al., 1981). The eigenvalues scaling of Swanson and Turkel (1987) and Martinelli and Jameson (1988) is implemented in 3D in order to guarantee a minimum amount of artificial dissipation. The turbulence closure is provided by the two-layer algebraic model of Baldwin and Lomax (1978). Computational efficiency is achieved through local time-stepping, implicit variable-coefficient residual smoothing, and a multigrid method. Those techniques have proven to be effective for steady as well as unsteady calculations (Arnone et al., 1993).

In turbomachinery blade row calculations, there are mainly four types of boundary conditions: solid walls, inlet, outlet, and periodicity. The classic no-slip condition is applied on solid walls. The pressure is extrapolated from the interior, and the wall temperature is prescribed. At the inlet, the presence of boundary layers, on hub and tip end walls, is accounted for by giving a total pressure and a total temperature profile whose distribution simulates the experimental one. According to the theory of characteristics, the flow angles, total pressure, total temperature, and isentropic relations are used at the subsonic-axial inlet, while the outgoing Riemann invariant is taken from the interior. At the subsonic-axial outlet, the value of the static pressure at the hub is prescribed, and the density and components of velocity are extrapolated. The radial equilibrium equation is used to determine the span-wise distribution of the static pressure. On the periodic boundaries where the grid is not periodic, linear interpolations are used to compute the value of the dependent variables. The clearance region is handled by imposing periodicity conditions across the airfoil without any modelling of the blade cross-section.

Multi-Row Solver

The TRAF3D code, briefly presented above, has been extended in order to handle multiple rows. There are several methods for giving a code a multi-block capability. In this application, as a similar number of grid points are used to discretize the blade passages, it was found useful to add another index which accounts for the number of the block (i.e. blade passage) to the matrices. This approach is easy to implement and has the advantage of automatically providing a strong link between blocks throughout the whole multigrid process, while it may waste memory if the blocks are very different in size.

The delicate aspect in coupling together blade rows is how and where information is transferred from one row to the other. In practice, most state-of-the-art turbine blades work near the choke condition, and stators and rotors are very close to each other. Consequently, pitch-averaging can produce undesired reflections of the shock system. One way of reducing this effect is allowing the grids to overlap so that inlet/outlet boundary conditions are imposed not too close to the blade passage. The disadvantage of this approach is that the averaging process to get the necessary flow quantities takes place close to the blade passage, and depending on the averaging procedure, the conservation of energy or momentum or mass can be poor if the flow exhibits strong gradients. Another possibility of linking blade rows, is to slightly separate them, and define unique mixing planes where pitch-averaging is performed. If the mixing planes are placed far enough apart, the flow will result quite uniform in the circumferential direction and the averaging process will be relatively accurate in conserving mass, momentum, and energy. On the other hand, the contoured endwall will result slightly modified with respect to the machine geometry.
It is believed that the above are only some of the several possibilities that researchers are investigating over this still very open issue.

The present application is dedicated to gas turbine bladings, where a trailing edge shock system is expected, and most of the endwall is contoured inside the blade passage. Therefore, it seemed convenient to separate rotors and stators and define unique mixing planes.

In order to minimize distortions due to pitch-averaging, the characteristic approach previously discussed for isolated blade rows is maintained. At the turbine inlet, total quantities and flow angles are prescribed. On the following rows, the total temperatures, total pressures, and flow angles, necessary as inlet conditions, are computed by mass averaging those quantities on the previous blade row at the desired radius. The pressure distribution, needed at the row exit comes from mass-averaging the successive rows. The radial equilibrium is applied only at the turbine exit (last blade row).

As in a multigrid multiblock approach, the boundary conditions are updated at each Runge-Kutta step in order to provide as much of a link as possible between blocks.

APPLICATION AND DISCUSSIONS

As an application of the procedure that has been briefly presented above, the TRAF3D code was used to analyse the two-stage transonic blading of a 2 MW industrial gas turbine designed and produced by Nuovo Pignone (PGT2). Although in the small power range, this turbine features state-of-the-art design with an inlet temperature of 1370 K, cooled first stage nozzles and blades, and 12.5:1 pressure ratio in two stages. Manufacturing constraints and a need to limit the production cost led to a design with low aspect ratio airfoils, especially for the first stage. Secondary flow effects were therefore expected. Having no instruments like TRAF3D available at that time, some account of secondary effects on losses and flow angles was
FIG. 3 PRESSURE CONTOURS AT MIDSPAN (FIRST STAGE).

made by means of semi-empirical correlations. A meridional view of the turbine under investigation is given in Fig. 1.

**COMPUTATIONAL GRID**

The three-dimensional grids were obtained by stacking two-dimensional grids generated on blade-to-blade surfaces at constant radii. In order to minimize the grid skewness, on blade-to-blade surfaces, a non-periodic H-type grid structure was implemented (see Fig. 2). The removal of mesh periodicity allows the grid to accommodate large camber airfoils with a low level of skewness. To minimize the interaction between strong flow gradients and non-periodic boundaries, the mesh correspondence is broken before the leading edge and on the wake, but not inside the blade channel. Even if non-periodic C-type grids are more suitable for modelling a rounded leading edge (Arnone et al., 1992), an H-type structure has been found effective in alleviating the memory overhead.

The inviscid grids are elliptically generated, controlling the grid spacing and orientation at the wall. Viscous blade-to-blade grids are then obtained from inviscid ones by embedding lines near the wall with the desired spacing distribution.

In the span-wise direction, a standard H-type structure is used. Near the hub and tip endwalls, geometric stretching is used for a specified number of grid points, after which the span-wise spacing remains constant.

Each blade channel has 73 points in the axial direction, 41 points are located in the blade-to-blade direction, and 33 in the span-wise one. A total of about four hundred thousand grid points has, therefore, been used to discretize the turbine.

Figure 3 refers to the first stage and reports blade-to-blade pressure contours at mid span for different values of the rows' axial-distance. Figure 3 (a) corresponds to the real row position in the machine. The shock system coming out from the stator interacts with the mixing plane where a constant pitch-average value of static pressure is imposed. As a result of that, a shock is formed on the stator suction side after the throat one. This second shock has also been noticed in the isolated row analysis if the exit boundary is place close to the blade passage, and rapidly disappears when moving the exit boundary downstream. When the axial row-distance is seated...
to about one airfoil axial-chord, no important shock reflection can be noticed (fig. 3 (c)). At this stage of the research, it is believed that this phenomena is mostly due to reflection caused by the exit constant pressure condition and not to the physical presence of a rotor blade-row. Therefore, as noted in the previous section, the solution of separating blade rows has been preferred.

Figure 4 shows the mass-flow discrepancy between the stator and rotor rows as function of the rows' axial-distance. As previously mentioned, the mass pitch-averaging is not very accurate in conserving mass if the flow exhibits consistent gradients. By separating the blade rows, gradients decrease and such errors can be reduced to about 0.3% for a (rows’ axial-distance)/(airfoil axial-chord) around the unit.

Figure 2 reports a three-dimensional view of the mesh along with meridional and blade-to-blade sections. The distance between rows has been set to about one airfoil axial-chord, and mixing planes are placed at the meshes interface (see Fig. 2).

**Distribution on The Mixing Planes**

A Full-Multigrid method with three grid levels was used, achieving four decades' reduction in the residuals in 150 multigrid cycles on the finest grid level. The calculation requires about 25 hours on a IBM risc 53H which corresponds roughly to one hour on a supercomputer like the CINECA (the Italian Universities' Computational Center) Cray Y-MP. Calculations were performed imposing a constant wall temperature of 0.7 times the total inlet one. The calculated mass flow error through the turbine was found to be less than one percent and distributed in a uniform way between the mixing planes.

Overall expansion efficiency predicted by the viscous analysis agrees within two percent with the measured one, after having been corrected to account for the estimated effects of coolant flow injection, circumferential incoming distortion, and inter-blade flow leakage.

The radial distribution of static pressure, total pressure, total temperature, and flow angle at design conditions is
reported in fig. 5. Estimates from a through flow analysis and preliminary measurements are also reported along with the TRAF3D predictions. The accordance is very good on the whole. The through flow calculation accounts for endwall losses by distributing them in the radial direction and does not provide any endwall boundary layer. However, the integrated values agree well with the 3D viscous analysis. The static pressure measured between blade rows on the shroud also agrees well with the predictions as well as the averaged exit total temperature.

Some appreciable disagreement between the through flow and the viscous prediction can be observed in the distribution of the exit angles from the two rotors. As will be discussed later, this is mainly due to the presence of strong secondary flows, which were accounted for in an approximate way in the design. Having measured the mass flow, total temperature, static pressure, rotational speed, and by estimating the exit flow angle in the relative frame, the absolute flow angle can be deducted. This quantity is not a real measurement but is quite reliable as errors in predicting the relative flow angle at the rotors exit are generally very small (Dixon, 1975). The deduction of this averaged angle applied to the second rotor exit (Fig. 5) is in very good agreement with the viscous prediction. Anyway, the deviations are limited to a few degrees from the axial direction and influence the overall performance of the machine very slightly. On the contrary, no
major total pressure and total temperature distortion due to secondary transport of low energy fluid can be observed.

Secondary Flow Structure
A study of secondary flow motion is presented in terms of particle traces close to solid boundaries. Figure 6 shows the particle traces on the suction and pressure sides of the blades, and on the hub and tip endwalls respectively.

The first stator is a typical high-turning nozzle. The flow enters at a very low Mach number and is rapidly accelerated and turned about 75 degrees. It has been noticed that, for those nozzles, secondary motions are generally not pronounced due to the strong flow acceleration. This is confirmed by the viscous analysis which shows just a little migration on the last part of the suction side, where pressure gradients become small.

On the contrary, the first rotor indicated a quite complicated secondary flow structure. On the pressure side, the strong blade camber produces a large spot of very low-velocity fluid which is driven mainly inward by the negative radial pressure gradients. Close to the hub, the trace of a corner vortex is visible, while on the tip, fluid migrates in the clearance region. Particle traces on the suction surface and on endwalls show the presence of passage vortices. The horse-shoe vortex on the hub endwall is transported toward the suction side, while the one on the casing seems to be sucked by the clearance. On the tip endwall, the trace of the leakage vortex is quite noticeable.

The second stator has a contoured endwall. With respect to the first nozzle, the inlet Mach number is now higher, and a net shock is predicted on the blade suction side. Secondary motion is present on the suction side, where the flow migrates toward midspan. When crossing the shock, particles lose axial momentum and the radial pressure gradient moves them toward the shroud. Pronounced secondary flows are also visible on the hub and tip endwalls.

The second rotor has a flow structure near the solid walls similar to the first one. The main difference being much less migration on the blade suction side.

It is believed that, even if a relatively coarse mesh has been used for each blade passage, most of the secondary flow details have been captured at least qualitatively.

CONCLUDING REMARKS
A three-dimensional code based on the Navier-Stokes equations has been extended to handle multiple blade-rows. The linking between blade passages is based on the concept of mixing planes. An application to a two-stage turbine demonstrates the capability of the procedure in capturing most of the flow features. Due to accelerating techniques, computer time was found feasible for industrial application.

It is believed that Navier-Stokes analyses will substantially help in further improving design, thanks to their inherent capability of exploring flow-field details previously only accounted for empirically.

ACKNOWLEDGEMENTS
The development of the TRAF codes has been a long-term project which was started at the University of Florence in 1985 under the supervision of Prof. Sergio S. Stecco, who unexpectedly passed away in May 1993.

The authors have worked on this project with his image and advice very clearly in mind, and would like to dedicate this technical paper to his memory.

REFERENCES


