UNSTEADY HEAT TRANSFER IN STATOR-ROTOR INTERACTION
BY TWO EQUATION TURBULENCE MODEL

V. Michelassi, F. Martelli
Energetics Department "Sergio Stecco"
University of Florence
Florence, Italy

R. Démos, T. Arts, C.H. Sieverding
Von Karman Institute for Fluid Dynamics
Rhode Saint Genèse, Belgium

Presented at the International Gas Turbine & Aeroengine Congress & Exhibition
This paper has been accepted for publication in the Transactions of the ASME
Discussion of it will be accepted at ASME Headquarters until September 30, 1998
ABSTRACT

A transonic turbine stage is computed by means of an unsteady Navier-Stokes solver. A two-equation turbulence model is coupled to a transition model based on integral parameters and an extra transport equation. The transonic stage is modeled in two-dimensions with a variable span height for the rotor row. The analysis of the transonic turbine stage with stator trailing edge coolant ejection is carried out to compute the unsteady pressure and heat transfer distribution on the rotor blade under variable operating conditions. The stator coolant ejection allows the total pressure losses to be reduced although no significant effects on the rotor heat transfer are found in both the computer simulation and measurements. The results compare favorably with experiments both in terms of pressure distribution and heat transfer around the rotor blade.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A⁺</td>
<td>transition model constant</td>
</tr>
<tr>
<td>C</td>
<td>chord</td>
</tr>
<tr>
<td>Cₚ</td>
<td>turbulence model constant</td>
</tr>
<tr>
<td>fᵢ</td>
<td>intermittency function</td>
</tr>
<tr>
<td>Fₕ,Gₕ</td>
<td>convective flux vectors</td>
</tr>
<tr>
<td>F₉,G₉</td>
<td>diffusive flux vectors</td>
</tr>
<tr>
<td>H</td>
<td>source vector</td>
</tr>
<tr>
<td>k</td>
<td>turbulent kinetic energy</td>
</tr>
<tr>
<td>M</td>
<td>Mach number</td>
</tr>
<tr>
<td>Nu</td>
<td>Nusselt number</td>
</tr>
<tr>
<td>P</td>
<td>pitch</td>
</tr>
<tr>
<td>q</td>
<td>mass flow rate</td>
</tr>
<tr>
<td>Q</td>
<td>vector of unknowns</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>S</td>
<td>mean strain</td>
</tr>
<tr>
<td>t</td>
<td>time</td>
</tr>
<tr>
<td>Tu</td>
<td>turbulence level</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Greek symbols</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>α</td>
<td>transition model constant</td>
</tr>
<tr>
<td>β</td>
<td>turbulence model constant</td>
</tr>
<tr>
<td>ε</td>
<td>turbulence dissipation rate</td>
</tr>
<tr>
<td>µ</td>
<td>viscosity</td>
</tr>
<tr>
<td>θ</td>
<td>momentum thickness</td>
</tr>
<tr>
<td>ρ</td>
<td>density</td>
</tr>
<tr>
<td>τ</td>
<td>turbulence time scale</td>
</tr>
<tr>
<td>ŵ</td>
<td>vorticity</td>
</tr>
<tr>
<td>ω</td>
<td>turbulence frequency</td>
</tr>
<tr>
<td>ξ,η</td>
<td>curvilinear coordinates</td>
</tr>
</tbody>
</table>

INTRODUCTION

Despite the inherently unsteady nature of turbomachinery flows, the numerical and experimental analysis of modern turbine stages is quite often done in steady conditions. With the constant improvement in the experimental and computational techniques it is now possible to investigate unsteady turbomachinery flows in more details although limited to two-dimensional cases (Moss et al. 1997). The possibility to study unsteady three-dimensional stages is still limited by the memory and computer time requirements and can be performed only by using quite coarse grids which do not allow the boundary layers to be accurately modeled. In case the three-dimensional (3D) nature of the flow is of primary importance, it is possible to use the pitch-averaging technique (Ho and Lakshminarayana, 1996). Still, the availability of unsteady simulations and measurements will contribute to a better comprehension of the unsteady phenomena together with the effect of changes in the operating conditions like the Reynolds and Mach number, the rotational speed etc.

Jameson (1991) proposed a time marching algorithm with a double physical-numerical time step which could be easily employed for the calculation of unsteady flows. From his work other authors computed unsteady flows in turbomachines (Rao and Delaney, 1990, Rao et al, 1992). Unfortunately, the unequal number of blades in the stator and
rotor rows can require the computation of the whole cascade which is often unfeasible because of the large memory and computer time requirements. Accordingly, the stage geometry is often slightly modified to allow a simulation with a reasonable computing effort.

The problem of heat transfer deserves a special attention especially for the first stage downstream of the combustor because of the high fluid temperatures. In fact, a large portion of the turbine blade boundary layer can be transitional. This implies that an accurate prediction of the effects of turbulence is of primary importance if losses and/or heat transfer are to be computed in presence or not of cooled blades (Tanuma et al. 1997).

The present work is devoted to the study of a transonic turbine stage with trailing edge coolant injection and heat transfer on the rotor blade under variable operating conditions. The investigation was carried out in cooperation with other international institutions which provided the design of the blade and the experimental results that are compared with the computations. To the knowledge of the authors, most of the unsteady calculations in the literature so far are devoted to the prediction of the pressure and velocity field. Still, the unsteady heat transfer phenomenon is of primary importance and there is no evidence that a steady heat transfer calculation will perform like an unsteady computation. Accordingly, the study of the interaction between the stator and rotor rows was carried out accounting for the unsteady transitional nature of the boundary layers and heat transfer.

DESCRIPTION OF THE SOLVER

Algorithm

The implicit time marching code MDFLOS3D solves the unsteady Favre averaged 3D Navier-Stokes (N-S) equations in terms of conservative variables. The code is here used in a 2D manner so that the two dimensional version of the algorithm will be shortly described. The solver is based on the scalar approximate factorization proposed by Pulliam and Chaussee (1981) and applied to the computation of transonic turbine flows by Michelassi et al. (1994, 1997). The transport equations, the stencil of which is given in equation (1), are discretized by using centered finite volumes in curvilinear non-orthogonal structured grids:

$$\frac{\Delta Q}{\Delta t} = \frac{\partial Q}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} - \frac{1}{Re} \left( \frac{\partial F}{\partial y} + \frac{\partial G}{\partial x} \right) + H = 0 \quad (1)$$

The implicit scalar approximate factorization is originally developed for the computation of steady flows. To take full advantage of the implicit formulation, the solver advances in time by using a local time step strategy. The unsteady time accurate solver introduces a double time step, as originally suggested by Jameson (1991) for explicit time marching algorithms, as shown in equation (1). The numerical time derivative (n) is used to advance in the numerical time with a local time step strategy. The physical time accurate derivative (p) acts like a source term, so that when the numerical time transient is eliminated, and the numerical time derivative is zero, equation (1) becomes an unsteady time accurate equation. Equation (1) is solved by the usual scalar approximate factorization in which the unsteady terms are treated implicitly as source terms. For further details about the algorithm see Michelassi et al. (1996).

Turbulence Model

The two-equation model by Wilcox (1988) is based on the characterization of the local state of turbulence by two parameters: the turbulent kinetic energy $k$ and the frequency $\omega = \epsilon / k$, where $\epsilon$ is the rate of dissipation of $k$. The eddy viscosity $\mu_e$ is related to $k$ and $\omega$ by

$$\mu_e = C_{\mu} \frac{p k}{\omega} \quad (2)$$

and the distribution of $k$ and $\omega$ is calculated from two model transport equations with the original set of constants (Wilcox, 1988).

The k-\omega model, as all the other two-equation turbulence models, was found to overestimate the turbulent kinetic energy in stagnation points (i.e., near the leading edge of turbine blades) followed by a developing boundary layer. As shown by Kato and Lauder (1993), the overprediction of $k$ is caused by the presence of normal stresses in the expression of the production. To avoid this, they reformulated the production rate as the product of rotational and irrotational contributions. This approach, while sorting a positive effect in terms of losses and transition prediction (Michellassi et al. 1997), lacks any physical justification and was therefore abandoned in favor of a constraint on the turbulence time scale $\tau$ recently proposed by Durbin (1996). This realizability constraint, which ensures that the turbulent kinetic energy is always positive, is formulated assuming that the velocity fluctuation component normal to the wall is known (i.e., a third transport equation is required). When retaining the two-equation model stencil under the assumption of isotropic turbulence, the constraint on $\tau$ can be reformulated as a constraint for the frequency $\omega$ as follows:

$$\omega = \frac{1}{\tau} = \frac{\epsilon}{k} = \max \left( \omega, 1 \left( \frac{1}{\beta}, \sqrt{\frac{1}{8 \beta^2}} \right) \right) \quad (3)$$

Equation (3) clips the specific dissipation rate which cannot reach values below a certain limit, which varies depending on the local mean strain $S$ and applies only in boundary layers where the local $M$ is below 0.2-0.3. The implementation of equation (3) in the k-\omega model helped solving a large portion of the numerical problems, together with the desired limitation of the production rate.

Transition Model

Transition to turbulence in flows over turbine blades has been extensively measured and computed (Mayle, 1991, Hazarika and Hirsch, 1995). The mechanism of by-pass transition is influenced by a number of parameters, like turbulence level, pressure gradient, flow curvature, which are difficult to include into a model. Transition has a dramatic impact on the heat transfer over turbine blades, the proper prediction of which has a crucial importance in the design phase. Most of the studies on transition for the flow in gas turbines has been carried out in steady situations (Johnson and Erkan, 1997) and in presence of turbulence levels which are often smaller than what are found in real turbomachines (Hoogendoorn et al., 1997). Addison and Hodson (1992) analyze an unsteady transitional boundary layer in a low-Re, low-M turbine stage, but the model they present still requires validation and tuning for high speed flows. Cho et al. (1993) measured an unsteady boundary layer over a turbine blade under the influence of a series of wakes generated by a squirrel cage. A transition model was tested positively against the measurements for
the low Mach number under consideration. Following this work, Michelassi et al. (1997) adapted the formulation to the k-ω turbulence model for the computation of steady transonic turbine blades. The model evaluates a critical Reynolds number, Re_c, which, if exceeded, indicates that transition has started. Re_c is computed by using the empirical formulation proposed by Abu-Ghannam and Shaw (1980).

The intermittency function is cast into the expression for the turbulent viscosity (2) as:

\[ \mu_t = f_t \cdot \frac{\nu_t}{\omega} \]  

in which \( f_t \) is computed by using the following expression (Michelassi et al. 1997):

\[ f_t = \left[ \frac{A^+}{A^+ + (300 - A^+) \left( 1 - \frac{\text{Reg} - \text{Retr}}{\text{Retr}} \right)} \right]^{\alpha} \]  

Equation (4), which refers to model version A, reaches unity when \( \text{Reg} = 2 \cdot \text{Retr} \), which indicates that the boundary layer is turbulent.

The parameter \( \alpha \) in equation (4) controls the rise of \( f_t \) in the transition region (the larger the value of \( \alpha \), the longer is the transition length).

Observe that both the Abu-Ghannam and Shaw correlation and the intermittency function (4) require the computation of the momentum thickness \( \theta \) and an estimate of the boundary layer thickness. Following the work of Cho et al. (1993), who used a similar approach, and after intense numerical testing and validation in typical turbine geometries (Michelassi et al., 1977, Migliorini and Michelassi, 1997), this was accomplished by defining the boundary layer edge as:

\[ \bar{\omega} = \bar{\omega}_{\text{min}} + \left( \bar{\omega}_{\text{max}} - \bar{\omega}_{\text{min}} \right) \cdot 0.01 \]

in which \( \bar{\omega}_{\text{min}} \) and \( \bar{\omega}_{\text{max}} \) are respectively the minimum and maximum values of the vorticity in the cross section.

The intermittency function was computed at each time step on the basis of the instantaneous velocity, pressure, and turbulence fields.

Using the previous model the predicted intermittency function might have large variations from one time step to the next. In order to introduce a sort of history effect on the intermittency and stabilize the boundary layer state it was decided to add a transport equation for \( f_t \) in the model version B as follows:

\[ \frac{\partial f_t}{\partial t} + \frac{\partial u_x f_t}{\partial x} + \frac{\partial u_y f_t}{\partial y} + \frac{\partial u_z f_t}{\partial z} = 0 \]  

The value of the intermittency function placed in equation (3) is the maximum of the values computed by equations (4) and (5) at each time step. The model is not used in a narrow region on the rotor leading edge since the assumption of a laminar or turbulent leading edge did not affect the further development of the boundary layer.

**Grids**

Although the experiments are carried out on a annular test rig, the computer simulations are run in a two dimensional midspan geometry to reduce the computational effort. The computer program, coded in 3D, is used here in a 2D manner by solving one control volume only in the radial direction. The stator and rotor blade rows are then assumed to be linear. The I-type 200x104 grid, shown in figure 1, is one of the most general possibilities when staying with simply connected or single block grids. The equivalent grid for the rotor blade with 258x153 points is shown in figure 2. The grids are nearly orthogonal in the crucial stagnation point region. A close up view of the trailing edges is shown in figures 1 and 2, with the cut on the pressure side of the stator blade for the coolant ejection.

The exact rotor pitch at midspan is \( P_s = 54.0427 \) mm, while for the rotor row the pitch is \( P_r = 36.3099 \) mm. To have a nearly periodic problem one can solve two stator blades and three rotor blades since the pitchwise extension of the stator is \( 2 \cdot P_s = 108.0854 \) and the pitchwise extension of the rotor is \( 3 \cdot P_r = 108.9297 \). To have a fully periodic calculation the previous two lengths should be equal. Since the mass flow rate is controlled by the size of the stator row, it was decided to keep the dimensions of the stator row unaltered and change the dimensions of the rotor row to have \( 2 \cdot P_s = 3 \cdot P_r \). The dimensions of the rotor row can be changed in three ways:

- **reshape the pitch and the blade size by a scaling factor SC=2xP_s/3xP_r=0.992249. In this way the overall rotor row is reduced in size by a factor 0.775%.**
- **change the pitch of the rotor row so as 2xP_s=3xP_r. In this way the overall shape and size of the rotor blade is kept unaltered, while the throat area is reduced approximately of 0.7%.**
- **operate the scaling of the previous point, but rotate the rotor blade, to maintain the original throat area.**

Of the three, it was decided to use the first. The third option, while theoretically able to give a realistic reproduction of the original geometry, alters the rotor exit flow angle which can cause the boundary layer separation on the suction side.

While the stator has a constant height, the rotor passage has the variable height in the test rig qualitatively sketched in figure 3. The small reduction in the rotor throat area was then compensated by the grid expansion in the radial direction. This shape of the channel requires a quasi-3D calculation which is simulated by using a variable cross section thickness in the radial direction. The spanwise expansion of the rotor channel was further adjusted so as to match the measured mass flow rate.

**Boundary Conditions**

The rotor inlet total pressure, temperature, and inlet flow angle, and the stator exit static pressure are given to match with the experimental conditions. The inlet turbulence level (1.6%) and turbulence length scale (1%P_r) are fixed at the stator inlet. The value of the turbulence length scale, which governs the decay rate of turbulence, was selected on the basis of a previous validation of the model (Migliorini and Michelassi, 1997) against similar wind tunnel data. At the stator inlet, the static pressure is extrapolated, while the transported quantities are extrapolated at the rotor exit. A viscous adiabatic condition is set on the stator blade surface, whereas the surface temperature is fixed on the rotor blade, where the velocities are set to zero and the static pressure is computed by using a zero derivative normal to the wall. The turbulent kinetic energy is set to zero and \( \omega \) has a constant value on the solid boundaries (Wilcox, 1988). On the interface of the five subdomains the grid point distribution is not periodic. The continuity of the solution is enforced by extending the grid of each subdomain which overlaps on the neighboring subdomain. Details of the grid overlapping can be found in Michelassi et al. (1997).
EXPERIMENTAL DATA SET

The wide set of measurements (Sieverding et al., 1998) have been carried out on an annular cascade. The test rig is a compression tube annular cascade of the blow-down type. Air is supplied by a large cylinder, which stores compressed air at 250 bars, through a free moving light-weight piston. The testing time varies from 0.5 to 1 s. The operating conditions are as close as possible to modern aeroengines, with the only exception of the inlet total temperature which was set to 450 K in the experiments against values of the order of 1800 K for real operating conditions. The stator and rotor Re based on exit conditions and axial chord length are $9 \times 10^5$ and $7 \times 10^5$.

The mid-span pressure distribution on the rotor blade is measured in 24 points by using flush mounted fast response pressure transducers the accuracy of which is approximately 1% (Sieverding et al., 1998). The mid-span heat transfer rate is measured in the same 24 positions by using thin film gauges. The gauges allow Nu to be measured with an accuracy of 7% (Sieverding et al., 1998). The transducers are connected to the in-shaft electronics.

The measurements are performed for several rotor revolutions to extract average and instantaneous values of $M_{1S}$ and Nu with a sampling frequency of 4300, 4658, and 4873 Hz for 6000, 6500, and 6800 RPM respectively. The design rotational speed is 6500 RPM which implies a tangential velocity of 250 m/s at midspan. The angular speed of the rotor blade changes approximately 5% during a typical 0.5 s test with an initial 6500 RPM and an acceleration of 700 RPM/s. The effect of the change on the heat transfer and pressure measurements on the rotor blade surface is negligible (Sieverding et al., 1997).

The stator exit $M_{1S}$ is approximately 1.05 with an inlet flow angle of 0-deg. The relative rotor inlet M is 0.45, and the relative exit Mach number of the rotor row is 0.934. Under these conditions the stator row is choked and controls the mass flow rate. The inlet turbulence level is approximately 1.6%. Measurements have been carried out with and without cool air ejected through a slot from the pressure side of the stator blade (see figure 1). The coolant mass flow rate was 3% of the inlet mass flow rate, and the coolant total temperature was 0.72 $T_0$. 
Computational Results and Comparison with Experiments

Computational Details

The computer simulations of the stator-rotor interaction are initialized with a still rotor. Once a steady converged solution is reached, the rotor grids are shifted in the tangential direction. Each rotor blade requires 50 or 100 physical time steps to pass in front of a full stator blade pitch. The CFL number is 15 for the N-S equations and 10 for the k-ω step. The artificial damping weight are 1/2 for the second order and 1/64 for the fourth order. All the numerical simulations were performed on a Digital-Alpha Workstation 600/333. The code size is 120Mb and it requires approximately 24 to 36 hours to compute a solution which is periodic in time.

Figure 4. Convergence histories

Figure 4 shows the typical convergence history of an unsteady run. The abscissa gives the cumulative iteration number, while the ordinate reports the equation residual. Each time step is considered converged when the residual drops below $10^{-6}$. Figure 4.a shows that the stator rows converge in approximately 5 to 15 iterations, while the rotor requires typically twice as much iterations to reach the same convergence level. This is due to the transonic nature of the stator according to which the stator row feels only very weak disturbances from the rotor. Conversely, the rotor row feels the passing stator wakes which can induce large modifications in the flow field from one time step to the other.

A good indicator of the time-periodicity of the solution is the load, computed as the integral of the pressure distribution around the stator and rotor blades. Figure 5 shows the behavior of the stator and rotor blade loads against the time step. 100 steps correspond to a rotor translation equal to the stator pitch. Figure 5.a shows that the stator reaches a steady state quite soon. The small load fluctuations are caused by changes in the backpressure induced by the passing rotor. Still, the fluctuations are very small, if compared to those of the rotor shown in figure 5.b. The rotor blade reaches a periodic state after approximately 200-250 time steps, thereby after a $2-2.5\times P$, shift of the rotor. This amount of iterations for the start-up constitutes a sort of computational overhead since the results obtained in this preliminary
phase are substantially useless. To reduce the computational time of the start-up time it was decided to perform the first 200 iterations doubling the physical time step thereby reducing by half the number of steps required to reach a periodic solution.

The numerical simulations are performed in two basic configurations which differ in the gap between the stator trailing edge and the rotor leading edge. In the first computed configuration the gap is 0.5\(x_{\text{s}}\), whereas in the second the gap is reduced to 0.35\(x_{\text{s}}\). The same two geometries were measured in VKI (Sieverding et al., 1997) with and without coolant ejection from the stator pressure side. Of the two, only the large gap case was modeled here with and without coolant ejection, while the small gap case was computed with the coolant jet only. A set of preliminary runs were performed to setup the quasi-3D grid able to reproduce the experimental geometry with the best accuracy possible. As mentioned in the section about the computational grid, the spanwise expansion of the rotor channel, which is present in the experiments (see figure 3), was adjusted to compensate the reduction in the rotor pitch. The quasi-3D nature of the grid was compulsory since a set of preliminary calculations indicated that the mass flow rate of the stage was limited by the throat area of the rotor when assuming a constant thickness of the stream tube in both the stator and the rotor rows. This caused the choking of the rotor and not of the stator. The constant thickness stream tube approximation was then abandoned in favor of the quasi-3D approach. Figure 6 shows the effect of the spanwise expansion of the rotor channel on the distribution of the isentropic Mach number on both the stator and rotor blades. The plots show the \(M_{\text{is}}\) profile averaged over 100 time steps for both the stator and the rotor blades. When expanding the rotor blade height from 16 to 21% there is a large change in the stator blade \(M_{\text{is}}\) (see figure 6,a) especially in the throat and supersonic sections. This change is due to the increase in the rotor throat area with a consequent increase on the mass flow rate. The design isentropic exit Mach number of the stator is 1.05, and this value is matched only when using the 21% spanwise expansion ratio. The kink visible at \(x/x_{\text{c}}=0.9\) on the pressure side is caused by the sharp edge in proximity to the coolant ejection hole. Figure 6,b shows the effect of the expansion variation on the rotor relative \(M_{\text{is}}\) profile, including the average unsteady measurements (Sieverding et al., 1998). Here the prediction on the suction side tends to the experiments when moving from 16 to 21%.

Isolated stator and Rotor rows

Prior to the final runs the flow around the stator and the rotor blades have been computed separately to assess the grid requirements. On the basis of previous experience with this solver it was decided to use a relatively coarse grid for the stator (200x104) because of the adiabatic nature of the boundaries. Figure 7 compares the computations and the measurements of the stator-only exit flow angle. The flow turning is overestimated of less than one degree which corresponds to a positive incidence on the rotor blade. Figure 8 shows the effect of the positive incidence on the \(M_{\text{is}}\) profile around the rotor blade. The leading edge section is highly affected and, as will be seen in the following sections, this can have an impact on the heat transfer predictions.

The rotor grid is more refined (258x153) because of the need to compute the Nusselt number profile around the blade. The turbulence and transition models implemented for the calculations has been validated in several steady flows (Michelassi et al., 1997, Migliorini and Michelassi, 1997).
The isolated rotor row was studied with an inlet turbulence level of 1.6% (equal to the stator inlet level) and an inlet turbulence length scale equal to 0.01xP.

Prior to the quantitative analysis, the overall rotor flow pattern is described. Figures 10 and 11 refer the large gap case with coolant ejection from the stator blade trailing edge.

Figure 10. Local Mach number at two consecutive time steps

Figure 11. Static pressure at two consecutive time steps

The unsteady flow pattern

Figure 9 compares the average experimental Nusselt number for the 0.5xCax case with the predictions obtained by computing the rotor row alone. The various curves refer to different values of the coefficient α in equation (4). The plot shows that the predicted level of Nu on the pressure side are always in good agreement with experiments. On the suction side the average experimental profile shows that the onset of transition is probably spread in the range 0.4<s/S<0.8. This comparison shows that, regardless of the choice of the transition model constant α, which allows the length of transition to be controlled, the computation of the isolated rotor gives an unrealistic description of the Nusselt number on the suction side. Although the differences can be partly imputed to deficiencies in the turbulence and transition models, most of them are caused by the unsteadiness and the turbulence characteristic changes across the stator row which can be accounted for only by computing the unsteady stator-rotor interaction, which is the subject of the next sections.
Figure 12. Turbulence level isolines at three time steps

(a) Tu=0-20% 10 levels

(b) Tu=0-40% 10 levels

(c) Tu=0-20% 10 levels

Figure 13. Streamlines at three time steps

(a)

(b)

(c)
On the pressure side the low Mach number spot can be easily detected and survives till the trailing edge. The static pressure isolines (figure 11) show the expansion and recompression waves interaction between stator and rotor. Apparently, the supersonic flow region departing from the stator suction side does not reach the rotor blade since the pressure isolines on suction side exhibit a pattern which is very weakly affected by the passing wakes. The static pressure pattern on the suction side of the rotor blade varies in time also after the expansion where the flow reaches its maximum velocity of the order on $M=1.1$. The small variations of the stator pressure distribution are limited in the supersonic region in between the weak shock and the trailing edge. Figure 11 indicates that the pressure fluctuations fade away in the stator vane due to filtering effect of the supersonic flow region. Figures 12 and 13 show the turbulence level based on local velocities and streamlines on the rotor for three consecutive time steps. In the first (figure 12,a and 13,a) the stator wake travels close to the rotor suction side, then hits the rotor leading edge (figure 12,b and 13,b), and finally reaches the rotor pressure side (figure 12,c and 13,c). Figure 12,a shows that $Tu$ reaches a peak of 18-20% in the core close to the leftmost side of the computational domain. Anywhere else $Tu$ is below 2 to 4%. When the wake hits the leading edge (arrow in figure 12,b) the large velocity gradients together with the large values of the kinetic energy convected from the stator trailing edge increase $Tu$ up to 30% and more. This peak is located in proximity to the rotor leading edge, and is then convected downstream, as pointed out by the arrow in figure 12,c where the peak of $Tu$ moves with the mainstream on the pressure side. Apparently the maximum $Tu$ spot moves downstream with a small rate of decay. Still, the flow acceleration on the pressure side decreases and eventually moves and stretches this spot towards the wall, as shown by the arrow in figure 12,a. When looking at the suction side, only the first plot indicates large values of $Tu$ which are then dissipated while the effect of the wake fades away. Unlike on the pressure side, on the suction side there is no evidence of high turbulence spots. In general the cooling jet produced small changes in the turbulence flow field in the wake.

The streamlines at the first of the three time steps (figure 13,a) show a thickening of the boundary layer on the pressure side in proximity to the $Tu$ spot evidenced in figure 12,a. This spot is then convected away in figure 13,b. Figure 13,c shows the formation of the next thickening of the boundary layer (in the same position of the high-$Tu$ spot) as visible on the pressure side in proximity to the leading edge.

**Large gap (0.5×Cax) with and without coolant ejection**

In this configuration the axial distance between the stator t.e. and the rotor l.e. is $0.5×Cax$. The large gap case is computed without and with the stator pressure side coolant ejection and at 6000, 6500, 6800 RPM. The coolant mass flow rate is 3% of the overall mass flow rate at the nominal 6500 RPM rotational speed.

Figure 14 compares the computed and measured average $M_\infty$. The profiles are averaged over a full rotor revolution for the experiments and over one passage of a rotor vane in front of two stators after a periodic-in-time solution is achieved for the computations. The stator row shows how large the aerodynamic effect of the coolant is in case of the pressure side ejection. When the coolant mass flow rate is zero, the first shock is caused by the expansion wave followed by a compression departing from the flow recirculation on the pressure side (see figure 15,a). The second shock is the usual fish-tail shock departing from trailing edge vortices. When switching on the coolant jet the first expansion-shock wave is washed away since the flow recirculation on the pressure side disappears (see figure 15,b). The effect of the coolant jet is evident in figure 14,a which shows that only the fish-tail shock survives. This effect was already observed in the steady calculations.

![Figure 14. Isentropic Mach Number profiles](attachment:image.png)

The effect of the coolant jet on the rotor fades away, as proved by the $M_\infty$ profiles in figure 14,b. There are only marginal differences in the averaged isentropic Mach number profiles, and these are due to the small changes in the wake depth caused by the coolant jet. The $M_\infty$ profile on the rotor is very similar to a set of steady 3-D calculations performed on the annular rotor cascade (Sieverding et al., 1998) and to those of figure 6,b. Some very small differences arise in the leading edge region on the suction side where the flow acceleration predicted by the unsteady computations is slightly stronger than that given by the steady calculations. This seems to indicate an interaction between the rotor and the wake which affects the rotor inlet flow angle.

The effect of changes in the rotational speed are shown in figure 16 where the large gap configuration is studied at 6000 and 6800 RPM. It is interesting to observe that the reduction in RPM induces a positive
incidence on the rotor, as proved by the acceleration on the suction side in proximity to the leading edge, and also a modification of the $M_{\infty}$ profile on the stator in which the shock moves slightly downstream. The rotor profile is largely affected by the RPM variation, but the computations showed no flow separation despite the large angle of attack. Figures 14 and 16 show that the simulations, while predicting the correct trend with RPM, always tend to underestimate the flow acceleration on the suction side in proximity to the leading edge. This disagreement with the experiments fades away while moving downstream, and cannot be imputed only to some inaccuracy in the prediction of the flow angle, but are probably also due to some shock smearing. In fact, a positive angle of attack always gives a subsonic flow in the region $0.2 < x/C_{\infty} < 0.4$ (see figure 8), whereas the unsteady measurements indicate that the average flow is transonic (see figure 14,b). In other words the measurements seems to indicate that the flow acceleration for $0.2 < x/C_{\infty} < 0.4$ is caused by the interaction with the trailing edge shock departing from the rotor.

The computed rotor inlet/outlet flow angles, mass-averaged over one pitch, are shown in figures 17 and 18. The lack of flow measurements prevents from any comparison, but the analysis of the computed angles helps in understanding the flow pattern. Figure 17,a shows that the stator exit flow angle ranges from 72.5 to 73.7-deg which is close to what is found for the isolated stator row in which the average exit flow angle (with respect to the axial direction) is 73.6-deg.

Figure 15. Flow pattern detail in proximity to the stator t.e.

(a) $q_{\text{jet}} = 0\%$

(b) $q_{\text{jet}} = 3\%$

Figure 16. Effect of Rotational speed variation

(a) absolute (stator exit)  
(b) relative (rotor inlet)

Figure 17. Flow angles in the stator-rotor gap.

Still, changes in rotor RPM and coolant mass flow rate do not affect the average and scatter of the flow angle. When considering the
relative flow angle, computed in the tangential plane immediately upstream of the rotor leading edge shown by the arrow in figure 2, the differences are amplified. The computations with and without coolant jet show a difference of the order of 3-deg, which explains the small changes observed in the $M_a$ profiles in figure 14,b. The large variations of the 6000 and 6800 RPM calculations are due to the change of the rotational speed. Still, the shape of the angle variation versus the time step is similar for the four calculations. This indicates that the overall flow pattern remains substantially unaltered. The computed relative exit flow angles, shown in figure 18, refer to the large gap case with and without coolant ejection at the nominal RPM (6500) and with coolant ejection at variable RPM (6000-6800). The plot shows that the rotor relative exit flow angle does not feel the presence or absence of the cooling jet and it is also insensible to the angular speed variations.

![Figure 18. Relative rotor exit flow angle](image)

The analysis of the heat transfer is made by comparing the measured and computed Nusselt number over the rotor surface. The leading edge is positioned at $s=0$ and the curvilinear wall distance is normalized by the distance of the last measured point on the suction side. Figure 19 compares the averaged Nusselt number in which the experiments have been conducted only with the cooling jet. There is a dramatic change with respect to the steady calculations of figure 9. The computations with the coolant jet are closer to the experiments on both the suction and the pressure sides, although in the leading edge region the code predicts values of Nu which are smaller than the experiments. The agreement rapidly improves when moving downstream. The small level of the predicted Nusselt number in the range -20% $s$ < 20% is not likely to be caused by an insufficient turbulence level predicted in the leading edge region since the turbulence levels shown in figure 12 are well above 5 to 10%, which is more than enough to induce transition. The situation is not substantially altered when introducing an extra transport equation for the intermittency function (model B in figure 20) which causes a steeper rise of Nu on the suction side and a small improvement on the pressure side. The exponent $\alpha$ in equation (4) allows the length of transition to be controlled. Values of $\alpha$ above unity and smaller than two induce a steep rise of the intermittency function thereby reducing the transition length. Moreover figure 20 indicates that when reducing $\alpha$ from 2.00 to 1.25 the situation marginally improves in the range -20% $s$ < 20%.

![Figure 19. Nu profile with and without jet, large gap - model A](image)

![Figure 20. Effect of model modifications on Nu.](image)

Apparently, the underestimation of Nu close to the leading edge is partly governed by some inaccuracy in the aerodynamics prediction. This is proved by figure 21 in which the predicted averaged, minimum and maximum values of $M_a$ are compared with the respective measurements. On the pressure side, and on the suction side for $x/Cax > 0.45$, computations and experiments agree on both the average and the minimum and maximum values. The agreement deteriorates on the leading edge suction side where the code predicts velocity levels up to 20% smaller than what is indicated by the experiments. This disagreement is stemming from an underestimation of the angle of attack in some wake interference configurations, as indicated by figure 8 which shows how a large positive angle of attack can modify the leading edge pressure distribution. Moreover, as proposed by Giles (1988), the shock departing from the suction side of the stator trailing edge impinges on the nose of the passing rotor blade thereby inducing large pressure fluctuations. The predicted pressure, and $M_a$, fluctuations are somewhat smeared because of the grid skewness which introduce some numerical diffusion.
A more significative comparison is done in figure 22 which shows the instantaneous measured and computed Nu in 24 instants with a 0.04×P₄ step. The leading edge Nusselt number is generally underpredicted at each step, but it is interesting to see that the transition point on the suction side is often detected correctly like at the steps 4-5-9-10 and others. The use of the extra transport equation for the intermittency function generally improves the predictions mostly on the pressure side where the experiments show a weak effect of the passing wake. Conversely, on the suction side it is possible to follow the passing wake by observing the peak of the measured and computed heat transfer which is shifted downstream and slowly smeared. This phenomenon is captured by the computations with a good degree of accuracy.

Figure 21. Isentropic Mach number fluctuation range

Figure 22. Unsteady Nu (comp1=standard transition model, comp2=added transport equation for intermittency)

Figure 23. Unsteady nusselt number.

The effect of the passing wakes on the rotor heat transfer can be traced in figure 23 which shows how the peak of Nu is moving...
downstream. It is now easy to follow the development of the transition point in time. The triggering effect of the wake is not very large since the transition point on the suction side ranges between 0.2<s/smax<0.38 regardless of the presence of the cooling jet. This is understandable on account of the large turbulence level which is mainly responsible for the onset of transition, at least in the computer simulation code. The peak on Nu tends to smear in time and apparently the heat transfer increases in time while approaching the trailing edge because of the mentioned effect of the wake. This phenomenon is caused by the developing transition of the boundary layer. When comparing the steps 0.5 and 0.7 the isolines spread in space and the Nusselt number increases also in the upstream direction for both the qj=0% and 3% mass flow rate cases. In such a situation the computations indicate that the boundary layer thickness has increased and the elliptic flow region close to the rotor blade suction side has grown. This elliptic effect can have a large impact on the determination of the hot spots on the rotor blade.

Small gap (0.35xCax) with coolant ejection

When reducing the gap between the blade rows the overall flow pattern remains unchanged. Figure 24 shows the isentropic Mach number profiles on both the stator and rotor blades. The stator behaves very similarly to the larger gap case, while the rotor shows some differences with respect to the larger gap case especially in the throat. Apparently the average pressure distribution on the rotor feels the stronger disturbances coming from the stator row because of the shorter distance the wake and the shocks have to travel. Surprisingly, this has weak impact on the averaged heat transfer rate that shows very little changes with respect to the 0.5xCax case (see figure 25). The computations like the experiments do not show a clear average onset of transition on the suction side, unlike the 0.5xCax case in which the onset of transition was clearly detectable at s/smax=30. Experiments and computations agree in indicating a nearly flat Nusselt number distribution on the blade suction side. The pressure side profile is almost identical to the larger gap case with the same underestimation of the heat transfer rate which extends up to 50% of the blade. Overall, the 0.35xCax gap case does not give any extra information with respect to the 0.50xCax case in terms of unsteady behavior and flow angles.

CONCLUSIONS

The availability of a set of unsteady experimental data on the interference between the stator and rotor rows of a transonic turbine stage allowed a careful testing of an unsteady flow solver. The various configurations differ in the gap in between the stator and rotor rows and in the mass flow rate of the pressure side coolant ejection in the stator blade. The computed steady and unsteady average isentropic Mach number distribution show very little differences for all the configuration reported here. The experiments show some interaction between the stator wake, and shock pattern, and the rotor leading edge pressure distribution which is slightly underestimated by the predictions. The measurements do not include any flow angle traversing immediately upstream of the rotor leading edge, but the numerical simulation proved that the isentropic Mach number distribution moves towards the right direction when applying a positive incidence angle. This corresponds to the flow status immediately upstream of the stator trailing edge shock, where the absolute velocity is maximum. Conversely, the rotor experiences a negative incidence angle deep in the wake so that the pressure distribution range, especially in proximity to the leading edge, is governed by the mutual effect of the wake and of the shock. Part of the inaccuracy of the predictions in the rotor leading edge area could be induced by the interaction between the stator trailing edge shock impinging on the rotor leading edge and the stator wake. Apparently the impact of the artificial dissipation, which is undeniably responsible of some of the numerical diffusion, could not be controlled by decreasing the artificial dissipation weights.

The heat transfer rate, here computed in terms of Nusselt number, changes dramatically from a steady to an average unsteady case. The large number of unsteady computations performed indicate that the Nusselt number profile on the rotor blade is not highly sensitive to the length of the gap between the stator and the rotor and to the mass flow rate of the coolant jet. The transitional nature of the rotor boundary layer seems to be governed mainly by the stator trailing edge shocks and static pressure unsteadiness for the leading edge region of the
rotor. When the stator blade shock does not impinge on the rotor blade, the boundary layer is mostly governed by the large turbulence level convected downstream of the stator row. This analysis is confirmed by the underprediction of the computed Nusselt number with respect to the measurements in the first 20% of the rotor blade where the effect of the stator shocks is large. The agreement progressively improves while moving downstream along the rotor blade surface. In this flow region, the effect of turbulence (the turbulence level reaches values or the order of 30%) on the boundary layer status and heat transfer rate is very large and the effect of the stator shocks fades away. The heat transfer rate predictions, and experiments, indicate that the transition point travels downstream and this presumably follows the high turbulence level spot induced by the stator wake.

In conclusion, the comparison between predictions and measurements definitely proves that the heat transfer rate on a rotor blade cannot be realistically computed by assuming a steady flow condition, and a full unsteady computation is compulsory. The adopted turbulence and transition models, although not specifically developed and/or tuned for this test case proved adequate for the determination of the unsteady heat transfer level with a reasonable degree of accuracy. It is necessary to observe that the computational model was validated so far only for steady flow situations. The computations also showed that the aerodynamic field and the heat transfer are strictly linked and that some of the inaccuracy in the predictions of the former have an impact on the latter.

ACKNOWLEDGMENTS
This research was sponsored by the EEC in the frame of the BRITE-EURAM Program, IMT Area 3 Turbine Project, Contract AER2-CT92-0044. The support of EEC is gratefully acknowledged.

REFERENCES