NUMERICAL CALCULATION OF THREE-DIMENSIONAL EULER FLOW THROUGH A TRANSONIC TEST TURBINE STAGE

Harald Paßnicker, Wolfgang Sanz, Herbert Jericha
Institute of Thermal Turbomachinery and Machine Dynamics
Graz University of Technology
A-8010 Graz, AUSTRIA

ABSTRACT
The objective of this work is the development of an implicit block-structured central-differencing scheme with implicit treatment of the boundary conditions for the calculation of three-dimensional inviscid compressible steady flow, primarily through transonic turbomachinery stages. The implicit boundary treatment is based on the mathematical theory of characteristics for hyperbolic systems of equations. Steady-state non-reflecting boundary conditions at the inflow and outflow boundaries and at the stator/rotor interface based on Fourier analysis applied to the linearized Euler equations are also used. Getting a monotonous resolution of shocks, a non-linear additive viscosity term of 2nd and 4th order with pressure-controlled factors is implemented. The three-dimensional blade geometry consists of Bézier-surfaces which are determined by two-dimensional blade profiles based on Bézier-curves. The grid for the flow calculation is obtained by parameterizing special Bézier-surfaces, respectively Bézier-volumes. Finally, the aerodynamic design of the one-stage highly loaded transonic test turbine of Institute of Thermal Turbomachinery and Machine Dynamics - Graz University of Technology (with these tools described above is illustrated with detailed presentation of the specific aerodynamic features, the design technology and the computational results.

INTRODUCTION
Basic flow processes in turbomachines are characterised by high Reynolds numbers. They are dominated by convection effects and hence they can be described at first approximation by the solution of the Euler equations. Though they do not cover the influence of viscosity, they still enable the analysis of flows with discontinuities characteristic for transonic flows which commonly occur in highly-loaded compression and expansion cascades (compressors and turbines). It should also be emphasised, that the currently used methods for the solution of inverse problems are based principally on the multiple application of an Euler solver (the use of Navier-Stokes equations for geometry optimisation of flow passages in practical designing will have to wait until the future). Accordingly, the use of the high accuracy algorithms should enable the turbomachinery designers to improve blade design.

In the field of industrial gas turbines, the specific power output is rising up rapidly with increasing turbine inlet temperature by adopting advanced cooling systems, and also with increasing of cycle pressure ratio to get higher performance. On the other hand, it is not desirable to increase the number of turbine stages, especially air-cooled stages, in view of cost advantage. Therefore it is necessary to develop a highly loaded turbine stage without deterioration of performance in order to satisfy the conditions described above. A higher stage loading inevitably requires a relatively high Mach number level and high deflection angle. These requirements are determining to the aerodynamic performance of nozzles and blades. To investigate the turbine performance operating in such severe conditions, a research program for air-cooled high pressure turbines with high stage loading is funded by the Austrian Science Foundation (FWF). For this reason a new turbomachinery test stand is built to collect high quality - experimental data within a turbine stage. The purpose is to produce a data base for the validation of CFD codes and to provide a platform for the investigation of highly loaded turbines operating in transonic regime.

Presented at the International Gas Turbine & Aeroengine Congress & Exhibition
NUMERICAL METHOD

GOVERNING EQUATIONS

Using non-dimensional variables, the three-dimensional unsteady Euler equations in conservative form can be written for a rotating blade passage in a curvilinear coordinate system $\xi$, $\eta$, $\zeta$ as:

$$\frac{\partial \vec{U}}{\partial t} + \frac{\partial \vec{E}}{\partial \xi} + \frac{\partial \vec{F}}{\partial \eta} + \frac{\partial \vec{G}}{\partial \zeta} + \vec{H} = 0 \tag{1}$$

where the Cartesian system $x$, $y$, $z$ is rotating with angular velocity $\Omega$ around the $x$ axis. $\vec{U}$ is the vector of conservative variables, $\vec{E}$, $\vec{F}$, $\vec{G}$ are the flux vectors in $\xi$, $\eta$, $\zeta$-direction and $\vec{H}$ is the rotational source term.

$$\vec{U} = \begin{pmatrix} p \\ \rho u \\ \rho v \\ \rho w \\ c_t \end{pmatrix}, \quad \vec{E} = \begin{pmatrix} \rho U' \\ \rho u U' + \xi_p \\ \rho v U' + \eta_p \\ \rho w U' + \zeta_p \\ \rho c_t \end{pmatrix}, \quad \vec{F} = \begin{pmatrix} \rho V' \\ \rho u V' + \eta_p \\ \rho v V' + \xi_p \\ \rho w V' + \zeta_p \\ \rho c_t \end{pmatrix}, \quad \vec{G} = \begin{pmatrix} \rho W' \\ \rho u W' + \xi_p \\ \rho v W' + \eta_p \\ \rho w W' + \zeta_p \\ \rho c_t \end{pmatrix}, \quad \vec{H} = \begin{pmatrix} eU' + pU \\ eV' + pV \\ eW' + pW \\ \rho u' + \xi_p \\ \rho v' + \eta_p \\ \rho w' + \zeta_p \end{pmatrix}$$ \tag{2}

$U'$, $V'$, $W'$ are the relative and $U$, $V$, $W$ the absolute contravariant velocity components, and $J$ is the Jacobian of the transformation. $\rho$, $u$, $v$, $w$, $c$ denote respectively density, the absolute velocity components in the $x$, $y$ and $z$ Cartesian direction and the specific total energy per volume. Pressure and temperature are obtained from the calorical and the thermal equation for an ideal gas with constant ratio $\kappa$ of the specific heat capacities $c_p$ and $c_v$, as follows

$$p = (\kappa - 1) \left[ e - \frac{1}{2} \rho (u^2 + v^2 + w^2) \right] \quad T = \frac{(\kappa - 1)e}{R} \tag{3}$$

BOUNDARY CONDITIONS

A particular choice or combination of boundary conditions can have a considerable influence on the accuracy and stability of the computational scheme. The most accurate numerical boundary conditions for Euler equations are characteristic compatibility relations. They guarantee a low entropy error and an accurate convection of total pressure at the boundaries. Furthermore, numerical calculations showed, that they improve the robustness of the procedure remarkably, especially, if skewed grids or short inlet and outlet regions are used. We followed the concept of Chakravarthy [2], which couples the compatibility relations and the time differenced physical boundary conditions, leading to a boundary condition matrix formulation. A physical boundary condition $B_m$ written as a time derivation yields:

$$\frac{\partial}{\partial t} B_m = \frac{\partial}{\partial \xi} B_m = \frac{\partial}{\partial \eta} B_m = \frac{\partial}{\partial \zeta} B_m = \frac{\partial}{\partial t} \vec{U} = \frac{\partial}{\partial \zeta} \vec{B}$$ \tag{4}

The compatibility relations are linear combinations of the conservation equations. With $M_{B_m}$ as the $B_m$ row eigenvector of the left eigenvector matrix $L_q$ of the Jacobian-matrix $A = \partial \vec{E}/\partial \vec{U} \cdot \vec{U}$ one obtains e.g. for a surface $\Sigma$=constant

$$\left( L_q \right) \left( \frac{\partial}{\partial t} \vec{U} + \frac{\partial}{\partial \xi} \vec{E} + \frac{\partial}{\partial \eta} \vec{F} + \frac{\partial}{\partial \zeta} \vec{G} + \vec{H} \right) = 0 \tag{5}$$

The combination of these two equations (4) and (5) gives

$$M_i \frac{\partial}{\partial t} \vec{U} + M_2 \left( \frac{\partial}{\partial \xi} \vec{E} + \frac{\partial}{\partial \eta} \vec{F} + \frac{\partial}{\partial \zeta} \vec{G} + \vec{H} \right) = \vec{F}_b$$ \tag{6}

where $M_i$ contains $m$ physical boundary conditions and $n=5-m$ left eigenvectors. $M_2$ is similar to $M_i$, with zeros replacing the physical boundary conditions. $\vec{F}_b$ contains the physical values $B_m$ of the $m$ boundary conditions and zeros. Thus the physical boundary conditions $B_m$ are received asymptotic. Multiplying (6) by $M_i^{-1}$ gives following equation with $M=M_i^{-1} M_2$ and $\vec{F}_b=M_i^{-1} \vec{F}_b$:

$$\frac{\partial}{\partial t} \vec{U} + M \left( \frac{\partial}{\partial \xi} \vec{E} + \frac{\partial}{\partial \eta} \vec{F} + \frac{\partial}{\partial \zeta} \vec{G} + \vec{H} \right) = \vec{F}_b$$ \tag{7}

The Matrix $M$ is equal to the unit matrix in the interior and at periodic boundaries, while at inlet, outlet and solid wall boundaries $M$ represents the physical and numerical boundary conditions. As physical boundary conditions for subsonic flow we choose total temperature, total pressure and the flow directions at inlet and static pressure at outlet. For supersonic flow $u$ is given additionally at inlet and no physical boundary condition at outlet. At solid walls the flow must be tangential.

It is now common practice in turbo-machinery CFD to use non-reflecting boundary conditions which allows waves to pass through inflow and exit planes without spurious reflections which can contaminate solutions. For non-steady flow approximate two-dimensional boundary conditions have been developed by Giles [4] based on eigenvector decomposition of the linearized Euler equations. Saxer and Giles [6] have extended this formulation to the third dimension.

The stator-rotor interaction is performed by transferring the mass flow circumferentially averaged values of $T_i(r), p_i(r), a_i(r)$ and $\beta_i(r)$ from stator to rotor and the area circumferentially averaged value of static pressure $p(r)$ from rotor to stator. By this, the solution vector at the coupling plane is calculated similarly to the boundary treatment at inlet and outlet, respectively, using the theory of characteristics. Non-reflecting boundary conditions at the stator-rotor interface allow modelling of a small axial gap between the stator and rotor.

SOLUTION METHLOGY

The governing equations are solved using the implicit approximate factorisation method of Beam and Warming [1]. After implicit time
differencing, linearization, differentiation with 2nd order central spatial differences $\delta$, addition of artificial dissipation terms $D$ and factorisation one obtains the discretized equation

$$
\begin{align*}
N_i' + \Delta M' \delta_i \left( A_i + D_{m+1} \right) N_i' & \\
N_i' + \Delta M' \delta_i \left( B_i + D_{m+1} \right) N_i' & \\
N_i' + \Delta M' \delta_i \left( C_i + D_{m+1} \right) \bar{U}_i' & \\
= -\Delta M' \left( \delta_i \bar{E}_i' + \delta_i \bar{F}_i' + \delta_i \bar{G}_i' + \bar{H}_i' + D_{m+1} \bar{U}_i' \right) + \bar{F}_i'
\end{align*}
$$

(8)

where $N'_i = \mathbf{I} + \Delta M' \mathbf{D}'$

(9)

$\mathbf{A}$, $\mathbf{B}$, $\mathbf{C}$ and $\mathbf{D}$ are the Jacobians of the flux vectors $\bar{E}$, $\bar{F}$, $\bar{G}$ and of the rotational source term $\bar{H}$. For steady state calculations with a time-marching approach, a faster expulsion of disturbances can be achieved by locally using the maximum timestep:

$$
\Delta t = \frac{CFL}{U + V + W + \alpha (\xi + \eta + \zeta)}
$$

(10)

By using a smaller time step on the implicit part of equation (9) the factorisation error is reduced and larger explicit time steps can be used:

$$
\Delta t_{\text{max}} = \omega \cdot \Delta t_{\text{exp}}
$$

(11)

With an optimum value of $\omega = 0.6$ the convergence speed could be doubled.

**NON-LINEAR DISSIPATION**

In contrast to upwind schemes, which provide enough internal damping, the central spatial discretization (and the three-dimensional factorisation) needs some additional damping terms for stability. Furthermore, an accurate and oscillation-free shock resolution can only be obtained by non-linear damping terms. A non-linear 2nd and 4th order blended dissipation model similar to Pulliam [5] is used. The explicit part is equal:

$$
D_m = \frac{1}{\Delta t} \left[ \frac{1}{J} \right] \left( \epsilon_{m-1}^{2} \Delta_i \Delta_j \epsilon_{m-1}^{2} \Delta_j - \epsilon_{m-1}^{2} \Delta_i \epsilon_{m-1}^{2} \Delta_j \right)
$$

(12)

with the non-linear coefficients:

$$
\epsilon_{m-1}^{2} = \epsilon_{m-1}^{2} + \Delta_{m-1} \max(\gamma, \gamma_{m-1})
$$

(13)

$$
\epsilon_{m-1}^{2} = \max(\epsilon_{m-1}^{2} \Delta_{m-1} \epsilon_{m-1}^{2} - \epsilon_{m-1}^{2})
$$

(14)

and the shock sensor and one-dimensional spectral radius:

$$
\gamma = \left[ \frac{p_{m-2} + p_{m-1}}{p_{m-1} + p_{m}} \right] \sigma_{\Delta} = |U| + a \sqrt{\xi_{y} + \xi_{y} - \xi_{x}^2}
$$

(15)

The implicit artificial dissipation terms are equal to:

$$
D_m = -\epsilon_{m-1}^{2} \Delta_i \left( \frac{1}{J} \right) \Delta_j
$$

(16)

$\Delta^{l}$ and $\Delta^{r}$ are first order forward and backward difference operators, the values at $i+1$ are arithmetic mean values of the value at $i$ and $i+1$ and $\epsilon_{m-1}^{2} = 0.04$, $\epsilon_{m-1}^{2} = 0.04$, 1.0 and $\epsilon_{m-1}^{2} = 0.25$ are optimum input values for stability, accuracy and convergence. The shock wave capturing capability is 4-5 nodes with this method.

The characteristic formulation of the boundary conditions needs artificial damping terms at the boundaries. At periodic boundaries the same damping terms can be used as in the interior as well as in tangential direction of inlet-, outlet- and solid wall boundary surfaces. Modifications are necessary for the latter boundaries in the normal direction, where one-sided differences are used. At the boundary- surface $i=1$ the fluxes in normal direction are discretized by one-sided differences, which provide enough internal dissipation. Therefore no additional viscosity terms are needed in normal direction. For a neighboring surface $i=2$ we extrapolate the flow for $i-2$ and use the same formulation (12) as in the interior flow field.

**COMPUTATIONAL GRID**

It is well known, that the truncation error increases with the mesh obliqueness and the local stretching. Therefore the quality of the grid is very important for accuracy. We developed a procedure for generating various grid types where special Bézier-surfaces respectively Bézier-volumes are parameterised [3]. The parameter lines of these special Bézier-surfaces respectively Bézier-volumes intersect the contour under a given angle. This seems very important to get orthogonal body fitted grid lines.

**AERODYNAMIC DESIGN OF THE TRANSONIC TEST TURBINE STAGE**

**DESCRIPTION OF TEST STAND**

The test stand consists of the test turbine, the brake compressor and the compressor station (two turbo compressors and one screw compressor, installed capacity 3 MW). The test turbine will be a 2 MW single-stage transonic unit driven by the air supply from the brake compressor and the compressor station. The arrangement is shown schematically in figure I.
**MERIDIAN CONTOUR AND PROFILE GEOMETRY**

The first step in the aerodynamic design of the transonic test turbine stage was a one-dimensional calculation (according Traupel [8]) on five blade sections to get the boundary conditions at inlet and outlet of the stage, the area ratio and the radial distribution of the flow angles. This one-dimensional calculation was done with the condition of constant exit flow angle of the guide vanes and with constant specific work over the rotor blade height. Besides, all components of the test stand (test turbine, brake compressor, compressor station) have to run at a stable operating point. The operating point of the test turbine is at 10500 rpm with a specific work of 105.2 kJ/kg and a mass flow of 18.1 kg/s. Figure 2 shows the meridian contour of the test turbine stage, which consists of 29 guide vanes and 36 rotor blades.

The inner contour is cylindrical for an easy fabrication and can also be used as reflection plane for optical visualization methods. The outer contour consists of straight lines and an arc to make it possible to implement windows for optical measurement methods. The planes and the span-wise constant values of the boundary conditions for the CFD calculation are also illustrated in figure 2 (0—inlet, 1—rotor/stator interface, 2—outlet).

Because of high turbine inlet temperature exceeding 1000°C, guide vanes and rotor blades of high pressure stage are air-cooled. The design of an air-cooled high pressure turbine stage differs from that of an ordinary non-cooled turbine, for instance the thickness of the leading-edge and trailing-edge becomes larger than that of an ordinary type blade for the cooling system.

**Guide vane:**

Considering high guide vane exit velocity, a straight back guide vane was designed, having large trailing-edge thickness for trailing-edge cooling system. The profiles of the guide vane at hub/mid/tip are shown in figures 3, 4, 5. The whole guide vane is illustrated in figure 6. The guide vane is slightly twisted angle to compensate the influence of the meridian contour and to get a nearly constant exit flow. The blade is defined by two profiles, one at the hub and one at the tip. The individual profiles consist of two Bézier-curves of 12th order for the suction and pressure side and one Bézier-curves of 3rd order for the round trailing edge. These profiles determine three Bézier-planes of first order in radial direction. The profiles in figure 4 are parameter curves of the Bézier-curves in the mid section. More details about designing blade profiles with Bézier-curves and three-dimensional blades with Bézier-planes are described in reference [3].

**Rotor blade:**

The requirement for higher stage loading is a high deflection angle of the rotor blades. If the blade has high reaction and a relatively small deflection angle, it is easy to construct the converged flow passage. But in the case of the newly designed blade, it is difficult to construct the fully converged passage owing to several characteristics, such as...
low reaction, high deflection angle, relatively thin blade thickness to reduce the centrifugal stress and large leading-edge radius for the cooling system. The profiles of the rotor blade at hub/mid/tip are also shown in figures 3, 4, 5. The rotor blade is of course twisted and is also defined by two profiles, one at the hub and one at the tip, which determine the whole blade consisting of Bézier-planes.

Finally the guide vanes and rotor blades are intersected with the meridian contour to get the final geometry of the bladings.

**CFD RESULTS AND DISCUSSION**

In figure 7 a three-dimensional view of the computational HO-type grids of the test turbine stage is shown. The guide vane section is formed by 117x30x21 and 141x6x21 nodes, the rotor blade section by 131x30x21 and 157x6x21 nodes. This calculation requires about 250 minutes of CPU time (~500 steps, CFL=10) on an Alphastation 500/500 to achieve a four decades' reduction in the residuals and it needs 117 MB working memory.

The boundary conditions for the calculation are illustrated in figure 2. Basically the flow through guide vanes and rotor blades is transonic. There are following distributions of guide vane outlet pressure, absolute Mach number, relative Mach number and flow angles in plane 1:

<table>
<thead>
<tr>
<th>rel. height [-]</th>
<th>0 (HUB)</th>
<th>0.25</th>
<th>0.5 (MID)</th>
<th>0.75</th>
<th>1 (TIP)</th>
</tr>
</thead>
<tbody>
<tr>
<td>p [bar]</td>
<td>1.29</td>
<td>1.44</td>
<td>1.58</td>
<td>1.71</td>
<td>1.83</td>
</tr>
<tr>
<td>M_{in} [-]</td>
<td>1.23</td>
<td>1.15</td>
<td>1.08</td>
<td>1.01</td>
<td>0.97</td>
</tr>
<tr>
<td>M_{in} [-]</td>
<td>0.71</td>
<td>0.62</td>
<td>0.52</td>
<td>0.44</td>
<td>0.40</td>
</tr>
<tr>
<td>(\alpha) [°]</td>
<td>68.9</td>
<td>69.1</td>
<td>69.4</td>
<td>69.0</td>
<td></td>
</tr>
<tr>
<td>(\beta) [°]</td>
<td>51.6</td>
<td>48.0</td>
<td>42.8</td>
<td>35.8</td>
<td>27.1</td>
</tr>
</tbody>
</table>

The outlet pressure, absolute Mach number, relative Mach number and flow angles after the rotor blade in plane 2 are:

<table>
<thead>
<tr>
<th>rel. height [-]</th>
<th>0 (HUB)</th>
<th>0.25</th>
<th>0.5 (MID)</th>
<th>0.75</th>
<th>1 (TIP)</th>
</tr>
</thead>
<tbody>
<tr>
<td>p [bar]</td>
<td>1.102</td>
<td>1.102</td>
<td>1.102</td>
<td>1.102</td>
<td>1.102</td>
</tr>
<tr>
<td>M_{in} [-]</td>
<td>0.42</td>
<td>0.40</td>
<td>0.37</td>
<td>0.38</td>
<td>0.41</td>
</tr>
<tr>
<td>M_{in} [-]</td>
<td>0.84</td>
<td>0.87</td>
<td>0.90</td>
<td>0.93</td>
<td>0.98</td>
</tr>
<tr>
<td>(\alpha) [°]</td>
<td>-21.3</td>
<td>-19.5</td>
<td>-17.5</td>
<td>-13.9</td>
<td>-9.3</td>
</tr>
<tr>
<td>(\beta) [°]</td>
<td>-62.5</td>
<td>-64.5</td>
<td>-66.5</td>
<td>-67.2</td>
<td>-65.9</td>
</tr>
</tbody>
</table>

Figure 8 shows corresponding velocity triangles for three characteristic section: hub, mid and tip.
The values above are circumferentially averaged values and the maximum values are much higher. This is the reason of producing a system of oblique shock waves at the round trailing edge of the guide vane and rotor blade. Another phenomena are the extreme high velocities at the rotor blade suction side after the leading edge as a consequence of the channel passage contraction at the inlet area. Figures 9, 10, 11 show static pressure contours at hub, mid and tip section. The grading of the pressure contour lines is 0.1 bar.

The flow is strongly accelerated in the guide vane channel. The highest velocities and thus lowest pressures are in the hub section in consequence of the nearly constant radial outflow angle in combination with the smallest radius (radial equilibrium). This is also manifested by stronger oblique shock waves outgoing from the round trailing edge of the pressure side. The shock waves are reflected on the suction side and decelerate the flow along the wall strongly. This is the reason to design the contour in this area as straight as possible to avoid flow separation.

The strength of the oblique shock waves cause different values of total pressure losses, too (hub-6.2%, mid-4.8%, tip-3.4%). The flow...
through the rotor blade channel is different along the radius with
differences in acceleration and deflection due to the rotation. This
results in following degrees of reaction: hub-0.13, mid-0.31 and
tip-0.43. There is high acceleration with following deceleration in the
inlet area of the suction side at hub section (low pressure - figure 9) in
consequence of the high inlet Mach number, the channel contraction
and the low reaction. The total pressure losses are different along the
radius (hub-6.2%, mid-4.8%, tip-3.4%) caused by the oblique shock
waves at the round trailing edge and the high Mach number zone on
the suction side. Figures 12 illustrates the divergence of streamlines on
suction and pressure side. The curvature of the streamlines is a
function of the local pressure gradient, which substantially depends on
the normal vector of the blade surface.

Figure 12: Special streamlines on suction and pressure side

For these closely spaced blades, the formulation of the boundary
conditions at rotor/rotor interface becomes a keypoint in the numerical
calculation. Typically, most of the codes available today are not
capable preventing spurious, non-physical reflections at inflow and
outflow boundaries. This leads to erroneous performance predictions,
since the calculated flow field is dependent on the position of the far-
field boundary condition. Figure 13 shows the pressure distribution at
outflow stator (plane 1), which is adjusted through the non-reflecting
boundary condition. The averaged radial pressure at outflow stator is
received by averaging out the radial pressure at inflow rotor. By the
way of contrast the pressure at outflow stator would be constant in
circumferential direction with normal reflecting boundary conditions.

Figure 13: PLANE 1 - static pressure contours

Figure 14 shows guide vane Mach number distributions at different
radial positions dependent on the relative height. The flow is
continuous accelerated to supersonic on the pressure side. This leads to
oblique shock waves outgoing of the round trailing edge over the
whole blade height. The oblique shock waves spread through the flow
channel and are reflected on the suction side. This causes a
deceleration of the flow with total pressure losses.

Figure 14: Guide vane profile Mach number distributions

Figure 15 distinctly displays rotor blade profile Mach number
distributions at different radial positions with high Mach numbers
(over 1.4) at hub on the suction side after the leading edge. The flow is
afterwards decelerated to subsonic due to the channel effect and again
accelerated to supersonic flow. The subsequent decrease over the
whole blade height is caused from the shock waves outgoing from the
pressure side of the round trailing edge. The high Mach numbers on
the suction side extend to the half blade height (see also figure 16).

Figure 15: Rotor blade profile Mach number distributions

Figure 16 illustrates the Mach number distribution on suction and
pressure side of guide vane and rotor blade. The grading of Mach
number contour lines is 0.05. Clearly you can see the extreme high
velocities (Mach number) on the suction side of the rotor blade with
subsequent deceleration. Furthermore it shows the reflection of the
shock waves on the suction side of guide vane and rotor blade
outgoing from the round trailing edges of the pressure side visible by the strong gradient of the contour lines.

Due to the strong deceleration in the inlet area of the suction side (downstream a high acceleration) the boundary is probably laminar and will separate. If this separation occurs the flow in this region will not only be dominated by convection effects but also by viscous effects. But a Navier-Stokes solver will also have troubles in prediction the separation length. The suction side of the rotor blade is not only in danger through the steady-state calculated high deceleration but also through the insteady behavior of the stage flow at all. In particular, the impact of the upstream guide vane oblique shock on the downstream rotor blade suction surface which produces reflected waves on both the adjacent rotor and on the upstream stator is critical. It has been observed that the shock striking the rotor suction surface does cause a temporary boundary-layer separation (Saxer and Giles [6]). These separation bubbles convect downstream along the blade surface and subsequently collapse into turbulent flow which increase the heat transfer rate and the viscous losses. It will be interesting to study the complex flow of this high-loaded turbine stage in nature.

The question how valuable is a steady-state calculation for a total unsteady flow shows a comparison between steady-state and unsteady calculations, that the steady-state results are extremely close to the time-averaged values of a unsteady calculation (Saxer and Giles [6]). For instance, the maximum variation in the blade static pressure between the steady-state and the time-averaged solution is less than 4% of the stator inlet stagnation pressure, a value 15 times smaller than the maximum local unsteady fluctuation. The agreement is even better for integral values such as the axial torque, with a difference less than 0.5% of the time-averaged value.

CONCLUSION

A numerical method for the solution of the three-dimensional Euler equations in transonic turbine stages is presented. It is possible to obtain a convergent solution reasonable computational costs as consequence of the implicit treatment of the boundary conditions and the fast grid generation with special Bézier-volumes. So it was possible to make various calculations for the design of the institute's test turbine. The Euler solution clearly shows the difficulties of highly loaded transonic stages like shock waves outgoing from the round trailing edges and high local velocities on the suction side of rotor blades with low reaction.

ACKNOWLEDGEMENT

The authors gratefully acknowledge the support of the Austrian Science Foundation (FWF) of this work and of ongoing research in efficiency improvement of thermal power plants.

REFERENCES