ABSTRACT

The differences of two distinct numerical schemes implemented in one code called ITSM3D are presented for a turbine stage test case. Thus both schemes are used with exactly the same computational infrastructure, e.g. same grids, boundary conditions, acceleration strategies, time-stepping, turbulence model etc. The two methods are based on an explicit Runge-Kutta-type finite volume scheme expressed in cylindrical coordinates and have been developed at the Institut für Thermische Strömungsmaschinen und Maschinenlaboratorium of the University of Stuttgart. One scheme is a node centered 3rd order TVD scheme according to Osher and the other belongs to the cell vertex central difference type with the concept of artificial viscosity. The model of Baldwin-Lomax is used in order to accelerate the computation.

The test case for this comparison is the last stage of a low-pressure turbine. The computational results obtained are discussed and compared to each other as well as to experimental data. They are presented as pressure and Mach number isoline contours and diagrams of circumferential averaged quantities at inlet and outlet planes of stator and rotor versus radial position.

GOVERNING EQUATIONS

The underlying viscous conservation law - the set of three-dimensional, unsteady, mass-averaged Navier-Stokes equations can be written in nondimensional variables and for a rotating cylindrical coordinate system \((x, \varphi, r)\) as

\[
\frac{\partial \mathbf{Q}}{\partial t} + \nabla \cdot \mathbf{F} = \mathbf{S},
\]

where the cell faces are defined as \(A_x = r \, dx \, d\varphi\), \(A_{\varphi} = dz \, dr\) and \(A_r = dz \, r \, d\varphi\) which enclose the control volume \(V\).

The vector of conserved quantities \(\mathbf{Q}\), the flux vectors \(\mathbf{F}_x, \mathbf{F}_{\varphi}\) and \(\mathbf{F}_r\), and the source term vector \(\mathbf{S}\), respectively, are denoted by

\[
\mathbf{Q} = \begin{bmatrix} 0 \\ \rho w_x \rho w_{\varphi} \rho w_r \end{bmatrix}, \quad \mathbf{F}_x = \begin{bmatrix} \rho w_x \rho w_{\varphi} \rho w_r \end{bmatrix}, \quad \mathbf{F}_r = \begin{bmatrix} \rho w_r w_r + p + \tau_{xx} \\ \rho w_r w_{\varphi} + \tau_{x\varphi} \\ \rho w_r w_r + \tau_{rr} \end{bmatrix}, \quad \mathbf{S} = \begin{bmatrix} \rho w_x \rho w_{\varphi} \rho w_r \end{bmatrix}, \quad \mathbf{S}_x = \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix}.
\]

The vector of conserved quantities \(\mathbf{Q}\) is built by the primitive variables density \(\rho\), components of the relative velocity vector \(w_x, w_{\varphi}, w_r\) and the relative specific total energy \(e_t\). \(\Omega\) denotes the angular velocity for rotating around the \(z\)-axis. The total enthalpy is given by \(h_t = e_t + \frac{p}{\gamma - 1}\). Assuming an ideal gas with constant ratio of specific heats \(\gamma\), pressure \(p\) and temperature \(T\) are defined by

\[
p = (\gamma - 1) \left[ e_t - \frac{\gamma}{2} (u_x^2 + u_{\varphi}^2 + u_r^2 - \Omega^2 r^2) \right], \quad T = \frac{e_t}{\gamma - 1}.
\]

For a Newtonian fluid the shear-stress tensor is given by

\[
\tau = -\mu \left( \frac{\partial u_x}{\partial x} + \frac{\partial u_{\varphi}}{\partial \varphi} + \frac{\partial u_r}{\partial r} + \frac{\partial u_{\varphi}}{\partial x} + \frac{\partial u_r}{\partial \varphi} + \frac{\partial u_{\varphi}}{\partial r} \right)
\]

with

\[
\lambda = \frac{2}{3} \left( \frac{\partial u_x}{\partial r} + \frac{\partial u_r}{\partial x} + \frac{\partial u_{\varphi}}{\partial \varphi} \right) + \Lambda.
\]

The effects of shear-stress and heat-transfer in the energy equation are considered by

\[
\begin{align*}
q_x &= \tau_{xx} w_x + \tau_{x\varphi} w_{\varphi} + \tau_{xx} w_r - k \frac{\partial T}{\partial x}, \\
q_{\varphi} &= \tau_{x\varphi} w_x + \tau_{\varphi\varphi} w_{\varphi} + \tau_{x\varphi} w_r - k \frac{\partial T}{\partial \varphi}, \\
q_r &= \tau_{xx} w_x + \tau_{r\varphi} w_{\varphi} + \tau_{rr} w_r - k \frac{\partial T}{\partial r}.
\end{align*}
\]

Presented at the International Gas Turbine and Aeroengine Congress & Exhibition

Birmingham, UK — June 10-13, 1996
By taking the eddy-viscosity hypothesis to simulate turbulence, the viscosity \( \mu \) and the thermal conductivity \( k \) in equation (4) are divided into a laminar and a turbulent part. The laminar viscosity \( \mu_0 \) is described by sutherland's law whereas the turbulent viscosity \( \mu_t \) is computed with a modified Baldwin and Lomax (1978) turbulence model. Spanwise and blade-to-blade contributions of turbulent viscosity \( \mu_t \) are computed separately (Arnone (1993)). The end wall shear layers are considered to be fully turbulent, whereas the blade surface shear layers are handled with the transitional criterion of Baldwin and Lomax. A detailed description can be seen from Merz et al. (1992, 1995).

**DISCRETIZATION**

The flow solver ITSM3D uses two different schemes to discretize the Navier-Stokes equations (1). Both are based on a finite volume approach. The first, a 3rd order TVD upwind scheme (TVD) uses the same node centered control volume for convective and diffusive fluxes, see Figure 1, whereas the central difference scheme (CD) makes use of different control volumes. Here the convective fluxes are balanced in combined cell vertex control volumes according to the grid section in Figure 1. The metric calculations for the control volumes and cell faces are done in the same way. For more detail see Merz et al. (1992, 1995). At boundaries different control volumes are used. For CD phantom cells are combined with boundary control volumes. TVD uses different control volumes for walls (half cells) and for other boundaries (half cells combined with phantom half cells). In CD-case 1st order extrapolation and in TVD-case 0 and 1st order extrapolations are used to obtain values for the phantom cells.

Figure 1: Grid with node centered control volume (dashed lines)

The semi-discrete form of equation (1) is written as

\[
\frac{\partial}{\partial t} \tilde{Q}_{i,j,k} + \nabla \cdot \left( \tilde{F}_{i,j,k} \right) = S_{i,j,k},
\]

where \( \tilde{F} \) and \( \tilde{F}_d \) account for the convective, and the diffusive terms, respectively, including their corresponding source terms. For central differencing \( \tilde{F}_{CD} \) is the artificial dissipation and \( \tilde{C}_{CD} \) is a constant correction term minimizing errors caused by the numerical grid and the source terms. The values of the solution vector are stored for the vertices of the curved hexahedrons for CD and the nodal centers for TVD.

**CENTRAL DIFFERENCE SCHEME**

Only a short outline of the numerical method for the CD scheme is given below. For more detail see Merz et al. (1992, 1995). The balancing method is based on the work of Radespiel et al. (1989). In order to stabilize the scheme Jameson's (1981) artificial dissipation method is used. It has been extended with Martinelli's (1987) and Radespiel's (1989) formulated scaled dissipation which minimizes the artificial dissipation.

All other techniques used such as boundary treatment, multi-grid method, implicit residual smoothing etc. are identical for both schemes.

**TVD UPWIND SCHEME**

In the TVD part of the code ITSM3D three different schemes can be chosen: Roe's scheme (see Chakravarthy/Szema (1985)), Osher's (1983) scheme and the Steger/Warming (1981) split flux technique, see also Raif et al. (1994). The TVD scheme presented in this paper is based on the method of Osher (1983). The upwind method is applied to the convective part of the equation (7) corresponding to the Euler equations. In Osher's approximate Riemann solver the velocity vector \( \tilde{w} \) is transformed to \( \tilde{w} \) referring to a cell face orthogonal system (here for the \( x \)-direction):

\[
\tilde{w} = \left( \begin{array}{c}
\tilde{n}_x \\
\tilde{n}_y \\
\tilde{n}_z \\
\end{array} \right) \cdot \tilde{w}.
\]

\[
\tilde{w} = \left( \begin{array}{c}
\tilde{n}_x \\
\tilde{n}_y \\
\tilde{n}_z \\
\end{array} \right) \cdot \tilde{w}.
\]

with the normalized cell face normal vector \( \tilde{n} = (\tilde{n}_x, \tilde{n}_y, \tilde{n}_z)^T \) and \( \tilde{n}_i^2 = \tilde{n}_x^2 + \tilde{n}_y^2 + \tilde{n}_z^2 \). A general formulation for the convective flux vector \( \tilde{F} \) for all directions \( m \)

\[
\tilde{F}_m = \left( \begin{array}{c}
\tilde{u}_{m} \\
\tilde{v}_{m} \\
\tilde{w}_{m} \\
\end{array} \right),
\]

is used in Osher's flux. In the following the subscript \( m = \frac{1}{2} \) abbreviates the index \( (i-\frac{1}{2}, j, k) \) for the \( x \)-direction. The Osher flux can be written as:

\[
\tilde{F}^{Osher}_{m-\frac{1}{2}} = \frac{1}{2} \left[ \tilde{F}_{i-\frac{1}{2},j,k} + \tilde{F}_{i+\frac{1}{2},j,k} - \int_{\tilde{Q}_{i-\frac{1}{2},j,k}} \left( \frac{\partial F^+}{\partial Q} - \frac{\partial F^-}{\partial Q} \right) d\tilde{Q} \right].
\]

This is equivalent to

\[
\tilde{F}^{Osher}_{m-\frac{1}{2}} = \int_{\tilde{Q}_{i-\frac{1}{2},j,k}} \left( \frac{\partial F^+}{\partial Q} - \frac{\partial F^-}{\partial Q} \right) d\tilde{Q}.
\]

where the left part of equation (12) is implemented in ITSM3D. Herein the first integral is utilized for some special paths where the so called Riemann invariants \( \tilde{\psi} \) are constant, see Table 1 and Figure 2. The Riemann invariants \( \tilde{\psi} \) can be described by

\[
\nabla \tilde{\psi} \cdot \tilde{r}_l = 0, \quad l = 1, \ldots, 5, \quad l \neq j,
\]

\[
\nabla \tilde{\psi} = \left( \begin{array}{c}
\tilde{\psi}_x \\
\tilde{\psi}_y \\
\tilde{\psi}_z \\
\end{array} \right),
\]

where \( \tilde{r}_l \) is the \( l \)th eigenvector of the flux Jacobian.
The extension of this originally first order method to higher order accuracy is done by van Leer's (1979) MUSCL extrapolation (3rd order) which is adapted to Benetschik's (1991) nonuniform grid spacing, see also Figure 2, where the way of extrapolation is sketched for the right side:

\[
Q_{i-1/2,j,k} = Q_{i-1,j,k} + \left( f_a \nabla Q + f_b^+ \Delta Q \right)_{i-1,j,k},
\]

\[
Q_{i+1/2,j,k} = Q_{i,j,k} - \left( f_a \nabla Q + f_b^+ \Delta Q \right)_{i,j,k}
\]

with the grid spacing ratio e.g. for the z-direction \( \sigma = \frac{h_z}{h_y} \) and \( \chi = \frac{1}{3} \) for 3rd order accuracy. Van Albada's (1982) limiter function \( s \) is defined as follows:

\[
s = \frac{\Delta Q}{(\Delta Q)^2 + (\nabla Q)^2 + \epsilon}, \quad \epsilon = 10^{-6}.
\]

Forward differences and backward differences as also used in the MUSCL extrapolation (17) are denoted by:

\[
\frac{\partial Q}{\partial x} = \frac{Q_{i+1,j,k} - Q_{i,j,k}}{x_{i+1/2} - x_{i-1/2}}.
\]

The time integration of equation (7) is realized by an explicit Runge-Kutta time-stepping scheme:

\[
\frac{Q_i^{n+1}}{Q_i^n} = 1 + \frac{1}{\Delta t} \sum_{i=1}^{5} \alpha_i \frac{\Delta Q}{\Delta V}, \quad q = 1 \ldots 5
\]

with Jameson's coefficients \( \alpha_1 = \frac{1}{6}, \alpha_2 = \frac{1}{3}, \alpha_3 = \frac{3}{4}, \alpha_4 = \frac{1}{2} \) and \( \alpha_5 = 1 \).

### COMMON METHODS

As mentioned the TVD scheme and the CD scheme are using several techniques in common. The following description presents only a short overview, more detail is available in Mez et al. (1992, 1995).

At first the boundary conditions are applied for both schemes. Six different treatments are implemented for turbine flow calculation: those for inlet and outlet, walls and also rotating wall surfaces, periodic boundaries and stator-rotor interaction. For the subsonic inlet of the stator four values have to be prescribed: the total pressure, the entropy, the circumferential and radial flow angles. For the subsonic rotor outlet only one value - the static pressure - has to be specified. The missing characteristics to complete each set of five characteristics which can't be calculated from the inlet or the outlet boundary conditions are computed by the boundary condition algorithm. The stator-rotor interface is also treated in this way. All the characteristics which have to be specified for instance for the stator domain come from the rotor domain. At walls a non-slip condition is applied. Walls are assumed to be adiabatic. The static pressure at the walls is computed from the vector of conserved quantities and is corrected according to the small neglected part of kinetic energy by applying the non-slip condition. The periodic boundaries are achieved by a simple restorage procedure. For the inlet, outlet and stator-rotor interface Giles' (1988) and Sazer's (1992) work of non-reflective boundary conditions has been extended to viscous flows. The non-equally distributed set of characteristic variables on the arbitrary grid spacing used for the Navier-Stokes calculation is replaced by an equidistant distributed set of characteristic variables using a cubic interpolation spline.
Secondly all convergence acceleration methods can be used for both schemes. There are mainly three techniques to accelerate the computation: local time-stepping, implicit residual smoothing by Jameson (1983) and full-multigrid based on the work of Jameson (1985).

In addition to that all other routines which do not directly concern the balancing, the flux and source term calculation are used in common. To name some, both schemes use the same metric calculation (cell faces and volumes), the same time step calculation and the same input and output modules.

RESULTS

The computational results of both the TVD scheme and the CD scheme are compared to each other as well as to experimental data obtained by Stetter et al. (1992). The test case is the last stage of a low-pressure turbine operated at the Institut für Thermische Strömungsmaschinen und Maschinenlaboratorium of the University of Stuttgart, see Figure 3.

Figure 3: Three dimensional view on the surface grid of two stator and two rotor blades

The strongly twisted rotor blades of the scaled 1/4.2 model turbine are rotating at a speed of 12,000 rpm. Flow measurements were performed with five hole cone probes and four hole wedge probes. The measuring planes are located at the inlet of the stator (plane 0), at an intermediate plane between stator and rotor (plane 1) and at the outlet of the rotor (plane 2), see Figure 4. The accuracy of the measurements (plane 0 and 2) can be specified for the circumferential flow angle with ±1°, the radial flow angle with ±3°, the static pressure with ±2%, the total pressure with ±1% and the Mach number with ±3%, respectively. For the transonic measurement in plane 1 uncertainties of about 20 - 30% exist, caused by probe blockage effects, see Truckenmüller et al. (1996).

The computational grids for the stator and rotor have 73 x 33 x 65 points each in axial, circumferential and radial direction, respectively. The total number of points is 313,170. The resulting wall spacing in terms of y+ can be specified with y+ = 20 to 40. A full-multigrid method with two coarser grid levels is used to accelerate convergence.

A detailed comparison of computed and experimental data can be seen in Figures 5 to 7. The computed data are obtained from the circumferential averaged primitive variables weighted by the circumferential angle.

Figure 5 shows the distribution of computed and measured Mach number, static and total pressure, circumferential and radial flow angle at plane 0. Total pressure and the flow angles distributions are taken as boundary conditions which explains the good agreement. The static pressure is overpredicted compared to the experiment, with the TVD scheme showing the smallest discrepancy. Accordingly, the Mach number is underpredicted, but in this case the CD results are in better agreement with the experimental data in the region from 20% - 100% span. The difference to the measurement could be explained by the use of the approximation of an ideal gas with constant ratio of specific heats instead of a real flow medium with vari-
able ratio of specific heats. The small constant offset of about 2 degrees in the radial flow angle (used as boundary condition) cannot be explained yet. For TVD a small discrepancy appears, as the smaller static pressure should give the higher Mach number. This could be caused by the non-isentropic circumferential averaging as well as by the different flux-balancing and extrapolation used for the boundary and phantom cells.

In plane 1 (Figure 6) the same variables are compared as in plane 0. The comparison for the total pressure shows distributions for TVD and CD which are underpredicted and nearly constant over the radius whereas the measured total pressure increases with radial position. Here differences between TVD and CD are negligible. On the contrary, the static pressure shows a big discrepancy to measurement. However, some uncertainties exist concerning the probe measurements for the static pressure in that region (probe blockage effect in a transonic flow). The TVD and CD curves cross at about 45% span and TVD has a steeper slope. The difference in the static pressure distributions of the two computations could be explained by the different phantom cell extrapolations. If, however, the experimental data for 10% and 90% span are accurately measured, then CD is closer to measurement. For the Mach number the same uncertainties mentioned above cause a discrepancy between the computations and the experiment. The relative Mach number is better predicted than the absolute Mach number. For both a small offset can be seen, larger for CD. The same behaviour is observed in the velocity components distributions, except for the radial direction, with a better distribution for TVD (not presented). The apparent discrepancy between the absolute and relative Mach numbers can be explained by a non-linear conversion from the absolute to the relative frame of reference, influenced by the flow angles, the radius and the rotation speed. The measured and computed characteristics of the flow angles are well predicted. For the circumferential angle there is an under-estimation of about 3 degrees for TVD as well as for CD. The radial flow angle is underpredicted in the region of 10% to 40% span. The maximum difference is about 5 degrees. TVD and CD distributions are almost the same. The good agreement of the angles is explainable because the narrow guide vane channel leads the flow with negligible deviation even though the probe blocks the channel. In this sense it is also reasonable that the blockage of the narrow channel increases the static pressure in the measurement.

At plane 2 (Figure 7) it can be seen that the static pressure is used as outlet boundary condition. The total pressure is predicted well by both methods, TVD and CD. In the tip and hub regions there is a slight deviation of these values. A similar tendency can be observed for the Mach number. The circumferential flow angle is not well predicted compared to the experimental data. Merz et al. (1995) show that a larger number of grid points in the axial direction and the inclusion of the tip gap yield a better result. TVD shows better results for the circumferential flow angles in the part below 50% span while in the upper half CD computed the best distribution. For the radial flow angle TVD yields the curve closest to the measurement. Hence the radial speed is better predicted by TVD (not presented). The differences between the flow angle distributions of both schemes can also be explained with the different boundary volumes and extrapolations.
In the Figures 8 to 10 the contour plots of the static pressure and the absolute Mach number are presented. They are shown for three azimuthal cuts: 5 %, 50 % and 95 % span, respectively. Globally the flow fields of TVD and CD seem to be nearly the same, but in detail local differences are found.

The comparison for the static pressure at 5 % span in Figure 8 shows a larger back pressure at the leading edge of the guide vane for TVD. In the passage between leading edge and trailing edge no significant differences can be seen on the pressure side of the guide vane. Some differences show on the suction side beyond the narrowest cross-section area. The pressure at the trailing edge is higher for CD. A similar behaviour can be observed for the rotor blades. Additionally it can be remarked, that the pressure on the pressure side is higher for CD. The suction side Mach number contours in the stator are different for TVD and CD. For CD an additional upstream deceleration can be observed near the suction side blade wall. The contours show some wiggles for CD and a higher speed level than for TVD. The suction side Mach numbers for the rotor are also higher for CD around the leading edge and after the trailing edge.

Near mid-span the TVD and CD pressure isolines are almost identical on the pressure sides of stator and rotor. Differences are seen on the suction side of the rotor blades. The local pressure minimum show different isoline shapes and has a lower level for CD. Additionally a partly higher downstream distribution is computed. Except in the leading and trailing edge pressure levels no differences are seen in the rotor. The Mach number contours show again distinct shape differences in the near wall regions as well as in the levels.

CD yields higher Mach numbers and doubly bent isolines near the walls.

In the tip region (95 % span) the pressure and Mach number levels are mostly similar but contour shapes are partly different. As seen in the hub region or for mid-span the CD Mach number contours are also doubly bent near the walls on the suction side of the stator.

The experimentally obtained mass flow is \( \dot{m} = 4.87 \frac{kg}{s} \). The comparison with the averaged mass flows for CD and TVD shows an underprediction, since both computations give a mass flow of \( \dot{m} = 4.80 \frac{kg}{s} \). An advantage of CD is the better accuracy in the mass flow conservation from stator-inlet to rotor-outlet with \( \Delta \dot{m} = 0.01 \frac{kg}{s} \). TVD shows a bigger mass flow difference of \( \Delta \dot{m} = 0.2 \frac{kg}{s} \), which divides into approximately equal parts for stator and rotor. The stator inlet mass flow is \( \dot{m} = 4.76 \frac{kg}{s} \). The bigger mass flow difference for TVD is probably caused by the 3rd order extrapolation. Another advantage of CD is the shorter computing time per step of about \( \Delta t = 214 \) s for a five stage Runge-Kutta time-stepping scheme with a CFL-number of \( CFL = 4.5 \). To compensate for this the Runge-Kutta time-stepping scheme has been reduced to three stages for TVD without loss of accuracy in time for the steady simulation. One time step then takes about \( \Delta t = 236 \) s, which can be reduced to \( \Delta t = 204 \) s by omitting the implicit residual smoothing, since in this case it does not allow to increase the CFL-number of \( CFL = 0.9 \). All computations have been performed on a 175 MHz DEC-Alpha workstation. However TVD shows a smoother convergence history without oscillations in the density residuals.

**CONCLUSIONS**

Two different numerical methods – a 3rd order TVD-upwind scheme and a central difference scheme with the concept of artificial viscosity – have been presented and compared. Both schemes are implemented in the code ITSM3D and use several modules in common, such as boundary conditions and acceleration techniques so that they can be compared well. In order to compare the capabilities of both schemes two computations have been performed for the last stage of a low-pressure turbine.

Globally a satisfactory agreement of computed and measured data has been achieved. The general features of the experimentally observed flow characteristics are well predicted. The differences of both computations have been shown in detail. The main differences are the discrepancies in the static pressure distribution at the planes 0 and 1 and the better resolution of radial flow angle and the radial velocity component at the plane 2 obtained with TVD. All other global comparisons gave insignificant changes in the flow field. The contours showed significant differences in levels, shapes and behaviour near the walls. Considering this it is hard to decide which is the most realistic computation so that further details of the real flow field would be necessary. For practical use CD was slightly better, with a shorter computing time per time step and a better conservation of the mass flow. On the whole it has been verified that both schemes - TVD as well as CD - lead to comparable and plausible solutions of the flow field.
REFERENCES


Figure 8: Comparison of static pressure and Mach number contours at 5% span. Order from top: p TVD, p CD, Ma TVD and Ma CD
Figure 9: Comparison of static pressure and Mach number contours at mid-span. Order from top: $p_{TVD}$, $p_{CD}$, $Ma_{TVD}$ and $Ma_{CD}$

Figure 10: Comparison of static pressure and Mach number contours at 95% span. Order from top: $p_{TVD}$, $p_{CD}$, $Ma_{TVD}$ and $Ma_{CD}$