AN EULER-BASED THROUGHFLOW APPROACH FOR AN AXIAL TURBINE AT SUPersonic FLOW REGIMES

S. Föllner, D. Brillert, F.-K. Benra

Chair of Turbomachinery, University Duisburg-Essen, Duisburg, Germany, sebastian.foellner@uni-due.de

ABSTRACT
An Euler-based throughflow approach with a novel setup of submodels for turbomachinery analysis at transonic flow regimes is presented. The method incorporates a loss free blade deviation and blockage model in combination with the state of the art loss model by Ainley and Mathieson with extensions that cover aerodynamic losses at transonic flow conditions. For validation, a comparative study between pitch-averaged Navier-Stokes and throughflow solutions is carried out over a wide operational range with transonic speeds for a 1.5 stage low aspect-ratio axial turbine with pronounced secondary flow structures. The prediction accuracy of two trailing edge flow angle models and the loss model are investigated at limit-load operating conditions. As a result, the throughflow model with prescribed exit flow angles adequately predicts the global flow pattern as well as the mass flow for subsonic and supersonic outlet flow conditions. Neither of the two trailing edge flow angle models is able to reproduce the influence of secondary flows on the flow angles in detail. Altogether, the presented throughflow method delivers reasonable results for outlet Mach numbers up to 1.2 and reduces computational process times by a factor of around 65 compared to three-dimensional CFD simulations.

KEYWORDS
MERIDIONAL, THROUGHFLOW, DISTRIBUTED LOSS MODEL

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>b</td>
<td>tangential blockage factor</td>
</tr>
<tr>
<td>e</td>
<td>internal energy</td>
</tr>
<tr>
<td>f</td>
<td>force</td>
</tr>
<tr>
<td>h</td>
<td>enthalpy</td>
</tr>
<tr>
<td>Ma</td>
<td>Mach number</td>
</tr>
<tr>
<td>p</td>
<td>pressure</td>
</tr>
<tr>
<td>s</td>
<td>entropy</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>t</td>
<td>time-step size</td>
</tr>
<tr>
<td>T</td>
<td>temperature</td>
</tr>
<tr>
<td>(w_x, w_y, w_z)</td>
<td>components of the relative velocity vector</td>
</tr>
<tr>
<td>U</td>
<td>conservative variables vector</td>
</tr>
<tr>
<td>F, G, H</td>
<td>cartesian flux vectors</td>
</tr>
<tr>
<td>(Y, \alpha)</td>
<td>loss coefficient, flow angle</td>
</tr>
<tr>
<td>(\rho, \theta, \tau, \xi)</td>
<td>density, tangential flow angle, pseudo time-step, correction factor</td>
</tr>
</tbody>
</table>

Indices
- \(c\) compressibility
- \(inc\) incidence
- \(in, out\) inlet/outlet conditions
- \(p\) profile
- \(sec\) secondary flow
- \(t, s\) total/static state
- \(tcl\) tip clearance
- \(te\) trailing edge
- \(T\) transposed vector
INTRODUCTION

Flexible, fast and reliable design tools are essential in the turbomachinery design process. Especially off-design performance characteristics of stationary gas turbines have become more important due to the increase of renewable powers and the demand for more flexible load operations in the last two decades. Hence, the preliminary design process of new turbine layouts without the existence of detailed airfoil geometries is being pushed to its limits in order to optimize for the highest possible turbine performance at relevant operating points. Consequently, meridional throughflow programs are still a core component of the design process. In the preliminary design phase, these programs take only key geometrical parameters into consideration and provide information about the two-dimensional axisymmetric flow conditions of turbomachinery flows. Moreover, contributions made by throughflow programs have a substantial impact on the whole development process therefore constant research has been carried out to improve these methods.

Presently, throughflow programs based on the streamline curvature method are the backbone of established methods in the preliminary design phase in industry, see for example Casey and Robinson (2010). Due to their underlying streamline theory, this method exhibits disadvantages covering possible flow conditions at off-design points sufficiently. Such flow conditions include transonic flows with shocks at limit-load operating points, local backflow areas at part load operating points or the radial mixing of cooling flows between adjacent streamlines. Therefore, recent research presented by Sturmayr and Hirsch (1999) and Pacciani et al. (2017) has focused on throughflow methods based on the Euler-equations with a finite-volume approach. The finite-volume methodology offers more flexibility with regard to radial mixing and is also capable of resolving locally choked blade channel areas using the Euler-equations with a time-marching approach. Since throughflow methods solve the axisymmetric flow conditions only on the meridional S2 flow surface, the computation times are still an order of magnitude faster compared to fully three-dimensional simulations. In this context, the S2 flow surface defined by Wu (1952) divides the mass flow in the blade channel into two equal parts and is therefore called the mean flow surface. Furthermore, throughflow methods allow the calibration of the results with existing fleet data which is not easily possible with 3D CFD simulations.

Encouraged by the advances in CFD-based throughflow methods and their advantages in terms of numerical stability and accuracy, an Euler-based throughflow program has been developed specifically for the analysis of modern stationary gas turbines. The computational method was designed to reliably solve flow conditions at part load and peak load conditions and is also expected to consider blade cooling flow injections with a higher level of detail than streamline curvature methods in the future. Within this paper, the novel throughflow program is applied to analyze the flow of an uncooled 1.5-stage axial turbine and to validate the different submodels for flow deviations and aerodynamic losses at critical operating points with supersonic exit conditions.

THROUGHFLOW MODEL

The computational domain of throughflow methods incorporates the meridional flow surface between two adjacent blades and therefore does not include the airfoil shape itself. Additional models are necessary to represent the effect of airfoil geometry on the axisymmetric flow. Basically, this includes a blade deviation model introducing a flow deflection and a blockage model accounting for the flow acceleration due to the reduced free channel area by the number and thickness of installed turbine blades. In tFlow, these two submodels are implemented as additional source terms within the governing Euler-equations. Both submodels act like volumetric
forces and are set up in a way that does not generate artificial entropy at the S2 plane. Aerodynamic losses are provided externally by an empirical loss model and by adding volumetric friction forces. The full set of governing equations in cartesian coordinates reads:

\[
\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}}{\partial x} + \frac{\partial \mathbf{G}}{\partial y} + \frac{\partial \mathbf{H}}{\partial z} = \rho (f_{\text{blockage}} + f_{\text{blade}} + f_{\text{friction}} + f_{\text{coriolis}} + f_{\text{centrifugal}}) \tag{1}
\]

with

\[
\mathbf{U} = \begin{pmatrix} \rho \\ \rho w_x \\ \rho w_y \\ \rho w_z \\ \rho e_t \end{pmatrix}, \quad \mathbf{F} = \begin{pmatrix} \rho w_x \\ \rho w_x^2 + p \\ \rho w_x w_y \\ \rho w_x w_z \\ \rho w_x h_t \end{pmatrix}, \quad \mathbf{G} = \begin{pmatrix} \rho w_y \\ \rho w_x w_y + p \\ \rho w_y w_z \\ \rho w_y h_t \\ \rho w_y h_t \end{pmatrix} \quad \text{and} \quad \mathbf{H} = \begin{pmatrix} \rho w_z \\ \rho w_x w_z \\ \rho w_y w_z + p \\ \rho w_z h_t \end{pmatrix}.
\]

The first three volumetric source terms \(\rho f_{\text{blockage}}, \rho f_{\text{blade}}\) and \(\rho f_{\text{friction}}\) on the right-hand side of Eq. (1) apply only to rotor and stator mesh blocks. Furthermore, the source terms for coriolis and centrifugal forces are only used in the relative frame of reference of moving rotor blocks. In mesh blocks representing free duct sections all volumetric source terms except the term for the blockage \(\rho f_{\text{blockage}}\) are disabled.

Normal shocks and locally choked areas inside the blade channel are strongly influenced by the blockage source term. The term is defined as

\[
\rho f_{\text{blockage}} = - \left( w_x \frac{\partial b}{\partial x} + w_y \frac{\partial b}{\partial y} + w_z \frac{\partial b}{\partial z} \right) [\rho, \rho w_x, \rho w_y, \rho w_z, \rho h_t]^T \tag{2}
\]

with \(b\) as the tangential blockage factor. Inside blade channels, the blockage factor is calculated as ratio of the available flow area from suction to pressure side of the airfoil and the pitch in circumferential direction. Thus, the blockage factor takes values less than one inside blade channels and is unity in free duct sections.

The magnitude of volumetric blade force is modeled according to the time-marching approach of Sturmayr (2004):

\[
\frac{\partial (\rho \| \mathbf{f}_{\text{blade}} \|)}{\partial t} = \left( \rho f_{\text{blade}}^{(k)} \right)^{\text{target}}(k) - \left( \rho f_{\text{blade}}^{(k)} \right)^{(k)} \cdot \Delta t \tag{3}
\]

with

\[
\rho f_{\text{blade}} = \rho \| \mathbf{f}_{\text{blade}} \| [0, -\sin \theta, \cos \theta, 0, 0]^T \tag{4}
\]

where \(w_{\theta}^{(k)}\) denotes the tangential component of the relative velocity vector at current time-step \(k\). Values with a tilde on top denote blade average values.

In contrast to other deviation models presented by Baralon et al. (1998) or Persico and Rebay (2012) the model of Sturmayr needs no case specific correction factors and offers numerical stability for different types of time-integration schemes. For throughflow applications, two different types of simulations can be distinguished. First, the analysis mode is used for the simulation of existing machines with defined geometry and known global flow patterns from measurement data or three-dimensional CFD results. Second, the design mode which is used for the calculation of new turbine designs with parameterized geometry. This mode requires a model-based estimation of trailing edge flow angles, blockage factor distribution and losses.
depending on the geometry and flow conditions. The deviation model given in Eq. (3) supports both application modes. The target tangential velocity component $w_{\text{target}}^\theta$ can be derived from prescribed analytical distribution functions of the flow angle $\theta$ in design mode or from the blade camberline surface in case of the analysis mode. A more detailed description and validation of the single source terms is presented by the authors in Föllner et al. (2020).

Flow losses are included via a distributed loss model described by Hirsch (1991) that is based on Crocco’s theorem:

$$
\rho \mathbf{f}_{\text{friction}} = -T \nabla s \frac{\mathbf{w}}{||w||}.
$$

In Eq. (5) the entropy gradient is a function of the loss coefficient

$$
Y = \frac{p_{t,\text{in}} - p_{t,\text{out}}}{p_{t,\text{out}} - p_{\text{out}}}.
$$

that is calculated according the empirical loss model of Ainley and Mathieson (1951), Dunham and Came (1970) and Kacker and Okapuu (1982). Because a full description of the empirical loss model would be beyond the scope of this paper only the overall loss equation is given here for comprehension:

$$
Y = 0.914 \xi_{Re} \xi_{Ma} \xi_c Y_p + Y_{p,\text{inc}} + Y_{te} + Y_{tcl} + \xi_{inc} Y_{sec}
$$

with

$$
\xi_{Ma} = \begin{cases} 
1 + 60(Ma - 1)^2, & Ma_{\text{out}} \geq 1 \\
1, & Ma_{\text{out}} < 1
\end{cases}
$$

Numerical Scheme and Computational Domain

In Fig. 1 the exemplary computational domain of a 1.5 stage axial turbine for throughflow calculation is illustrated. Periodic boundary conditions are applied to the front and back faces of the domain. An explicit four stage Runge-Kutta scheme is used for time integration in combination with the flux difference scheme of Roe (1981). Due to the fact that steady-state solutions are of most interest, local time-stepping and residual smoothing is employed to speed up convergence of the time explicit solver.

![Computational domain of the throughflow domain of the 1.5 stage axial turbine](image)
One And A Half Stage Axial Turbine

In general, the appropriate prediction of single loss components for different turbine geometries over a wide range of operating points is a challenging task. To validate the developed throughflow method together with the empirical loss model described in Eq. (7) the public available test-case of a one and a half stage axial turbine is chosen. The turbine consists of one complete stage with an exit guide vane (stator 2) as illustrated in Fig. 2. Originally, the turbine was designed to investigate the flow structure of well developed secondary flows and their interaction between rotors and stators. To create a secondary flow structure the turbine consists of low aspect ratio blades with prismatic airfoil shapes that are consciously not optimized for three-dimensional flow structures. The geometrical key parameters of the turbine are listed in Tab. 1. A more detailed explanation of the turbine geometry and the test-rig setup is given by Walraevens and Gallus (1994), Walraevens and Gallus (1996) and Walraevens et al. (1998). For the reason mentioned above, the test-case covers all sources of aerodynamic losses and can be regarded as a suitable and for some aspects even a critical test-case for the validation of empirical loss models.

Figure 2: CFD domain of the 1.5 stage axial turbine with leading (LE) and trailing edge (TE) evaluation planes for the losses of stator 2

Numerical Setup and Boundary Conditions

The applied boundary conditions for the CFD and throughflow simulations are related to the so called bigsteady operating point by the original authors. Taking these initial conditions, the first operating point is defined by a total inlet pressure of 1.7 bar and a static outlet pressure of 1 bar. All seven investigated operating points listed in Tab. 2 are calculated with a total inlet temperature of 306 K and swirl free inlet conditions. Starting from the first operating point, the outlet static pressure for the further operating points (OP2 - OP7) is reduced in steps of 0.1 bar to increase the aerodynamic turbine load and average outlet Mach number. At the outlet face a pressure distribution based on the radial equilibrium condition is enforced within the CFD setup due to the strong exit swirl behind stator 2. For the validation of the throughflow model, the pitch-averaged radial distributions of total pressure and total temperature at the inlet face and static pressure at the outlet face are taken from the CFD results and set in the throughflow model setup. Thus, the boundary conditions for all operating points are identical in the CFD and throughflow simulation.

Because the turbine is operated with relatively low pressure ratios and ambient air the fluid is
modelled as ideal gas with a specific gas constant of \( R = 287 \, \text{J/(kg \cdot K)} \) and a ratio of specific heat capacities of \( \kappa = 1.4 \).

For the CFD simulations with ANSYS CFX the k-\( \omega \) SST turbulence and the standard \( \gamma - \theta \) transition model are used. The three-dimensional computational domain is discretized with approximately 2.9m cells. Thereby, the computational domains of the individual blade rows are connected via mixing plane interfaces. The average computational process time for each operating point is about 2h 15min on 8 CPU cores. Contrary to the CFD simulations, the two-dimensional throughflow domain is discretized with only 32 cells in radial direction and 120 cells in axial direction which sums up to 3840 cells in total. Qualitatively the mesh structure is comparable to the mesh shown in Fig. 1 with a higher mesh density for rotor and stator blocks. The average process time for a single operating point calculated by tFlow is about 2 min on a single CPU core.

**Analysis of Throughflow Results with Pitch-Averaged CFD Data**

In the first place, the trailing edge deviation models predicting the exit flow angles of each row are switched off and the exit flow angles are taken directly from the pitch-averaged CFD results. As the empirical loss model given by Eq. (7) is designed for larger turbines with optimized airfoil shapes the incidence and secondary flow losses overvalue the flow conditions for the present turbine configuration. Therefore, the incidence loss component is scaled to a value of 50\% and the secondary flow loss component is scaled to a value of 25\% of the original values to be within a reasonable range of the test-case. These two scaling factors are kept constant over the entire operational range of the turbine. The scaling itself is justifiable in terms of validation, since the trend and the difference of loss values compared to the reference point are decisive for the accuracy of the empirical model.

In Fig. 3 the results of the pitch-averaged CFD and the throughflow calculation are plotted at 50\% span along the axial (x-) direction. The throughflow results of the Mach number in absolute frame of reference and the static pressure are in good agreement with the CFD results for OP1. Even at supersonic outflow conditions the values agree reasonably well as shown in the lower graphic of Fig. 3. This implies the correct influence of the source terms for blade deviation and aerodynamic losses, since these two flow quantities are sufficient to compare the flow along a streamline.

The calculated mass flow at OP1 of the throughflow solution is 8.855 kg/s compared to
8.887 kg/s of the CFD solution which leads to a relative error of around 0.36% between CFD and throughflow solution. Such an error is sufficiently small for the proper prediction of the machine operating point and performance inside a performance map. Even for OP7 with $Ma_{out} = 1.22$ an acceptable relative numerical error with respect to the mass flow of only 2.5% is calculated.

Quantities inside the blade channels highlighted in grey in Fig. 3 arise from the prescribed target angles in Eq. (3) and blockage factors in Eq. (2). As the axisymmetric tangential flow angle depends on blade shape and number, these distributions can be improved with new target-angle distributions taken from CFD parameter studies. Since the applied throughflow model should also be used for the design mode with limited information about the blade geometry, a simple target angle distribution based on a power-law is applied at this point.

In Fig. 4 the outlet flow angles over the normalized blade heights of the three airfoils are shown. The pronounced vortices of secondary flow structure affect the flow angles of the CFD solution. This can be observed for all airfoils of the turbine. For stator 1, the influence of two horseshoe vortices near the endwalls at 10% and 90% span lead to peaks within the radial distribution of outlet flow angles. Within rotor 1 and stator 2 effects of secondary flow vortices become stronger and different vortices seem to overlap which produces a wavy distribution for the outlet flow angle. The red and blue lines plotted in Fig. 4 represent the outlet flow angle predictions for the sine-rule (blue line) by Traupel (2001) and the deviation model by Aungier (2006) (red line). Since the sine-rule solely depends on the geometrical parameters of throat and pitch area the calculated outlet angles are constant for all operating points. In contrast, the Aungier model depends additionally on flow conditions and trailing edge losses which vary between OP1 and OP7. Neither the Aungier model nor the sine-rule is able to predict the wavy structure of the CFD outlet flow angle distribution and will therefore result in a slightly different operating point.
and calculated mass flow if applied in a throughflow simulation. These model properties form a significant disadvantage in case of the design mode where no detailed information about outlet flow angle distributions is available.

Moreover, the CFD results of OP7 with an outlet Mach number of Ma=1.22 is plotted with dashed lines in Fig. 4. It can be observed that the expansion and change of flow quantities is mainly conducted within stator 2 as the flow angle distributions of the rotor and stator 1 remain nearly the same for both operating points. At supersonic conditions of OP7 the flow is turning towards the axial direction which results in less flow deviation at a higher mass flow rate of stator 2.

![Figure 4: Outlet flow angles of the CFD result at OP1 and OP7 with predictions of two deviation models](image)

In Fig. 5 the loss coefficient $Y$ is plotted over the average outlet Mach number. The loss coefficients of the CFD solution are evaluated from the mass-averaged values of static and total pressures at the TE and LE plane, as highlighted in Fig. 2. Qualitatively, the losses from CFD shown in Fig. 5 increase strongly starting at an outlet Mach number of unity which corresponds to OP5. For the subsonic regime (OP1 - OP4), the loss coefficient calculated from the CFD result remains nearly constant at a level of 4% compared to the empirical model that predicts slightly increasing values between 5% and 6%. The quantitative difference between CFD and throughflow result are mainly due to the calibration of the empirical model with fixed constants for all airfoils of the turbine. Whereas a calibration of the loss model to a single blade row is not practical with respect to the calculation of the total turbine performance with a throughflow method. In general, the predicted aerodynamic losses calculated based on the empirical model show the same trend like the CFD results and represent a good prediction for the design and analysis mode. The increasing losses starting at a supersonic flow regime have been expected for the airfoil shape of the stator 2 which is designed for fully subsonic flows.

A reason for the increasing losses at supersonic flow conditions can be drawn from Fig. 7 which illustrates the Mach number contour lines of the stator 2. The sonic line is highlighted in red. At OP4 with an average turbine outlet Mach number of 0.85 the flow becomes transonic at
the suction side of the airfoil. With further acceleration the sonic lines moves upstream and influences the profile boundary layer with negative effects for the aerodynamic losses. Moreover, a reduced flow deflection appears at the trailing edge at OP5 (Fig. 7b) and strong shocks are a source for entropy generation. Apart from the boundary layer the upstream sonic line at OP6 (Fig. 7c) is almost closed between the trailing edge of adjacent airfoils and the suction side. The widened boundary layer affects the profile losses in the rear part of the suction side which is reflected by the increased loss for the last three operating points in Fig. 5.

The last two investigated operating points OP6 ($Ma = 1.09$) and OP7 ($Ma = 1.22$) show a difference in calculated losses $Y$ between CFD and throughflow results of around 2% and 3% respectively, as illustrated in Fig. 5. For OP7, the radial distributions of the loss coefficient and Mach number of the CFD and throughflow solutions are plotted over the normalized blade height in Fig. 6. The Mach number distribution of the throughflow result matches the CFD results almost exactly except from the endwall boundary layer region which is not modelled in the
throughflow simulation. Between 50% and 100% span the predicted losses of the throughflow result correlate with the CFD result for Mach numbers up to values of 1.22. At higher Mach numbers in the hub region between 1.2 and 1.3 the empirical loss model predicts significantly higher losses than evaluated from the pitch-averaged CFD results. Hence, the supersonic loss correction factor of Eq. (7) is able to predict the overall trend for low sonic conditions but fails for local flow conditions with Mach numbers higher than 1.2.

Conclusions

In this paper, a novel CFD-based throughflow program has been applied for the analysis of a one and a half stage axial turbine at operating points with transonic flow conditions. Different deviation models and the prediction accuracy of the state of the art loss model of Ainley-Mathieson with extension by Kacker and Okapuu have been investigated at supersonic speeds with strong shocks and secondary losses. Summarizing the presented results it can be concluded that:

- the developed throughflow model calculates very accurately the transonic flow conditions over a wide operational range at the analysis mode with prescribed trailing edge flow angles.
- the program is able to reliably calculate maximum load conditions of modern gas turbines with transonic outlet flows, which is not easily possible with streamline curvature methods.
- the numerical error of the predicted mass-flow compared to CFD results was less than 1% at fully subsonic operating conditions and less than 2.5% at transonic operating conditions.
- none of the investigated deviation models is able to predict the wavy radial distribution of exit flow angles for low aspect turbine blades with pronounced secondary flow patterns. Hence, for the design mode with activated deviation model the predicted mass-flows and aerodynamic losses deviate considerably from the CFD results.
- the overall trend of aerodynamic losses at low supersonic exit conditions is comparable to CFD results for the investigated turbine.
- the correction factor for supersonic shock losses of the applied loss model overestimates the aerodynamic losses at Mach numbers greater than 1.15 and needs to be adapted.

An adaption of the shock loss correction factor requires further investigations with different airfoil shapes to determine geometrical key parameters that influence the evolving shock pattern. To yield an improved and generally valid correction factor airfoil shapes specifically designed for transonic flow regimes have to be considered as well.

ACKNOWLEDGEMENTS

The presented work is funded jointly by the Bundesministerium für Wirtschaft und Technologie (BMWi) of the German government and Siemens Energy AG as part of the AG-Turbo program; contract number 03ET7091B. The corresponding author is very thankful for the financial support without whom the obtained results would not have been possible.

REFERENCES


