GT2011-46%\$

OPTIMIZATION STRATEGY FOR A COUPLED DESIGN OF THE LAST STAGE AND THE SUCCESSIVE DIFFUSER IN A LOW PRESSURE STEAM TURBINE

Christian Musch Siemens AG Energy Sector 45478 Mülheim an der Ruhr Germany christian.musch@siemens.com

Heinrich Stüer Siemens AG Energy Sector 45478 Mülheim an der Ruhr Germany heinrich.stueer@siemens.com

Georg Hermle

University of Kassel Department of Turbomachinery 34109 Kassel Germany georg.hermle@uni-kassel.de

ABSTRACT

In this study, an effective yet numerically simple approach for a coupled design of the last stage running blade and diffuser is presented. The method applied uses a 2-dimensional streamline curvature code combined with a boundary layer solver for the prediction of flow separation within the diffuser. An accurate representation of the diffuser flow is vital for the assessment of the overall performance. Thus the major influences from the turbine stage on the diffuser flow, i.e. the tip leakage jet and the swirl of the flow, are taken into account. Secondary effects like blade wakes are neglected.

The basic capability of the method to correctly represent the flow is demonstrated by a comparison with 3-dimensional CFD simulations of a sample configuration. Solid correlation can be found between both cases. For the optimization process, a genetic algorithm is used. Optimization parameters include the blade exit angle and the diffuser contour. The results of the optimization are again scrutinized with the assistance of 3-dimensional CFD simulations.

NOMENCLATURE

- A cross sectional area
- *a* specific technical work
- *c* absolute velocity
- c_p pressure recovery coefficient
- c_f friction coefficient
- *h* static enthalpy

- h_t total enthalpy
- *p* static pressure
- p_t total pressure
- j dissipation
- *u*⁺ dimensionless velocity
- v^+ dimensionless wall distance
- Ma Mach number
- Re Reynolds number
- α absolute swirl angle
- η stage efficiency
- ρ density
- τ_w wall shear stress
- γ isentropic exponent

Subscripts

- ax axial
- m meridional
- ref reference value
- ∞ freestream value
- 0 stator vane inlet
- 1 stator vane outlet / rotor blade inlet
- 2 rotor blade outlet / diffuser inlet
- 3 diffuser outlet
- 3s isentropic change of state from p_0 to p_3

INTRODUCTION

One goal in the design process of modern steam turbines for power plant applications is aiming for high efficiency. A large gain in efficiency is expected from the optimization of the last stage and diffuser of the low-pressure (LP) turbine, especially as LP turbines are usually multiflow arrangements. The goal of such an optimization will always include the minimisation of leaving loss, as the kinetic energy contained in the exit velocity of the diffusor cannot be converted into shaft work and thus lowers the efficieny of the turbine. In order to fully evaluate the efficiency of a last stage and diffusor design the leaving loss must be included. To account for all the effects of the large radial extent of the last stage, a 3-dimensional (3D) investigation of the flow is considered state of the art within the design process. Nevertheless, a numerical optimization using 3D computational fluid dynamics (CFD) is, even with the increase of computational resources over the last years, still very time-consuming and often not feasible within the design process. Separating the design of the stage and the diffuser is, therefore, not unusual and optimization potential remains unexploited.

In order to improve the diffuser design, numerous experimental studies have been carried out in the past. These studies mostly focused on individual effects on the diffuser flow, like swirl or wall jets at the casing. For example it has been shown by McDonald et al. [1], that moderate swirl has a positive effect on pressure recovery. Likewise Nicoll et al. [2] showed the positive effect of wall jets on conical diffuser performance. Those studies were done with isolated diffuser configurations. As has been shown by Uvarov [3] the transfer of those results on diffusers behind a turbine stage is somehow limited due to the inhomogeneous flow field at the turbine outlet. Comprehensive studies on the aerodynamic interaction of the last stage and a conical diffuser have been carried out by Kruse et al. [4]. The positive influence of the wall jet and the swirl can be confirmed. In addition it could be shown that for a medium sized tip gap the efficiency of stage and diffuser was maximal. Zimmermann et al. [5] varied the tip clearance gap in a LP model turbine with an axial radial diffuser and also found an increase in pressure recovery with an increased clearance, but the overall efficiency was decreased.

Numerical investigations on the impact of rotor tip clearance on the diffuser have been carried out by Willinger et al. [6]. The results confirmed the findings by Zimmermann et al. Kreitmeier et al. [7] showed very vivid the importance of taking the entire system of turbine stage and diffuser into account.

From all those studies it can be deduced that the main effects (first order effects) on the diffuser flow field are the tip jet and swirl of the last stage. For the assessment of the performance of the last stage, an accurate representation of those effects is vital. These considerations imply that for an axial radial diffuser (in contrast to a purely axial diffuser) special attention must be paid to one particular region which is most likely to have boundary layer separation. This region is at the diffuser casing wall, where the effects of flow deceleration due to flow area expansion and streamline curvature are superimposed. In most cases, and this is especially true for load points with only little or no swirl (like the design point), it is thus sufficient to prevent flow separation at the diffuser casing wall to get a healthy diffuser flow. The design philosophy presented focuses on the optimization of the stage exit flow and diffuser shape in order to avoid flow separation on the outer diffuser wall and minimization of diffuser leaving loss. It will be shown that this approach leads to an overall efficiency gain for the turbine.

NUMERICAL MODEL

The 2D code which is used for the optimization in this study is the well-known through-flow code SLEQ by Denton [8]. A test case, which will be discussed in section OPTIMIZATION, has been chosen for the optimization. The tip jet is modelled by a simple boundary layer (BL) solver, as descripted by Schetz [9], which is adapted for the presented investigations to calculate the propagation of the wall jet along the diffuser wall. The freestream velocity at the outer diffuser wall calculated by SLEQ is used as an input for the BL solver. The method applied is a finite-difference method on a stretched grid. Turbulence closure is modelled using a three layer approach, using the formulation by Reichardt [10] for the inner region of the boundary layer. For the outer region of the boundary layer the model by Clauser [11] is deployed. The turbulence in the shear layer of the jet is modelled by Prandtl's third mixing-length eddy viscosity model for the turbulent plane jet. Effects due to compressibility within the boundary layer are assumed to be negligible. For all calculations of the jet flow a mesh of 500 points normal to the wall is chosen. The accuracy of the computed results will be demonstrated in the next section.

These two codes are coupled to a genetic optimization algorithm. The algorithm is called Covariance Matrix Adaptation Evolution Strategy (CMA-ES) (see Hansen [12]). One main advantage of the CMA-ES algorithm is, that it is quasi parameter free and hence very easy to handle. Furthermore, it has been empirically successful in many applications and is particularly powerful for rugged or noisy objective functions, which makes it especially suitable for the purposes described in this paper.

VALIDATION OF BOUNDARY LAYER SOLVER

To validate the BL solver with regard to its capability to calculate a wall jet flow, two test cases have been chosen. Firstly a 2D plane wall jet, for which comprehensive measurements are available, is considered. A second case represents a wall jet with an adverse pressure gradient.

2D Plane Wall Jet

The first test case considered is taken from the ERCOFTAC classic database (for details see Karlsson [13]). In this study Laser-Doppler measurements in a 2D plane wall jet were con-

ducted. The Reynolds number based on the inlet velocity was approximately Re=9600. Measurements were conducted at different positions downstream of the initial slot. Representative comparisons for distances of roughly 200mm and 2000mm are shown in figures 1a and 1b respectively. As a second reference,



(a) Jet Profile 200mm Downstream of Initial Slot



(b) Jet Profile 2000mm Downstream of Initial Slot

FIGURE 1: VELOCITY PROFILES OF WALL JET



FIGURE 2: BOUNDARY LAYER PROFILE OF WALL JET

two Navier-Stokes flow simulations with CFX, using the standard $k - \varepsilon$ model and the SST turbulence model, are also presented. Good correlation between the computed and measured velocity distributions can be found. The good prediction of the flow in the near wall region by the BL solver is demonstrated from a plot of the boundary layer. In figure 2 the Velocity u^+ against the wall distance y^+ for distances of 400mm, 700mm, 1000mm and 1500mm downstream of the slot is shown. The self-similar behaviour of the boundary layer which can clearly be seen from the measurements is also predicted by the boundary layer solver, showing again a solid correlation to the measurements.

2D Wall Jet With Adverse Pressure Gradient

The second test case for the validation of the boundary layer solver is an annular axial radial diffuser including a wall jet. This test case serves to validate the accuracy to calculate a wall jet with adverse pressure gradient. The diffuser geometry consists of two arcs. The exact values are given in Table 1.

TABLE 1: GEOMETRICAL DATA OF AXIAL RADIAL DIF-FUSER TEST CASE



FIGURE 3: TEST DIFFUSER ARRANGEMENT

The diffuser area ratio is chosen to ensure a separated flow even in the presence of a moderate wall jet. This also allows validation of the accuracy to predict the point of separation by the boundary layer code. CFX was used to calculate the reference flow field. Again the $k - \varepsilon$ and the SST turbulence model are considered. Both models are widely used in industrial applications. As the SST model predicts an early onset of flow separation and the $k - \varepsilon$ model a late onset, these models are suitable to demonstrate the range of accuracy for a typical 3D CFD simulation. The inlet velocity of the main flow is set to be 200 m/s which corresponds to a Mach number of roughly Ma=0.5 for wet steam conditions. This corresponds to a typical load case of a last stage in a steam turbine. The jet velocity is 350 m/s, and the initial height of the wall jet is set to 15mm. This does not correspond to the actual tip clearance of a last stage running blade. Still the value yields a good representation of an actual wall jet in a last stage. Reasons are discussed in the next section. In order to calculate the wall jet with the boundary layer solver, the free-stream velocity outside of the boundary layer has to be provided. The free-stream velocity along the diffuser casing wall is evaluated from an inviscid simulation without wall jet. Figure 4a shows a comparison of



FIGURE 4: WALL JET WITH PRESSURE GRADIENT

the wall jet velocity between the 3D CFD simulation and the result of the boundary layer solver. The acceleration due to the streamline curvature at the diffuser inlet is in very good agreement with the 3D CFD. The gradient of the deceleration due to the flow area expansion and reduction of streamline curvature is predicted slightly lower by the BL solver than in the 3D CFD. As shown in figure 4b this also results in a slightly different gradient of the wall shear along the diffuser wall. However, the general characteristic of the wall jet and the predicted point of separation lies within the range of the 3D CFD. This shows that the BL solver provides satisfactory results with regard to flow separation in an axial radial diffuser.

OPTIMIZATION

As the reference case an old existing design is used, where the blade and diffuser were designed separately. Firstly, from a steady state simulation, a references flowfield is created using the 3D Navier-Stokes solver CFX from ANSYS. The effects of turbulence are captured using the Shear-Stress-Transport (SST) model with wall functions. The SST model is a hybrid model, blending between the $k - \omega$ model in the near wall region to the $k - \varepsilon$ model in the core flow. The solver is based on a finite volume approach. A hybrid scheme for the spatial discretization is employed resulting in second-order accuracy. Figure 5 shows the topology of a typical investigated geometry. The flow domain is divided into four regions (stator vane, rotor blade, diffuser and extension at outlet) which are represented by different colours. The extension behind the axial radial diffuser is em-



FIGURE 5: FLOW DOMAIN OF LAST STAGE AND DIF-FUSER

ployed for reasons of numerical stability and to avoid negative influences from the outlet boundary onto the diffuser flow. The overall amount of grid nodes is roughly about 1 Million. Between stator vane and rotor blade a mixing plane is introduced. The diffuser domain is modelled in the same frame of reference as the rotor to avoid nonphysical behaviour of the blade wakes in the simulation (see Musch et al. [14] for details). The inlet boundary condition is taken from a through-flow calculation. The total pressure, total temperature and the flow direction are prescribed as radial profiles. At the outlet the static pressure is prescribed, using an opening type boundary condition, which allows for fluid to exit and enter the domain. The flow direction at the outlet is in radial direction. The wet steam was modelled as an ideal gas. Sound correlation with measurements from a model turbine can be found for similar numerical setups using CFX in the literature, e.g. Polklas [15]. Additionally the code has been successfully used by Becker [16] to calculate the flow through an axial radial diffuser test rig.

Initial Setup

Through-flow calculations are done including the last three standard size stages in order to get reliable flow conditions for the last stage. To get viable results from the optimization it is furthermore crucial to match the results of the through-flow calculations as good as possible to the 3D Navier-Stokes simulations. In order to achieve this the exit flow angles of each blade row in the through-flow calculations were adjusted to the 3D CFD. Comparisons for the test case of the meridional velocity, the swirl angle and the total pressure profile are shown in figure 6. As next step the BL solver has to be matched to the 3D simulations as well. This must be done for mainly one reason. As was already mentioned before, the tip jet forming above the running blade is not axissymmetric. Due to the three dimensional nature of the flow in the interaction zone of the supersonic and subsonic regions of the tip jet (vortex breakup etc.), it is not straight forward to derive a correlation for the initial jet height based on the tip clearance. Nevertheless, if the initial jet height and velocity are approximated from the a circumferential averaging of the 3D CFD, it can be shown, that it is possible to get reasonable results from the BL solver when compared to the data of the 3D simulations. The initial jet height is thus chosen to 13mm and the initial jet velocity to 350 m/s. The following two figures show the results of this adjustment. In figure 7a the wall shear as calculated by the



FIGURE 6: COMPARISON OF SLEQ WITH 3D-CFD



FIGURE 7: COMPARISON OF BL-SOLVER WITH 3D-CFD

BL solver is compared to the circumferential averaged 3D CFD results. Figure 7b shows the jet velocity calculated by the BL solver along the diffuser casing wall, compared to values from circumferential averaged 3D results at some distinct locations in

the diffuser. It can clearly be seen that the BL solver gives a rather good prediction of the flow behaviour as derived from the 3D Navier-Stokes simulations. Once the values for the initial jet height and velocity are adjusted, they are kept constant throughout the whole optimization process. The parameters which are allowed to change during the optimization procedure are the exit flow angles of the last running blade and the diffuser shape at the casing. A simplified diffuser shape is used, which consists of straight lines and one arc. In addition to the length of the lines the angle is also allowed to vary. The same applies to the arc. Not only the radius, but also the start and end angle are varied during the optimization. All diffuser parameters are shown in figure 8.



FIGURE 8: 2D-FLOWPATH AND DIFFUSER PARAMETERS

The efficiency of the last stage is chosen as the objective function. No attention is paid to the efficiency of the first two stages, as last stages of the size studied in this paper are typically fully or nearly chocked in the stationary last stage blade. Thus changing the last rotor blade does not affect the upstream stages. The kinetic energy at the diffuser outlet is included as an additional loss. This yields the following definition of the efficiency.

$$\eta = \frac{a}{a-j} = \frac{\Delta h_t}{\Delta h_t + \underbrace{(h_{3s} - h_3)}_{j} - \underbrace{\frac{1}{2}c_3^2}_{\text{Leaving Loss}}} = \frac{h_{3t} - h_{0t}}{h_{3s} - h_{0t}} \quad (1)$$

For an ideal gas as assumed in this study the isentropic change from condition 0 to 3s can be derived from the isentropic relation.

$$\frac{h_{3s}}{h_0} = \left(\frac{p_3}{p_0}\right)^{\frac{\gamma-1}{\gamma}} \tag{2}$$

To be able to rate two configurations with the same efficiency, the minimal skin friction coefficient $c_f = \frac{\tau_w}{\frac{1}{2}\rho c_w^2}$ was added to the objective function, to favor a design that is far away from separation over a design close to separation. In case separation is

detected a modified objective function is returned depending on the point of separation and the flow area expansion at the point of separation. A configuration which separated shortly after the blade exit hence gets a smaller value as one where the flow stays attached further downstream. In addition it has to be assured that the returned value is always smaller than in case of no separation. This approach helps to accelerate the convergence of the optimizer.

Results

Results of the optimization include a new set of flow angles for the last stage running blade and new geometrical parameters for the diffuser contour. The new shape compared to the reference diffuser is shown in figure 9. The area ratio of the optimized



FIGURE 9: 2D-FLOWPATH: OPTIMIZED VS REFERENCE

diffuser is roughly 20% larger as the reference. To check the optimization results with 3D CFD a new 3D design of the blade has to be created. The 3D design of the blade is created with a simple procedure. It is assumed that the flow angles behave approximately in the same way as the metal angles. The blade is thus twisted according to the difference in flow angle before and after the optimization, based on the SLEQ results. The shape of each section of the blade was not modified. This procedure seems quite feasible in order to show the general success of the optimization strategy.

Results of the 3D simulations are compared to the 2D optimization results. From the comparison of the calculated wall shear at the diffuser casing in figure 10, it can be seen that the flow behaviour in the diffuser is sufficiently captured. Same applies for the flow within the last stage, as the radial distribution of total pressure, swirl and velocity, presented in figure 11 show. The following remarks can be deduced from the 3D CFD results concerning the blade and diffuser design of the presented configuration. Taking a look at the meridional velocity, it is apparent, that the leaving loss at diffuser exit is considerably lower in the optimized design. This is a direct result of the changed total pressure distribution behind the blade. Whereas the reference case



FIGURE 10: COMPARISON OF BL-SOLVER WITH 3D-CFD

has a quite inhomogeneous total pressure profile, the optimized design shows a more or less constant distribution except for a small region near the hub. The distinct enhancement of the total pressure profile not only leads to a more homogeneous velocity distribution at the diffuser outlet but also to a higher pressure recovery within the diffuser. This is directly evident when looking



FIGURE 11: COMPARISON OF SLEQ WITH 3D-CFD

at the part load behaviour. In figure 12 the difference in pressure recovery to a reference value $\Delta c_p = c_p - c_{p,ref}$ and the stage efficiency are plotted against the axial Mach number at blade exit. The pressure recovery for the reference design drops markedly with decreasing load, whereas for the optimized design the pressure recovery stays almost constant. To confirm this an additional



FIGURE 12: PRESSURE RECOVERY AND EFFICIENCY

optimization has been conducted, where only the blade has been optimized. It can be seen that for the optimized blade the c_p value is significantly higher, although the diffuser is the same as in the reference case. For the diffuser design a coupled approach is thus essential as the diffuser flow field is obviously depending on the flow conditions provided by the last stage blade. A comparison of the reference design to the optimized diffuser shape (see figures 10 and 7a) shows that the skin friction in the optimized design is kept more or less constant at a very low level in the whole diffuser. This seems to point in the same direction that Stratford [17] already proposed as a criterion for an ideal diffuser. It seems desirable to keep the skin friction as constant as possible to avoid distortions or rapid changes of the velocity profile in the boundary layer.

Furthermore it can be seen that the meridional velocity distribution at blade exit has changed and is even more distorted in the optimized design. With regard to the leaving loss at blade exit this shows only a minor influence as illustrated in figure 13, where the difference between the calculated leaving loss and a reference value $\Delta P_{\text{loss}} = \frac{P_{\text{loss,ref}}}{P_{\text{loss,ref}}}$, at either the blade or diffuser exit is plotted against the axial Mach number at the blade exit. While the leaving loss calculated at diffuser outlet is reduced in the optimized design, the leaving loss calculated at blade exit stays basically the same compared to the reference case. From a further optimization where only the diffuser has been optimized one can see an appreciable increase of the leaving loss behind the blade, although the blade design has not been changed. This indicates the strong influence of the diffuser design on the pressure distribution behind the rotor blade (see also Kreitmeier [7]). This once again shows the need for a true coupling of both components in the design process.



FIGURE 13: LEAVING LOSS

All these effects add up to a considerable increase in stage efficiency over the whole investigated load range (see figure 12). Although it has to be pointed out that obviously no investigations have been conducted to validate the mechanical integrity of the optimized blade and it is hence questionable if the blade could be designed as proposed here.

CONCLUSIONS

It has been pointed out in the literature that by a separated stage and diffuser design a high potential for further efficiency increase remains unexploited. This affects both, the stage design, as it is for example depending on the pressure distribution impressed by the diffuser shape, and the diffuser design which strongly depends on the flow field entering the diffuser. A diffuser design based on some standard correlations will thus always lead to an underperforming combination of both. In the presented study a numerically simple and thus cost effective approach is demonstrated to allow for a coupled optimization of the last stage and diffuser. The study solely concentrates on axial radial diffusers. The tip clearance jet is especially significant for such an arrangement as has been pointed out before. The combination of a through-flow code and boundary layer solver have been validated and successfully coupled to an optimization algorithm. The capability of this approach has been demonstrated with the optimization of a representative last stage and diffuser. In contrast to most of the recent studies, where the emphasis is placed on higher order effects (e.g. rotor wakes) on diffuser performance, the present study concentrates only on first order effects. The results of this approach should therefore be seen as a good initial guess for a more sophisticated and detailed design. This is especially helpful as the authors are not aware of any standard correlations for bend diffusers including a wall jet or swirl.

ACKNOWLEDGMENT

The investigations were, in part, conducted within the joint research programme COOREFF-T in the frame of AG Turbo. The work was supported by the Bundesministerium für Wirtschaft und Technologie (BMWi) as per resolution of the German Federal Parliament under grant number 0327714C. The authors gratefully acknowledge AG Turbo and Siemens for their support and permission to publish this paper. The responsibility for the content lies solely with its authors.

REFERENCES

- McDonald, A., Fox, R., and Dewonstine, R., 1971. "Effects of Swirling Flow on Pressure Recovery in Conical Diffusers". *AIAA Journal*, 9(10), pp. 2014–2018.
- [2] Nicoll, W., and Ramaprian, B., 1970. "Performance of Conical Diffusers With Annular Injection at Inlet". *Journal of Basic Engineering*, 92, pp. 827–835.
- [3] Uvarov, V., Shkurikhin, I., and Molyakov, V., 1976. "Investigation of Joint Operation of Turbine Stages and of a Radial-Annular Diffuser with a Controlled Boundary Layer". *Thermal Engineering*, 23(5), pp. 18–20.
- [4] Kruse, H., Quest, J., and Scholz, N., 1983. "Experimentelle Untersuchungen von Nabendiffusoren hinter Turbinenstufen". *MTZ Motortechnische Zeitschrift*, 44(1), pp. 13–17.
- [5] Zimmermann, C., and Stetter, H., 1993. "Experimental Determination of the Flow Field in the Tip Region of a LP-Steam Turbine". In Proceedings of ASME Turbo Expo. 93-GT106.
- [6] Willinger, R., and Haselbacher, H., 1998. "The Role of Rotor Tip Clearance on the Aerodynamic Interaction of a Last Gas Turbine Stage and an Exhaust Diffuser". In Proceedings of ASME Turbo Expo. 98-GT-094.
- [7] Kreitmeier, F., and Greim, R., 2003. "Optimization of Blade-Diffuser Interaction for Improved Turbine Performance". *Journal of Power and Energy*, 217(4), pp. 443– 451.
- [8] Denton, J., 1978. "Throughflow Calculations for Transonic Axial Flow Turbines". *Journal of Engineering for Power*, *100*, pp. 212–218.
- [9] Schetz, J., 1993. Boundary Layer Analysis. Prentice-Hall.
- [10] Reichardt, H., 1951. "Vollständige Darstellung der turbulenten Geschwindigkeitsverteilung in glatten Leitungen". ZAMM, 31, pp. 208–219.
- [11] Clauser, F., 1956. "The turbulent boundary layer". Advances in Applied Mechanics, IV.
- [12] Hansen, N., and Ostermeier, A., 2001. "Completely derandomized self-adaptation in evolution strategies". *Evolutionary Computation*, 9(2), pp. 159–195.
- [13] Karlsson, R., Eriksson, J., and Persson, J., 1998. "An ex-

perimental study of a two-dimensional plane turbulent wall jet". *Experiments in Fluids*, **25**(1), pp. 50–60.

- [14] Musch, C., Sievert, R., Stüer, H., and Stoff, H., 2009. "Performance aspects of the shroud and cavity design in the last stage of a low pressure turbine". In Proceedings of the 8th European Turbomachinery Conference.
- [15] Polklas, T., 2004. "Entwicklung eines numerischen Verfahrens zur strömungsmechanischen Auslegung des Abströmgehäuses einer Niederdruck-Dampfturbine". PhD thesis, Universität Duisburg-Essen.
- [16] Becker, S., Gretschel, E., and Casey, M., 2005. "Influence of a tip clearance jet on a swirling flow in an axial-radial diffuser". In Proceedings of the 6th European Turbomachinery Conference.
- [17] Stratford, B., 1958. "An experimental flow with zero skin friction throughout its region of pressure rise". *Journal of Fluid Mechanics*, **5**, pp. 17–35.