[Extended Abstract]

**Improved Turbulence Prediction in Turbomachinery Flows and The Effect on Three-Dimensional Boundary Layer Transition**

Christoph Bode and Jens Friedrichs, Institute of Jet Propulsion and Turbomachinery, University of Braunschweig, Braunschweig, Germany

Dominik Frieling and Florian Herbst, Institute of Turbomachinery and Fluid Dynamics, Leibniz University of Hannover, Hannover, Germany

**Introduction**

Nowadays, the industrial design of turbomachines and their components will be done with three-dimensional Navier-Stokes solvers (CFD). These mostly RANS solvers are able to simulate multistage 3D blade passages with unsteady flow effects. Hence these types of solvers are the key design tool for today and tomorrow [1]. As an up-to-date numerical method for turbomachinery flows and their applications the two equation $k-\omega$ turbulence model after Wilcox (1988) [2] in combination with the $\gamma$-Re$_\Theta$ transition model after Menter and Langtry [3] which is used to incorporate laminar boundary layers and their transition to turbulence is widely spread. Whilst these numerical methods reduced the need for experimental investigations, even so they need to be validated against highly accurate experimental data since the turbulence and transition models should be able to correctly reproduce the physical flow phenomena inside a turbomachine like the transitional process on the viscous surfaces or the secondary flow.

From a present-day perspective these above mentioned models are able to numerical predict the midspan boundary layer behavior on the airfoils. Nevertheless even in a simple cascade there are still uncertainties in the prediction of the three-dimensional boundary layer behavior on the airfoils and the sidewalls (hub and tip) and their interaction with the secondary flow phenomenas, cf. [4, 5]. Furthermore, in a multistage component environment the interaction of these effects are increased and the prediction accuracy of the downstream blade rows is highly dependent on the prediction of the upstream blade rows. Thus, an improved steady and unsteady numerical method is necessary for the design of new multistage turbomachines and their components. For example an increased prediction accuracy of the turbulent kinetic energy (turbulence intensity) and its dissipation will lead to an improved boundary layer transition prediction. This in turn leads to a better prediction of the wake of the airfoils and hence more accurate flow condition for the downstream blade row.

The $\gamma$-Re$_\Theta$ transition model and its extinction to three-dimensional boundary layer transition after Menter and Smirnov [6] in combination with the SST model [7] was already validated against general testcases and also successfully applied to three-dimensional turbomachinery flows in Bode et al. [8] and showed its good agreement to experimental data. In the present paper the $k-\omega$ turbulence model
after Wilcoxon (1988) [2] with a modification after Bode et al. [9] to improve the turbulence prediction in combination with the transition model after Menter and Langtry [3] and its extinction to three-dimensional boundary layer transition after Menter and Smirnov [6] will be used to further improve the turbulence prediction and hence the transitional behavior and its impact on the loss prediction. Therefore the CFD solver will be validated against test cases with increasing complexity and will be presented to show the ability of the used numerical method to accurately predict the turbulence and transitional behavior of steady three-dimensional single and multistage turbomachinery components.

1. Numerical Method

An up-to-date numerical method, the parallel CFD-solver TRACE of DLR Cologne has been applied, cf. Nürnberg [10], Kügeler [11], Marciniak et al. [12]. In this solver, the three-dimensional Reynolds-averaged Navier-Stokes equations are solved on multi-block meshes by a finite volume technique. The convective fluxes are discretized by the 2nd order TVD upwind scheme of Roe [13] and the diffusive fluxes by a central differencing scheme. An implicit predictor-corrector time integration algorithm has been used for the steady simulations. The turbulence is modeled by the two-equation $k - \omega$ model of Wilcoxon (1988) [14], together with the Kato-Launder [15] fix for the stagnation point anomaly. The boundary layer transition has been modeled by the two-equation $\gamma - Re_\theta$ model of Menter and Langtry [3]. The model evaluates the local flow features to facilitate natural, bypass and separation induced transition as well as relaminarization and wake-induced transition. Furthermore the model is extended to incorporate cross-flow induced transition in three-dimensional boundary layers, cf. Bode et al. [8]. Turbulence length scale effects on turbulence and transition have been incorporated in the respective models, cf. Bode et al. [9]. In the post-processing integral boundary layer parameters are determined by integration of the velocity field perpendicular to the blade surface up to a point where the total pressure has increased by 99% of the whole velocity defect. A more detailed description of the method is given in Kožuluvić [16]. Furthermore, non-reflecting boundary conditions by Saxer and Giles [17] have been applied to the inlet and outlet boundaries.

1.1 Incorporation of Turbulence Length Scale Effects on Turbulence and Transition Prediction

The validation of today’s CFD-solvers especially on experimental cascade data with high inflow turbulence intensity from $3 \leq Tu \leq 10\%$ and in combination with moderate turbulence length scales ends up in unphysical too high the eddy viscosity leading to a wrong prediction of the turbulence and hence transitional flow. To avoid this behavior the CFD user often changes the turbulence length scale to fit the transitional data which is most probably wrong. Also the application of modified turbulence models sometimes leads to unphysical behavior around the leading edge and along more than 60% of the passages suction side where the eddy viscosity is damped to harsh. Therefore the $k - \omega$ turbulence model after Wilcoxon (1988) is modified, so that the "correct" behavior regarding overall characteristics and boundary layer development is given but the unphysical behavior of the eddy viscosity is reduced. For this reason, a criterion for the determination of viscous regions (boundary layers and wakes) has been developed as an additional element of the implemented approach (cf. [9]). This criterion is based on the large values of turbulent dissipation rate $\omega$. It takes the relationship between the turbulent dissipation rate estimated from the $k - \omega$ turbulence model and the turbulent dissipation rate in the free stream of the flow estimated by the new approach. The effect of the very high ratio in the boundary layer and wakes is used to separate them from the free stream.

$$b_v = \min\left(\max\left(\frac{\omega}{\omega_{FS}}, 0.1\right), 1.0\right).$$

The time-scale bound is only applied in these viscous regions, effectively preventing the eddy viscosity destruction in non-viscous areas by multiplying the time-scale bound by a factor $b_v$, which is 1.0 in
the boundary layer and the wake region and 0.1 in the free stream (cf. [9]).

\[ \mu_T = \frac{\rho_k}{\max(\omega, b, S)}. \]  

(2)

1.2 Extended γ-Re\(\Theta\) Transition Model For Three-Dimensional Boundary Layers

In Bode et al. [8], an extinction for crossflow induced transition according to [6] of the new γ transition model [18] was implemented in the γ-Re\(\Theta\) model framework [3]. In [8] the model in combination with Menter’s SST turbulence model was validated on different testcases with varying parameters like sweep angle and Reynolds number. Besides the classical swept wing testcases the new transition model was validated against a 6:1 spheroid which exhibits complex three-dimensional flow structures and therefore represents a challenging testcase for transition models. For these three testcases the new model showed good results in predicting the correct transition position compared to the available experimental data. For the purpose of turbomachinery flow the well known Durham cascade [19] was taken into account. Within this testcase the capability of the extended model with regard to boundary layer transition on the sidewalls of this testcase was shown.

2. Durham Cascade

The Durham turbine cascade is one of the most known and used CFD validation test cases. The cascade has been described in detail earlier, see for instance Walsh [20], Moore [19] and Moore and Gregory-Smith [19]. The cascade consists of six blades which have a profile typical for a high pressure turbine rotor. For a design inlet angle of \(\beta_1 = 42.75^\circ\) a turning of over 110° is achieved. With an axial blade chord of \(l_{ax} = 181\text{mm}\) and an outlet Mach number of \(M_{a2} = 0.1\) a Reynolds number, based on axial chord and exit velocity, of \(Re = 4.0 \cdot 10^5\) is obtained.

\[ Tu_1 = 4.6\% \text{ and } l_T = 4.4mm, \text{ where in Moore et al. [19] the turbulence intensity of } Tu_1 = 5.6\% \text{ and turbulence length scale of } l_T = 9.4mm \text{ is given. } \]

Contrary to the design an inlet angle of \(\beta_{IN} = 43.5^\circ\) was measured in the experiments. The computational domain for the Durham cascade is shown in figure 1. The applied grid (OH-structure) consists of 5,513,536 nodes (124 nodes in spanwise direction, 440 nodes around the blade surface, 53 nodes normal to the surface, half-span simulation) with a high low Reynolds resolution of the boundary layers. This results in an average dimensionless wall distance of \(y^+ \approx 1.0\) in a cell-centered scheme.

Inlet Flow Free stream flow conditions are derived experimentally and compared to the numerical ones at \(-1.0 \cdot C_{ax} \approx \) at position IN of Slot A, B and C, cf. figure 1. In Moore [21] detailed inlet velocity, turbulent kinetic energy coefficient and turbulent intensity profiles are given. Representative, figure 1 shows that the prescribed inlet velocity, turbulent kinetic energy coefficient as well as the turbulent intensity matches the experimental ones.

Spanwise Distribution For evaluation purpose numerical results of pitchwise averaged spanwise distribution of total pressure loss coefficient and outflow angle are shown and compared against experimental results [21] in Figure 2. Besides numerical results of the present investigated \(k-\omega\) Wilcox (1988) turbulence model with the extinction after Bode et al. [9] in combination with the extended γ-Re\(\Theta\) transition model after Menter and Langtry (namend hereafter with VB-CF) additional results from Bode et al. [8] for Menter’s \(k-\omega\) SST turbulence model in combination with the extended γ-Re\(\Theta\) transition model (SST-CF) are shown. The comparison of the numerical and experimental results shows an overall adequate prediction of the total pressure loss coefficient where the SST-CF gives a closer agreement with experimental data in the near sidewall region and the VB-CF gives better results in the midspan section downstream of the cascade.
Boundary Layer Behavior  The reason for that is seen in Figure 3. Here numerical results for the intermittency at boundary layer edge are shown for both combinations. Starting with the suction side surface in Figure 3 (a) and (b) it is well seen that the laminar region or transition location is more upstream predicted with the SST-CF compared to VB-CF resulting in more total pressure loss downstream of the cascade as seen in Figure 2. Contrary to that is seen in Figure 3 (c) and (d) where the SST-CF gives also a smaller laminar region compared to the VB-CF results but this is closer to the experimental data in Figure 2. All in all both numerical combinations show adequate results in predicting the laminar turbulent transition process on suction side and sidewall of the Durham cascade compared to experimental data.

3. Langston Cascade

Besides the well known Durham cascade the Langston cascade is one of the famous cascades in open literature and is also used by other researchers than Langston et al. [22], cf. Graziani et al. [23] and Holley et al. [24]. Like the Durham cascade the Langston cascade is also used to determine the state of the new formed boundary layer on the sidewall of the cascade.
3.1 First Numerical Results for Langston Cascade

Figure 4 shows first preliminary numerical results for the Langston Cascade in comparison with experimental data from Holley et al. [24] and Graziani et al. [23]. From a first view there is a good agreement between numerical and experimental derived saddle point and separation line infront of the cascade. Further investigations will be done to the final paper.

4. Low-Speed Axial Compressor

4.1 Low-Speed Axial Compressor Rig of the Institute of Turbomachinery and Fluid Dynamics at LU Hannover

Figure 5 gives a schematic overview of the low-speed axial compressor of the Institute of Turbomachinery and Fluid Dynamics. Contrary to the general convention in the turbomachinery community,
the flow in Fig. 5 goes from the right to the left. The test rig is designed as a closed air loop and exists of 20 inlet guide vanes, 30 blades and 26 vanes in each stage. The two stator and rotor rows as well as the inlet guide vanes are NACA 65 profiled. The geometry of the test rig, with a blade height of 140 mm and an axial distance between two blade rows of approximately 25 mm, is chosen to enable equipping measurement probes without unduly affecting the flow on the one hand, and obtaining a distinct quasi two-dimensional main flow region at mid span on the other hand. A detailed description of the test rig, the experimental data and their underlying post-processing is given by [?]. In order to investigate the rotor-stator interaction in the low-speed axial compressor, different steady and unsteady flow measurement techniques like surface-mounted hot film sensors, split-fibre probes and pneumatic probes have been deployed. The split-fibre probes, which are used to measure characteristic flow values like turbulence intensity, velocity and flow angles, are located upstream and downstream of stator 1. It is possible to traverse the probes over the complete blade height and over a 19° circumferential arc behind each blade. The split-fibre probe measurements, in conjunction with the surface-mounted hot-film sensors, deliver the required experimental data to analyze both the influence of rotor-wakes on the boundary layer development and the quality of its numerical prediction. In [?] the numerical prediction quality of the state-of-the-art turbomachinery design code TRACE has already been validated against the experimental data. Wolff et al. [?] conducted steady and unsteady RANS simulations and showed that only the first and last approx. 20% of the blade height are influenced by secondary flow effects. At mid span a two dimensional flow can be assumed. Therefore, only 15% of the blade height at mid span can be considered in quasi three-dimensional (Q3D) numerical simulations. The measurements and the numerical simulations have been conducted at the steady state rotor speed of 3000 rpm for three different operating points. The normalized operating parameters of the operating point near best efficiency (OPS70), the operating point near stability limit (OPS80) and the operating point near choke limit (OPS50) are given in Table 1.

### Table 1. Measured and Simulated Operating Points

| Operating Point | Mass Flow [kg/s] | Total Pressure Ratio | Description          |
|-----------------|------------------|----------------------|----------------------|
| OPS50           | 20.8             | 1.0525               | near choke limit     |
| OPS70           | 19.1             | 1.0725               | near best efficiency |
| OPS80           | 16.5             | 1.0825               | near stability limit |
**Figure 5.** Low-Speed Axial Compressor Rig at LU Hannover

### References

[1] J. D. Denton. Some limitations of turbomachinery. In *Proceedings ASME Turbo Expo, Glasgow, UK, Paper No. GT2010-22540*, 2010.

[2] D. C. Wilcox. *Turbulence Modeling for CFD, 3rd Edition*. DCW Industries, 2006.

[3] R. Langtry and F. Menter. Correlation-based transition modeling for unstructured parallelized computational fluid dynamics codes. *AIAA Journal, Band 47*, 12:2894–2906, 2009.

[4] B. M. Holley, S. Becz, and L. S. Langston. Measurement and calculation of turbine cascade endwall pressure and shear stress. In *Proceedings of ASME Turbo Expo, Reno-Tahoe, Nevada, USA, Paper No. GT2005-68256*, 2005.

[5] J. D. Denton and G. Pullan. A numerical investigation into the sources of endwall loss in axial flow turbines. In *Proceedings of ASME Turbo Expo, GT2012-69173, June 11-15, Copenhagen, Denmark*, 2012.

[6] F. R. Menter and P. E. Smirnov. Development of a rans-based model for predicting crossflow transition. In *Contributions to the 19th STAB/DGLR Symposium München, Germany*, 2014.

[7] F. R. Menter. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32, No. 8, August:1598–1605, 1994.

[8] C. Bode, J. Friedrichs, and D. Kozulovic. Abschlussbericht zum lufo iv/4 gtf-turb. Technical report, Institut für Flugantriebe und Strömungsmaschinen, Technische Universität Braunschweig, 2016.
[9] C. Bode, T. Aufderheide, D. Kozulovic, and J. Friedrichs. The effects of turbulence length scale on turbulence and transition prediction in turbomachinery flows. In Proceedings of the ASME 2014 Turbine Technical Conference and Exposition, Paper No. GT2014-27026, 2014.

[10] D Nürnberg. Implizite Zeitintegration für die Simulation von Turbomaschinenströmungen. PhD thesis, Ruhr-Universität Bochum, auch erschienen als DLR-Forschungsbericht 2005-11, 2005.

[11] E. Kügeler. Numerisches Verfahren zur genauen Analyse der Kühleffektivität filmgekühlter Turbinenschaufeln. PhD thesis, Ruhr-Universität Bochum, auch erschienen als DLR-Forschungsbericht 2007-20, 2007.

[12] V. Marciniak, E. Kügeler, and M. Franke. Predicting transition on low-pressure turbine profiles. In In: V European Conference on Computational Fluid Dynamics ECCOMAS CFD 2010, Lissabon, Portugal, 2010.

[13] P. Roe. Approximate riemann solvers, parameter, vectors and difference schemes. In Journal of Computational Physics, Band 43(2):S. 357-372, 1981.

[14] D. C. Wilcox. Reassesment of the scale-determining equation for advanced turbulence models. AIAA Journal, Band 26, 11:1299–1310, 1988.

[15] M. Kato and B. E. Launder. The modelling of turbulent flow around stationary and vibrating square cylinders. In Proceedings of the Ninth Symposium on Turbulent Shear Flows, Kyoto, Japan, August 16-18, 1993.

[16] D. Kozulovic. Modellierung des Grenzschichtumschlags bei Turbomaschinenströmungen unter Berücksichtigung mehrerer Umschlagsarten. PhD thesis, Ruhr-Universität Bochum, auch erschienen als DLR-Forschungsbericht 2005-11, 2005.

[17] A. P. Saxer and M. B. Giles. Quasi-three-dimensional nonreflecting boundary conditions for euler equation calculations. AIAA Journal of Propulsion and Power, Band 9(2):263–271, 1993.

[18] F. R. Menter, P. E. Smirnov, T. Liu, and R. Avancha. A one-equation local correlation-based transition model. Flow Turbulence Combustion, DOI 10.1007/s10494-015-9622-4, 2015.

[19] H Moore and D. G. Gregory-Smith. Transition effects on secondary flows in a turbine cascade. In Proceedings of ASME Turbo Expo, Paper No. 96-GT-100, 1996.

[20] J. G. C. Walsh. Secondary Flows and Inlet Skew in Axial Flow Turbine Cascade. PhD thesis, School of Engineering and Computer Sciences, The University of Durham, 1987.

[21] H. Moore. Experiments In A Turbine Cascade For The Validation Of Turbulence And Transition Models. PhD thesis, School of Engineering and Computer Sciences, 1995.

[22] L. S. Langston, M. L. Nice, and R. M. Hooper. Three-dimensional flow within a turbine cascade passage. Journal of Engineering for Power, 99:21–28, 1977.

[23] R. A. Graziani, M. F. Blair, J. R. Taylor, and R. E. Mayle. An experimental study of endwall and airfoil surface heat transfer in a large scale turbine blade cascade. ASME Journal of Engineering for Power, 102:257–267, 1980.

[24] B. M. Holley and L. S. Langston. Surface shear stress and pressure measurements in a turbine cascade. Journal of Turbomachinery, 131:031014–1, 2009.