AEROTHERMAL PREDICTIONS ON A HIGHLY LOADED TURBINE BLADE INCLUDING EFFECTS OF FLOW SEPARATION

E. Lutum – F. Cottier

MTU Aero Engines GmbH, Dachauer Str. 665, 80995 München, Germany,
ewald.lutum@mtu.de – francois.cottier@mtu.de

ABSTRACT

Aerothermal CFD investigations were conducted on a highly loaded turbine blade cascade for steady and periodic unsteady inflow conditions. Steady-state and transient computations were employed to investigate flow and heat transfer behaviour in separated flow regions in order to validate the applied numerical method. Effects of turbulence modelling were investigated, i. e. k-ε, k-ω, SST and BSL-EARSM models were applied in steady-state and SST and SAS models were used for transient computations. Modelling of laminar-turbulent transition was employed for SST, BSL-EARSM and SAS models. The influences of Reynolds number and free-stream turbulence level on flow separation were investigated concerning aerodynamic and heat transfer aspects. In general an over prediction of the pressure side flow separation was observable, i. e. the size of the separation zone was over predicted, which resulted in an under prediction of heat transfer in the computed separated flow regions compared to measurements. This effect was more pronounced with increasing free-stream turbulence intensity and Reynolds number level. Nevertheless, steady-state and transient computations with SST turbulence modelling qualitatively captured the investigated aerothermal effects. Supplementary effort undertaken with transient SST or SAS computations yielded no essential improvement compared to steady-state calculations.

NOMENCLATURE

c	[m]	chord length
h	[W/m^2/K]	heat transfer coefficient
Ma	[-]	Mach number
Nu	[-]	Nusselt number ($Nu=h\cdot c/\lambda$)
p	[m]	Airfoil pitch
Re_2	[-]	Reynolds number ($Re_2=U_2\cdot c/\nu$)
s	[m]	surface length
Sr	[-]	Strouhal number ($Sr=\omega\cdot c/U_1$)
Tu	[%]	turbulence level
U	[m/s]	velocity magnitude
λ	[W/m/K]	thermal conductivity
ν	[m^2/s]	kinematic viscosity

Subscript

1	inlet
2	exit
is	isentropic

INTRODUCTION

Environmental aspects as global warming or the shortage of natural resources require improvements and new technologies to further optimisation of the efficiency of gas turbines. This and economic issues create a continuous force on the aero engine and gas turbine industry towards lighter, more efficient, cleaner and cheaper products. Since inlet temperature and pressure ratio are also a main parameter to reduce the specific fuel consumption while increasing the power output, the inlet temperatures are rising continuously to increase the efficiency of the overall engine and achieving a reduction in CO_2 emissions. Since nowadays the HP turbine inlet temperatures are well above endurable material temperature limits advanced cooling concepts are required to ensure airfoil life requirements at lower coolant consumption. The combination of the requirements for lighter turbines and increased turbine inlet temperature is leading towards very high-lift technology, which might enable distinct separated flow regions on turbine airfoils. Hence a detailed understanding of flow separation and its impact on airfoil surface heat transfer is of great interest for a further improvement of the cooling methods. Accurate CFD prediction of turbine blade aerodynamics and heat transfer is an essential requirement to enable further improvements on turbine cooling designs. To achieve such a goal, low Reynolds number turbulence modelling in combination with well resolved near wall meshes are pre-conditions to enable accurate flow and heat transfer predictions, in particular in case of separated flow. Furthermore, correct modelling of laminar-turbulent boundary layer transition is another pre-condition (Luo and Lakshminarayana (1997)). A comprehensive review of laminar-turbulent transition phenomena and its role in aerodynamics and heat transfer in gas turbine engines has been provided by Mayle (1991). Due to the high free-stream turbulence levels and periodic unsteady passing wakes, as typically found in gas turbines, the most important transition phenomena are bypass and separated flow transition. Butler et al. (2001) studied the effects of turbulence intensity, length scale and Reynolds number on turbine blade heat transfer and its impact of laminar-turbulent transition as well as effects of suction side flow separation. Effects of distinct flow separation on the pressure surface were investigated by Wolff et al. (2001). The experimental investigations were performed on a large-scale plane T106-300 cascade blade, were the flow separation was triggered by off-design incidence angles. Corresponding steady state predictions were conducted by de la Calzada and Alonso (2002). Subsequent unsteady simulations by Alonso and de la Calzada (2003) confirm that the pressure surface separation is an unsteady phenomenon with significant impact on time dependent bubble structure and heat flux. Schobeiri et al. (2008) presented a detailed experimental investigation on a highly loaded low pressure turbine blade. Suction side flow separation was studies for individual and combined effects of free-stream turbulence intensities and periodic unsteady wake flows on aerodynamic and heat transfer behaviour.

Recently Ladisch et al. (2009) presented aerodynamic and heat transfer measurements on a highly loaded low pressure turbine airfoil with a separation bubble on the pressure surface. The experiments were conducted in a linear cascade at various free-stream turbulence intensities and Reynolds number levels. The effects on heat transfer, flow separation and laminar-turbulent transition were quantified. The results reveal a considerable influence of the boundary layer separation on the local heat transfer. The size of the separation region was strongly influenced by free-stream turbulence and Reynolds number levels.

The present investigation is part of the European research project AITEB-2 (Aerothermal Investigations of Turbine Endwalls and Blades), that is aimed at optimising cooling configurations of highly loaded turbine blades (Janke and Wolf (2010)). Steady state and transient computations are conducted to investigate flow and heat transfer behaviour in separated flow regions in order to validate the aerothermal predicting capability.

Turbine Blade Cascade

The intent of the blade design was to generate a large flow separation zone on the blade pressure side at nominal conditions to support the current investigations. The turbine blade profile is based on a design for a typical cooled low pressure turbine blade with a moderate to nearly high loading (p/c=0.772). As mentioned before a detailed experimental investigation with this blade cascade was conducted by Ladisch et al. (2009). These measurements were conducted in an atmospheric linear cascade facility as shown in Fig. 1, which allowed a variation of free-stream Reynolds number and turbulence intensity. The free-stream turbulence levels could be modified by using different turbulence grids at different positions in the inlet section of the test facility. In addition effects of periodic unsteady inlet conditions have been achieved by a spoke wheel wake generator upstream of the blade cascade. The wheel was placed outside of the channel and the spokes enter the test section through a slot on the side of the test section. By controlling the rotational speed of the wake generator different Strouhal conditions could be achieved. Local heat transfer distributions were obtained at mid span of the blade by using an iso-thermal measurement technique.

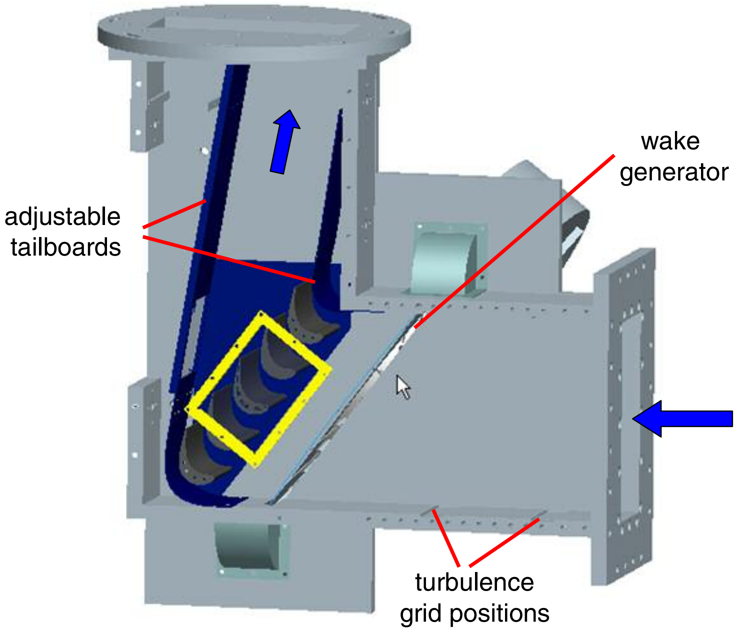

Fig. 1: Setup of the blade cascade test facility (Ladisch et al. (2009))

The commercial grid generator ICEM HEXA (ANSYS CFX) was used to generate multi-block structured meshes. The current mesh was optimised in a grid dependency study. In order to save computing resources only half span of the blade cascade was modelled using symmetry condition at mid span. The current mesh consisted of about 0.6 million nodes (Fig. 2) providing a high quality boundary layer resolution at the blade and endwall surfaces, i. e. $y^+<1.5$ for $Re_2=250000$.

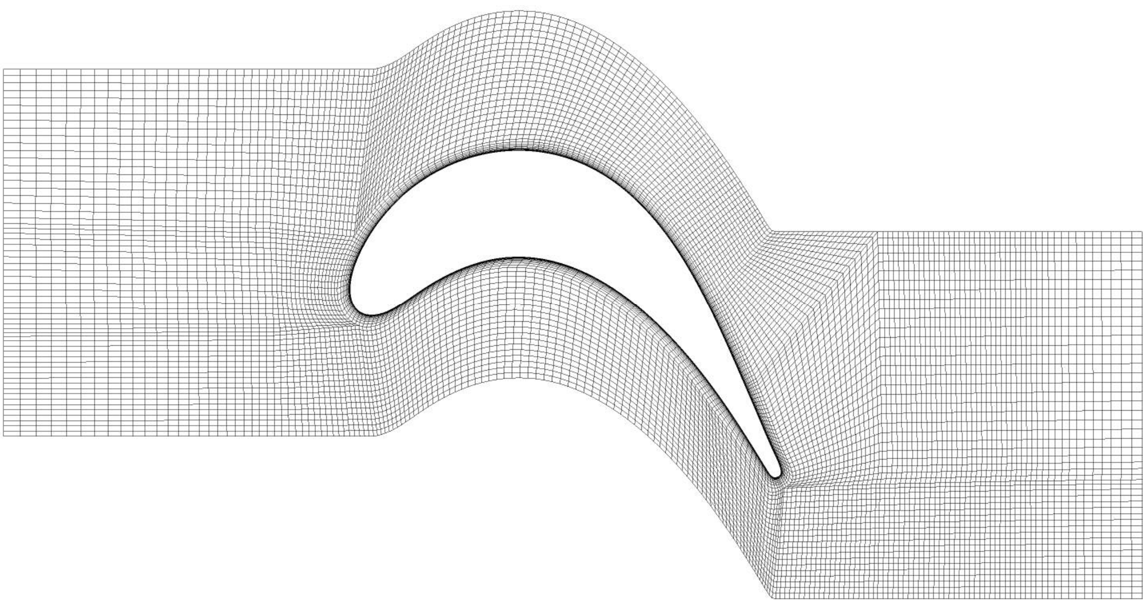

Fig. 2: Applied multi-block structured mesh

This numerical investigation was performed with the commercial solver CFX version 11.0 (ANSYS CFX). Automatic near wall treatment was applied, which means that the formulation (low Reynolds or wall function) depends on the local grid spacing at the wall. As the current mesh provides a well near wall resolution the low Reynolds formulation has been applied. The spatial discretisation is quasi 2nd order. Boundary condition profiles were obtained from corresponding experiments (Ladisch et al. (2009)) to perform a CFD validation. Tab. 1 provides an overview of the range of the investigated boundary conditions. An iso-thermal heat flux boundary condition, i. e. a constant wall temperature of T_{Wall}=300[K] was applied to be consistent with the experimental investigations. A constant incidence angle of 37° relative to the axial direction were applied for all investigated cases.

Parameter	Re=75000		Re=150000		Re=250000		Re=350000		Re=500000	
	Inlet	Exit	Inlet	Exit	Inlet	Exit	Inlet	Exit	Inlet	Exit
pressure, total [Pa]	101570		102234		103915		106529		112096	
pressure, static [Pa]		101209		101256		101259		101255		101249
temperature, total [K]	347		349		351		351		350	
inlet turbulence intensity [%]	1.6, 5.8, 7.9 and 13.2									
inlet length scale [mm]	16, 8.3, 20.7 and 13.6									

Tab. 2: Summary of boundary conditions at blade mid span

RESULTS AND DISCUSSION

Effects of turbulence modelling including laminar-turbulent transition influences were investigated first. Subsequently the impact of free-stream turbulence intensities and Reynolds number levels, especially regarding heat transfer and flow separation, will be studied. Finally transient simulations were performed, which are apparently appropriate for this separated flow case to verify if the prediction quality could be improved.

The simulation quality strongly depends on the used turbulence model. Modelling errors are very difficult to avoid, as they cannot be reduced systematically. There are several turbulence models available in ANSYS CFX, where each one shows limitations and strengths for different applications. Hence, four different turbulence models are used in this investigation to determine modelling errors: 1) Standard k-ε (k-epsilon), 2) Standard k-ω (k-omega), 3) Shear Stress Transport (SST) and 4) Baseline Explicit Algebraic Reynolds stress model (BSL-EARSM). Prediction of laminar-turbulent transition were enabled by applying the Langtry Menter gamma theta transition model ((Menter et al. (2006) and Langtry et al. (2006)) for the SST and BSL-EARSM turbulence models. Fully turbulent flow was assumed by k-ω and k-ε turbulence models. All used models were applied with default values as supplied by ANSYS CFX.

These computations were performed at airfoil design condition, i. e. for an exit Reynolds number of Re_2=250000 and inlet turbulence level of Tu_1=5,8%. Fig. 3 shows the isentropic Mach number distributions along the blade mid span. Negative values represent the pressure side and positive values the suction side. Except of the k-ε results the differences between the applied models are fairly small and a large flow separation zone on the pressure side has been predicted. The k-ω model show slightly better agreement with experiments concerning flow separation. On the suction side the SST and BSL-EARSM models indicated slightly better agreement with measurements, which might be explained by transition features.

Corresponding Nusselt number distributions at mid span are presented in Fig. 4. Leading edge heat transfer level is generally under predicted by all simulations in comparison with the experiment. The laminar-turbulent transition behaviour on the suction side is fairly well predicted by the SST model. Although the overshoot in heat transfer above fully turbulent values seen in measurement is not visible in the predictions. The heat transfer evolution along the pressure side is generally under predicted up to 50% blade chord. Further downstream, i. e. in the reattached region a good agreement between experiment and SST prediction is observable. The other turbulence models indicate a poor prediction quality compared with the measurements. Therefore, further steady state computations applied to investigate the effects of varied free-stream turbulence intensity and Reynolds number are conducted with the SST turbulence model.

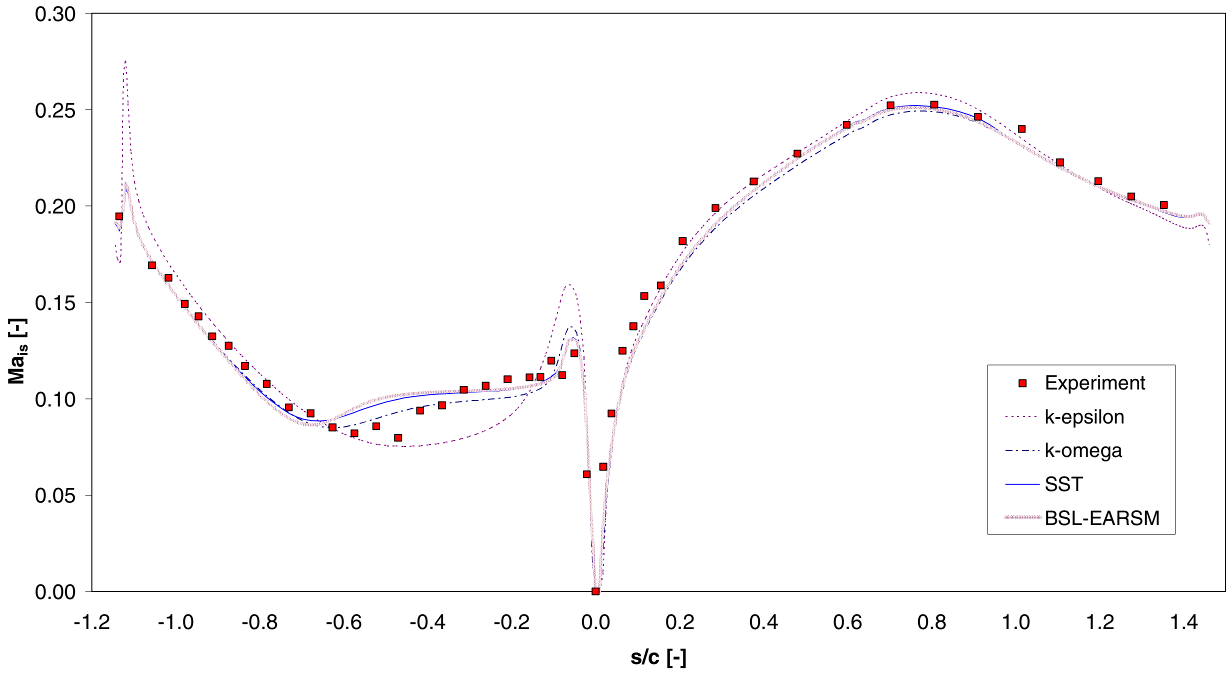

Fig. 3: Experimental and numerical distributions of isentropic Mach number at constant free-stream conditions (Re_2=250000 and Tu_1=5.8%)

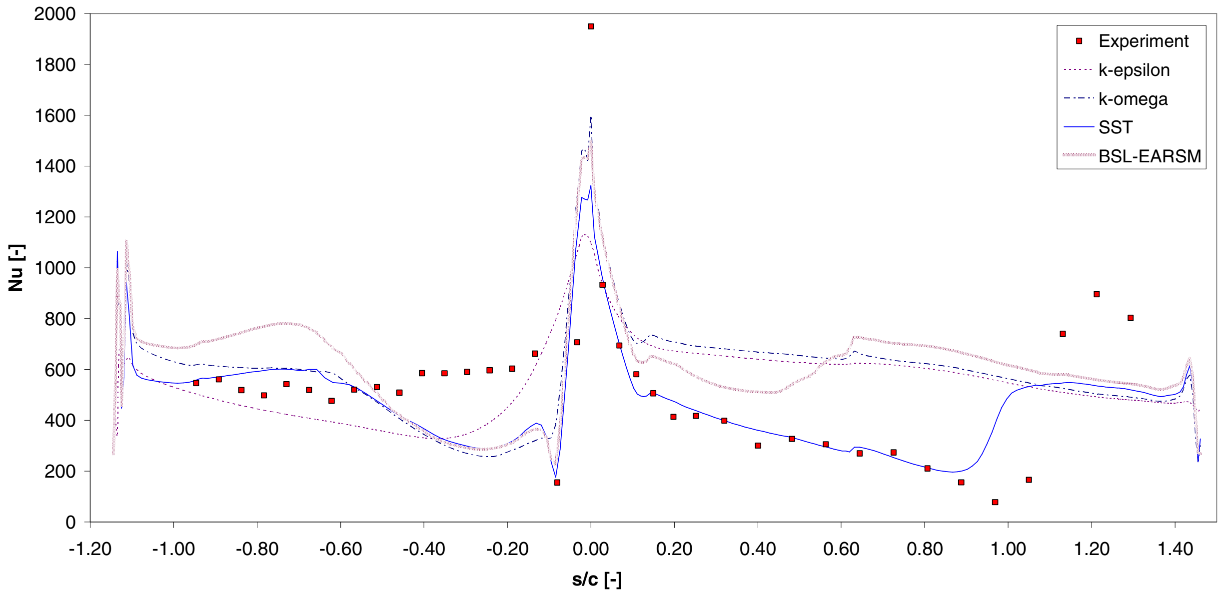

Fig. 4: Experimental and numerical distributions of Nusselt number at constant free-stream conditions (Re_2=250000 and Tu_1=5.8%)

Four different inlet profiles with free-stream turbulence intensities of Tu_1=1.6%, 5.8%, 7.9% and 13.2% at mid span have been applied during the following SST simulations at blade design Reynolds number condition of Re_2=250000. Fig. 5 shows the isentropic Mach number distributions at mid span along the blade pressure and suction side surface. The predictions achieve good agreement with the measurements on the suction side of the blade. The effects due to varied turbulence intensities are predicted correctly, especially the small flow separation occurring at about s/c≈1.0 at low free-stream turbulence intensity of Tu_1=1.6% was well predicted. On the pressure side significant differences are observable in the predicted flow separation region (-0.63<s/c<-0.08). The measurements show a decreasing flow separation length with increasing free-stream turbulence intensity. At the highest turbulence intensity (Tu_1=13.2%) only a small flow separation is observable. In contrast to this, all simulations indicate a large flow separation, which seems only very weekly affected by turbulence intensity. A good agreement is visible at low free-stream intensity. With increasing level of free-stream turbulence discrepancies grow. Downstream of flow separation again a good agreement between experiment and simulation can be observed.

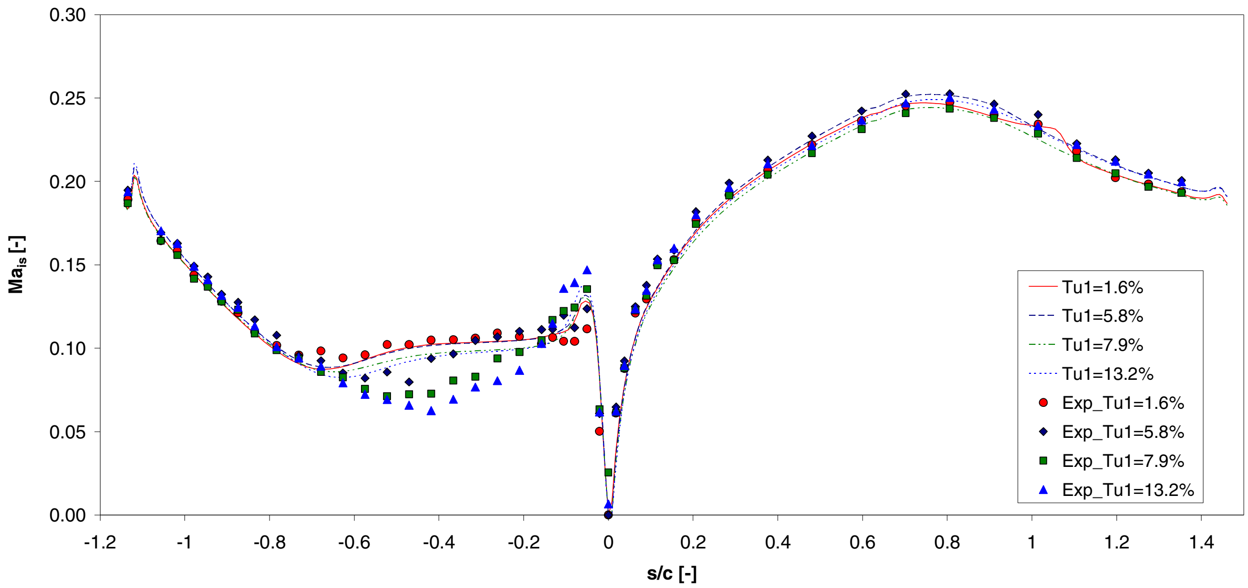

Fig. 5: Experimental and numerical distributions of isentropic Mach number at Re_2=250000 and varying free-stream turbulence level

Fig. 6 shows the corresponding Nusselt number distributions at varied free-stream turbulence intensives. As mentioned before, leading edge heat transfer level is generally under predicted compared with the experiment. Both production limiter available in CFX (clip factor and Kato Launder) to avoid unphysical production of kinetic energy in stagnation regions were applied. However, in this case no relevant difference could be found. All results shown in this paper are obtained with the clip factor production limiter. The laminar-turbulent transition on the blade suction side is very well predicted at the lowest free-stream turbulence level investigated. However, with increasing free-stream turbulence level predicted transition locations occur too early in comparison with experimental data. The measurements show only a weak dependence regarding free-stream turbulence according transition location.

The predictions indicate very little sensitivity to the free-stream turbulence level on the development of pressure side flow separation as discussed earlier in the context of isentropic Mach number results. For the investigated cases largely developed separation zones are predicted resulting in lower heat transfer levels in this region compared to experimental data. This is also true for the lower turbulence intensities at which experiments indicate also strong flow separation. Downstream of reattachment measurement and simulation converge and differences reduce.

Nusselt number results obtained at different Reynolds numbers at constant free-stream turbulence level of Tu_1=5.8% are shown in Fig. 7. Leading edge heat transfer is generally under predicted. The laminar heat transfer on the blade suction side is well predicted for the investigated range of Reynolds numbers. The predictions indicate in general an earlier transition compared with measurements. However, it is believed that this is mainly caused by the free-stream turbulence intensity as seen before in Fig. 6. The prediction of the pressure side flow separation length seems nearly not affected by the free-stream Reynolds number level. Best agreement with experiment is visible at the lowest free-stream Reynolds number cases, where also a strong developed separation zone occurs in the measurement. With increasing Reynolds number level the discrepancies between measurements and predictions grow, as the experiments indicate a strongly reduced flow separation for Reynolds numbers higher than Re_2=350000. This causes heat transfer levels to be under predicted within the estimated flow separation region and over predicted downstream of flow reattachment.

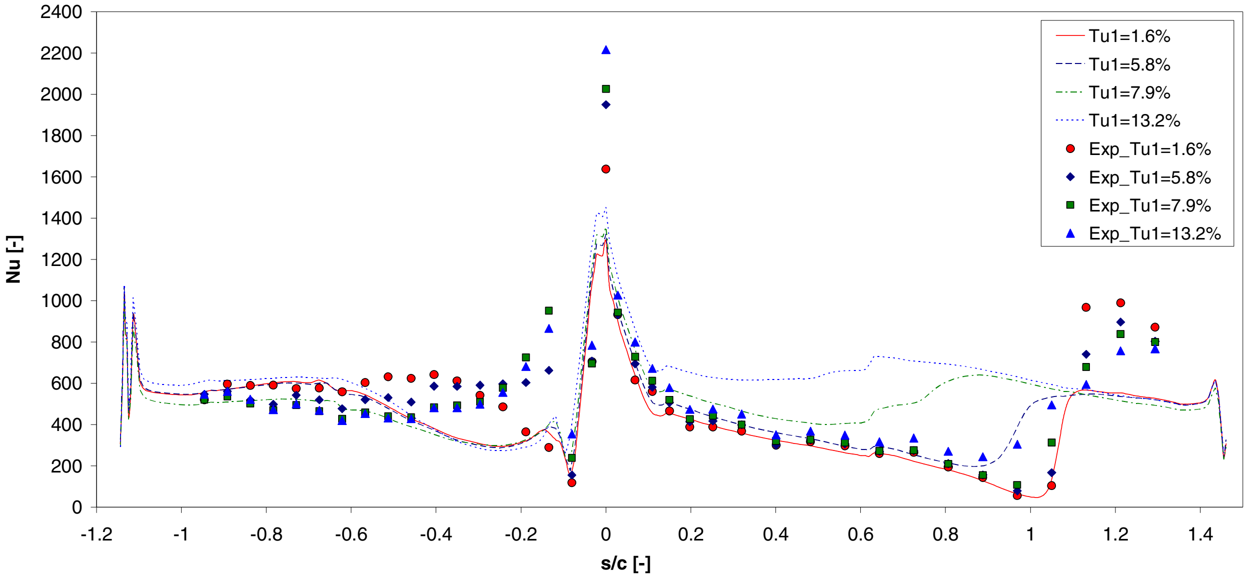

Fig. 6: Experimental and numerical distributions of Nusselt number at Re_2=250000 and varying free-stream turbulence level

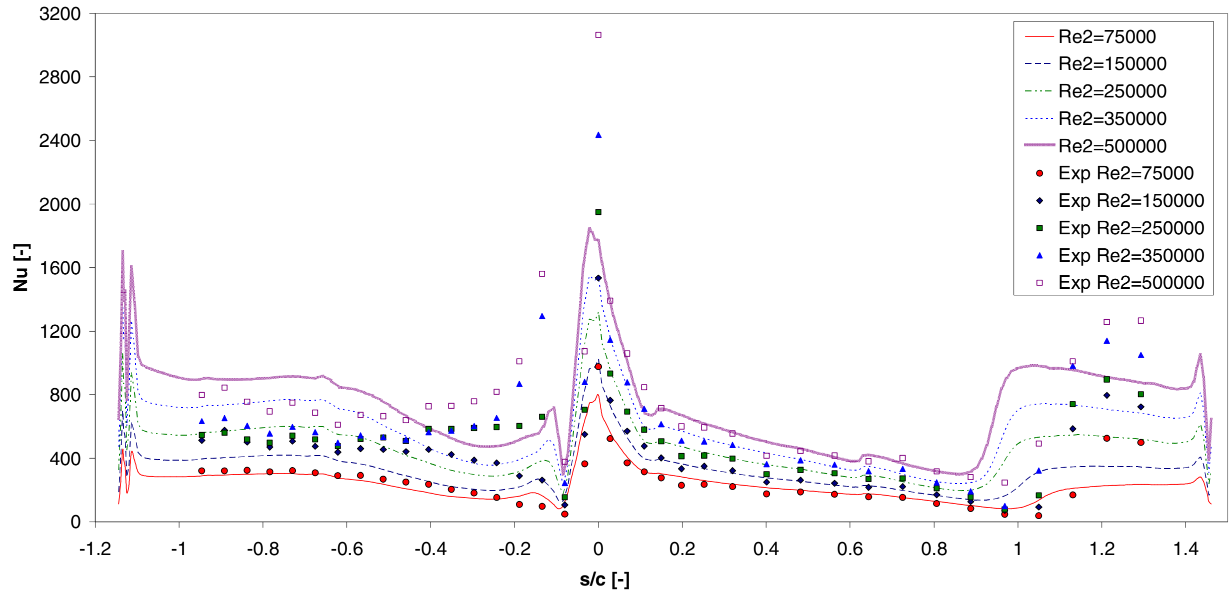

Fig. 7: Experimental and numerical distributions of Nusselt number at a free-stream turbulence level of Tu_1=5.8% and varying Reynolds number

The effects of periodic unsteady wakes result in a temporal periodic variation of the inflow conditions. These effects have been modelled here by periodic unsteady inlet conditions, such as velocity magnitude and an incidence angle variation as shown in Fig. 8. It can be observed that each wake is characterized by a decreased velocity magnitude and simultaneous increased incidence angle.

Transient CFD predictions have been conducted for periodic unsteady inlet conditions for a low Reynolds number level of Re_2=150000 at low free-stream turbulence intensity of Tu_1=1.6%. At this low Reynolds number and turbulence intensity level wake effects were expected to be present. In order to be consistent with steady state computations similar inlet profiles have been prescribed, which are super imposed with the above mentioned periodic unsteady wakes. The total temperature level at the inlet was kept constant.

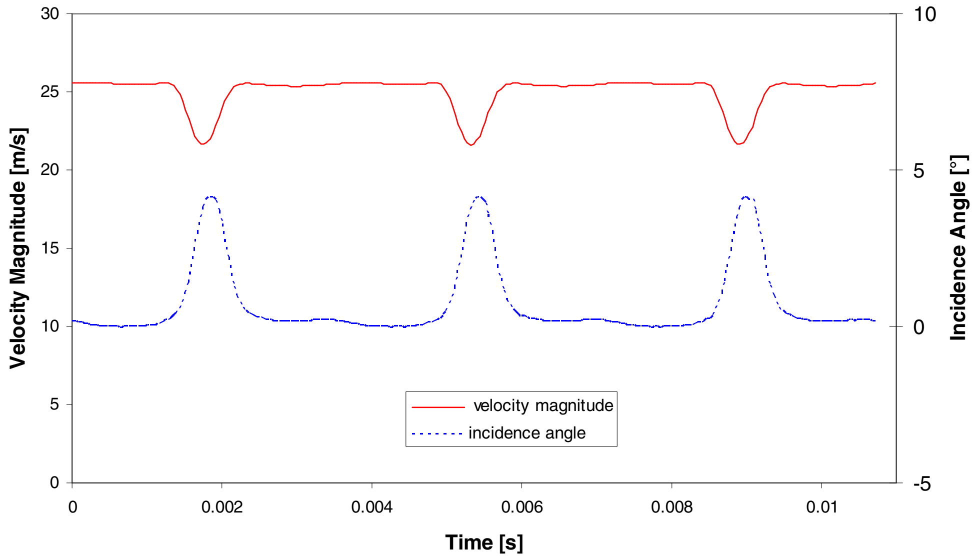

Fig. 8: Periodic unsteady inlet conditions, i. e. velocity magnitude and incidence angle over 3 wake periods for Re_2=150000, Tu_1 =1.6% and Sr=0.8

The Scale-Adaptive Simulation (SAS) turbulence model, which is based on the SST model, was additionally applied during these transient computations. The comparison between the steady state and the time averaged results derived from measurements and computations is shown in Fig. 9. The experimental heat transfer results show a slightly earlier suction side transition due to wake effects, which was also well captured by both transient computations. The measurements show a significantly weaker pressure side flow separation due to the periodic unsteady wakes. In contrast, the variation of inflow conditions cause no important difference between steady state and periodic unsteady predictions. In general only small differences can be observed between steady state and transient computations.

It is known that the SAS model requires sufficient mesh resolution. It was therefore decided to perform computations with a refined mesh. The symmetrical model indicates numerical errors for this low Reynolds number and low turbulence intensity case, which occurred close to mid span causing a non homogenous heat transfer distribution in span wise direction and generating a region of lower heat transfer. These features were not observed previously for higher Reynolds number and turbulence intensity levels and seemed to be linked with the symmetrical interface at mid span. Therefore, the current symmetrical mesh was extended to a full span geometry mesh. In a first step a full span mesh was created by mirroring the current mesh leading to a mesh consisting of 1.2 million nodes (full-span). Secondly, a refinement of approximately 1.3 in each direction of this mesh was established, which resulted in a mesh with 2.7 million nodes (full-span-fine).

Fig. 10 shows the results derived from the full span meshes for steady state and transient computations. The differences observable on the suction side are due to turbulence modelling aspects and have been seen earlier. The results on the pressure side indicate some interesting differences. Both, steady state and transient predictions conducted with the full span mesh indicate a significant increased heat transfer level on the late pressure side compared to the symmetrical mesh results (compare with Fig. 9). However, the use of the full-span-fine mesh for the SST or SAS computations does not indicate any improvement in pressure side flow separation prediction quality. The main conclusion from these unsteady analysis is that the time averaged results are similar to the steady state predictions.

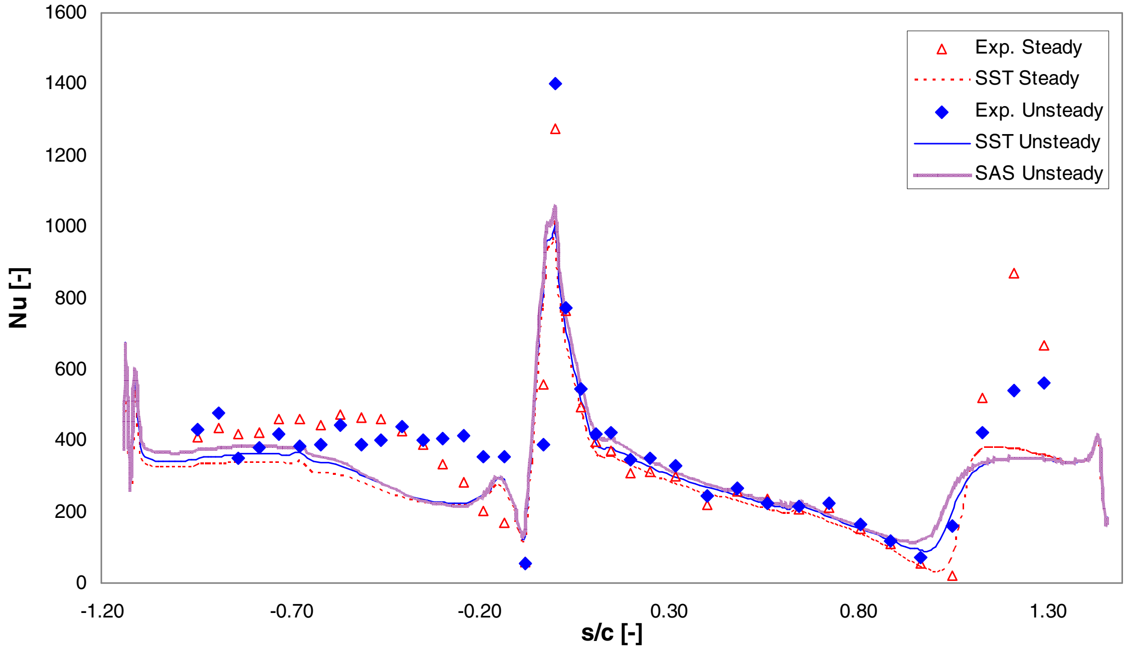

Fig. 9: Steady state and periodic unsteady experimental and numerical distributions of Nusselt number (Re_2=150000, Tu_1=1.6%, Sr=0.8)

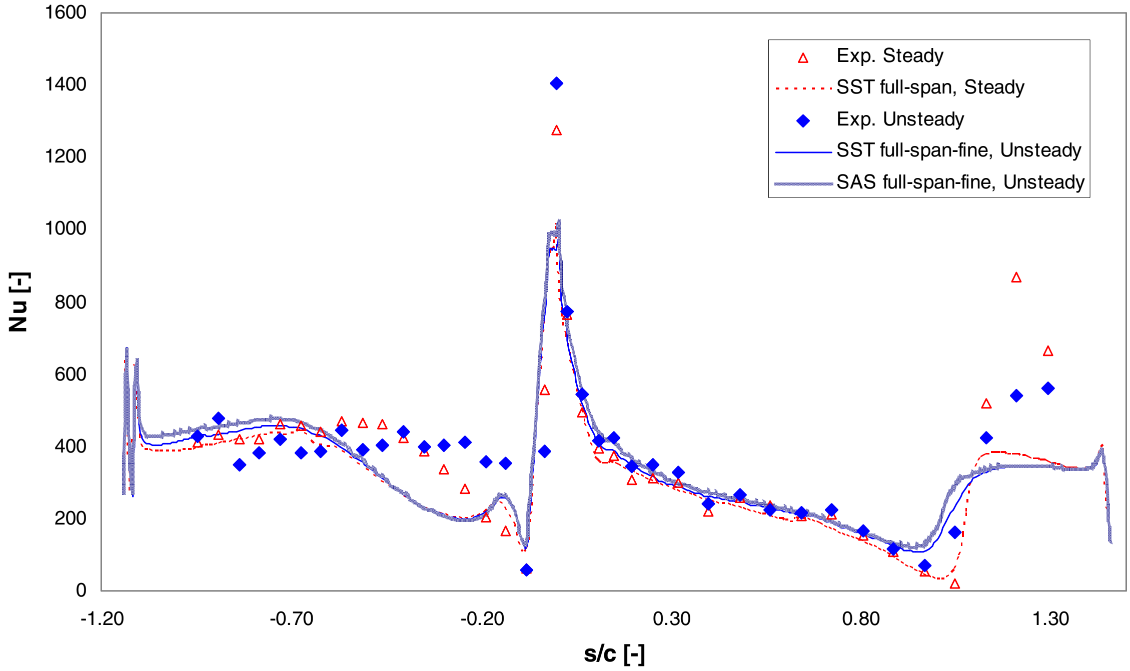

Fig. 10: Mesh influence on steady state and periodic unsteady experimental and numerical distributions of Nusselt number (Re_2=150000, Tu_1=1.6%, Sr=0.8)

CONCLUSIONS

Aerothermal CFD investigations on a highly loaded turbine blade cascade for steady and periodic unsteady inflow were conducted to investigate flow and heat transfer behaviour in separated flow regions. Effects of turbulence modelling were investigated, i. e. k-ε, k-ω, SST and BSL-EARSM models were applied in steady-state. Fully turbulent flow assumption is not able to correctly capture the suction side heat transfer development, which causes over predicted heat transfer levels in the pre-transition regions. Hence modelling of laminar-turbulent transition is essential. Laminar-turbulent transition was predicted by the SST and BSL-EARSM models using the gamma theta transition model. However, predicted transition locations occur upstream compared with measurements. This behaviour was stronger with increasing free-stream turbulence intensity and more pronounced for the BSL-EARSM model. The k-ε model did not predict any flow separation on the pressure side, while the k-ω based models showed strong flow separation zones. The SST model shows best overall prediction quality compared with the other applied models.

Effects due to varied free-stream turbulence intensities and Reynolds number levels on flow separation and heat transfer development were mainly studied by steady-state computations using the SST model with gamma-theta transition modelling. Leading edge heat transfer is generally significantly under predicted. The laminar-turbulent transition occurring on the blade suction side was also affected by the free-stream turbulence intensity. It was observed, that for higher free-stream turbulence levels, transition occurs far too early in comparison to experiments. In the region of laminar flow on the suction side, it was observed that heat transfer agrees well with experiments. However, an increase of free-stream turbulence intensity led to an upstream prediction of transition compared with the measurements and therefore an over prediction of heat transfer.

The pressure-side flow separation was detected by the simulations. However, compared to measurements the separated flow region was over predicted especially at higher free-stream turbulence intensity levels causing an under prediction of heat transfer in these regions. Downstream of flow re-attachment, calculated and measured results agreed well.

The laminar heat transfer on the blade suction side is well predicted for the investigated range of Reynolds numbers. In general, the predictions indicated an earlier transition compared with measurements and a lower heat transfer level directly downstream of transitions. It is believed that this discrepancy is mainly caused by the free-stream turbulence intensity. The prediction of the pressure side flow separation length seems nearly unaffected by free-stream Reynolds number level. Best agreement with experiment occurs at the lowest free-stream Reynolds number cases, where also a strongly developed separation zone occurs in the measurements. With increasing Reynolds number level, the discrepancies between measurements and predictions grow, as the experiments indicate a strongly reduced flow separation for Reynolds number higher than Re_2=350000. This causes heat transfer levels to be under predicted within the estimated flow separation region and over predicted downstream of flow re-attachment.

Transient computations performed with periodic unsteady inflow conditions show very little impact on the predicted results. No improvement in prediction quality could be achieved similarly to steady-state computations. Pressure side flow separation and associated heat transfer development could not be adequately captured.

These investigations showed that steady-state and transient computations are able to qualitatively capture the aerothermal effects of Reynolds number and free-stream turbulence intensity for a turbine blade cascade with a distinct flow separation region. However, to some extent large deviations between numerical and experimental results were observed. Additionally, these deviations could not be improved by transient computations.

ACKNOWLEDGEMENTS

The authors would like to express their gratitude to Mr. D. Böpple for his contributions to this work. This research was performed within the European research project "Aero-Thermal Investigation on Turbine End-Wall and Blades" (AITEB-2, 6FP, AST4-CT-2005-516113). The permission for the publication is gratefully acknowledged by the authors.

REFERENCES

Alonso, A. and de la Calzada, P.; "Steady/Unsteady Numerical Simulation of Heat Transfer in Turbine Cascades With Separated Flows", European Turbomachinery Conference, Prague 2003.

ANSYS CFX, http://www.ansys.com/products/fluid-dynamics/cfx/.

Butler, R. J., Byerley, A. R., VanTreuren, K. and Baughn, J. W., “The Effect of Turbulence Intensity and Length Scale on Low Pressure Turbine Blade Aerodynamics”, Int. J. Heat and Fluid Flow, 22, 2001, pp. 123-133.

De la Calzada, P.; Alonso, A.; "Numerical Investigation of Heat Transfer in Turbine Cascades with Separated Flows", ASME Turbo Expo, GT-2002-30225.

Janke, E. and Wolf, T., 2010, “Aerothermal Research for Turbine Components - An Overview of the European AITEB-2 Project”, ASME Turbo Expo, GT-2010-23511.

Ladisch, H., Schulz, A. and Bauer, H.-J., “Heat Transfer Measurements on a Turbine Airfoil with Pressure Side Separation”, ASME Turbo Expo, GT2009-59904.

Langtry, R. B., Menter, F. R., Likki, S. R., Suzen, Y. B., Huang, P. G., and Voelker, S., 2006, “A Correlation-Based Transition Model Using Local Variables – Part II: Test Cases and Industrial Applications”. ASME J. of Turbomachinery, 128 (3), pp. 423–434.

Luo, J. and Lakshminarayana, B., 1997, “Numerical Simulation of Turbine Blade Boundary Layer and Heat Transfer and Assessment of Turbulence Models”. Journal of Turbomachinery, Vol. 119, pp. 749-801.

Mayle, R. E., 1991, “The Role of Laminar-Turbulent Transition in Gas Turbine Engines”. ASME J. of Turbomachinery, 113 (3), pp. 509-537.

Menter, F. R., Langtry, R. B., Likki, S. R., Suzen, Y. B., Huang, P. G., and Völker, S., 2006, “A Correlation-Based Transition Model Using Local Variables – Part I: Model Formulation”. ASME J. of Turbomachinery, 128 (3), pp. 413–422.

Schobeiri, M. T., Öztürk, B. Kegalj, M. and Bensing, D., “On the Physics of Heat Transfer and Aerodynamic Behavior of Separated Flow Along a Highly Loaded Low Pressure Turbine Blade Under Periodic Unsteady Wake Flow and Varying of Turbulence Intensity”, J. Heat Transfer, Vol. 130, 2008.

Wolff, S., Homeier, L. and Fottner, L.; "Experimental Investigation of Heat Transfer on Separated Flow on a Highly Loaded LP Turbine Cascade", Proceeding of the RTO/AVT Symposium, Norway 2001.