Proceedings of the International Gas Turbine Congress 2003 Tokyo
November 2-7, 2003

IGTC2003Tokyo TS-083

Conjugate Heat Transfer Analysis of a Test Configuration for a Film-cooled Blade

Karsten KUSTERER[1], Dieter BOHN[2], Takao SUGIMOTO[3], Ryozo TANAKA[3]
[1]B&B-AGEMA, Gesellschaft für Energietechnische
Maschinen und Anlagen GmbH,
Jülicher Str. 338, 52070 Aachen, GERMANY
Phone: +49-241-5687810, FAX: +49-241-5687879, E-mail: kusterer@bub-agema.de
[2]Institute of Steam and Gas Turbines, Aachen University, GERMANY
[3]Kawasaki Heavy Industries, LTD., Akashi, JAPAN

ABSTRACT

The conjugate calculation technique is used for the three-dimensional thermal load prediction of a film-cooled test blade of a modern gas turbine. The comparison with thermal index paint experiments shows that with respect to regions with high thermal load a qualitatively good agreement of the conjugate results and the measurements can be found although the numerical models contain several simplifications for the internal cooling. In particular, the tip region of the trailing edge is exposed to a high thermal load, which requires further improvement of the cooling arrangement. Altogether the achieved results demonstrate that the conjugate calculation technique is applicable for reasonable prediction of three-dimensional thermal load of complex cooling configurations for blades. Furthermore, the conjugate approach permits the calculation of the specific heat transfer rate distribution on the blade surface without prescribing heat transfer boundary conditions. The results show a distinctly inhomogeneous distribution of the heat transfer rate in the leading edge region of the blade.

NOMENCLATURE

q	[Wm^{-2}]	heat transfer rate
s^+, s^-	[-]	blade surface coordinate
T	[K]	temperature
λ	[$Wm^{-1}K^{-1}$]	heat conductivity
Indices		
fl		fluid
o		stagnation value
ref		reference value
s		solid
w		wall
Abbreviations		
CCT		Conjugate Calculation Technique

INTRODUCTION

Great efforts are still put into the design process of advanced film cooling configurations. In particular, the vanes and blades of turbine front stages have to be cooled extensively for a safe operation. Therefore, precise heat transfer analysis is essential in the design process in order to reach a necessary reliability and availability of the components. A design failure can lead to a malfunction of the turbine in a very short time, which causes high repair costs and downtime costs. For improving the design process by reducing time and costs, the further development of modern numerical tools is required, which are capable to detect possible deficiencies in the cooling design, e.g. hot spots, as early as possible.

Conventional design processes rely on empiric correlations for the internal and external heat transfer mainly based on extensive experimental results for standard flow situations. For real blade applications this strategy includes various assumptions and uncertainties. Thus, the results suffer from some inaccuracies leading to a higher demand of numerous expensive test runs with longer development time.

The Conjugate Calculation Technology (CCT) offers a significant potential for improving the cooling design process. The heat transfer at contacting walls with external and internal flows is the result of the calculation and, thus, no additional boundary conditions on the heat transfer are necessary. A remaining question is on the applicability of this technology on real blade cooling configurations. Therefore, extensive numerical investigations on a test configuration for a turbine blade together with an industrial gas turbine manufacturer have been performed. The results demonstrate that the CCT implemented to the CHTflow solver (Bohn et al., 2001) is applicable for modern gas turbine cooling simulation and can be used for the successful numerical testing of real cooling configurations.

TEST CONFIGURATION

An experimental test configuration has been developed by Kawasaki Heavy Industries (KHI), LTD., for the film cooling of a first stage blade of a modern gas turbine (Sugimoto et al., 2002). The test configuration has been used for investigations of the influence of off-design conditions on the thermal load of the blade. At the blade leading edge, the configuration consists of three rows of radially inclined cooling holes (indicated as "P1", "LE", and "S1" in Fig. 1), which are supplied by a single cooling channel. Furthermore, the experimental test configuration also includes two rows of shaped holes, one on the suction side (indicated as "S2") and one on the pressure side (indicated as "P2") respectively, supplied by further internal cooling passages as shown in Fig. 1. The leading edge is supplied directly by a separate cooling channel (no. I) whereas the other two channels are typically serpentine-shaped (Fig. 2). The trailing edge chamber is supplied by channel no. III through several cross-over holes before the cooling air is ejected through a row of small slots at the trailing edge. For augmentation of the convective heat transfer, the internal walls of the passages are equipped with small squared ribs. Furthermore, a large number of pin-fins are to be found in the trailing edge

Manuscript Received on March 31, 2003

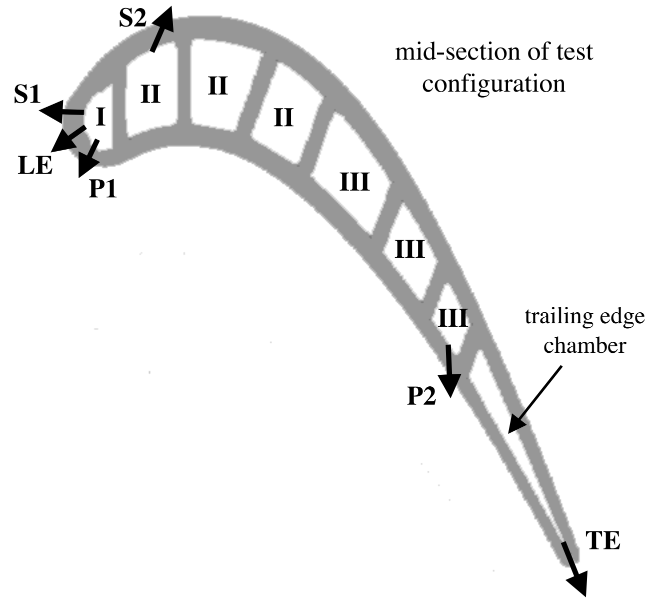

Fig. 1 Film-cooled test blade

a) internal cooling system (w.o. trailing edge chanmber)

b) leading edge channel

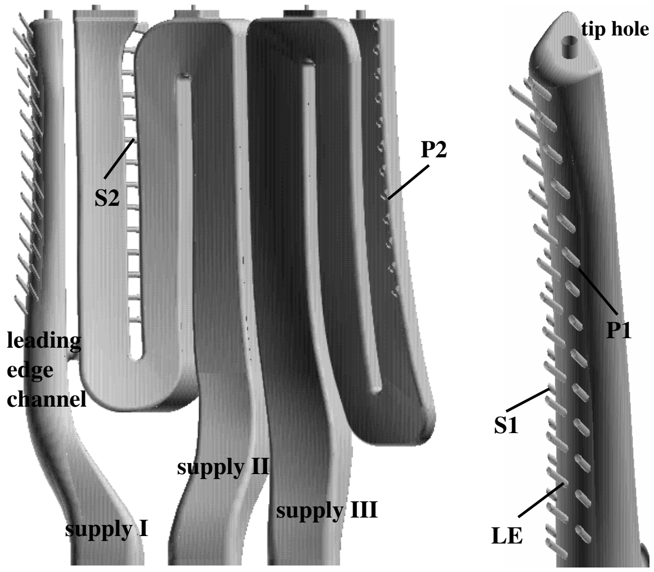

Fig. 2 Internal cooling system of test configuration

chamber. The configuration has been analysed experimentally under hot gas conditions by KHI at Akashi R&D Center. Measurement data on the thermal load were obtained by thermal index paint experiments. All conjugate calculations for the test configuration were done in a blind test case and the experimental results had not been presented before the simulation results had been submitted to KHI.

NUMERICAL METHOD AND MODELS

Conjugate Calculation Technique

It is obvious that conventional approaches on the heat transfer determination suffer from some uncertainties and inaccuracies, in particular, if the data is transferred to the real blade flow. The interaction of the heat transfer and the fluid flow is of importance for the precise determination of the heat transfer. The heat transfer will affect the development of the film cooling flow, in particular the secondary flow structures in the cooling jets. Thus, a variation of heat transfer coefficients depending on the flow structure is the result. Furthermore, for a real blade the additional convective cooling effects are also of importance. One main effect is that the cooling fluid is heated convectively on the way through the supply channels and the cooling holes. Thus, the cooling fluid condition at the hole exit varies with the internal heat transfer and, furthermore, has an influence on the external cooling performance.

For the numerical simulation, the conjugate calculation technique used in the CHTflow code (e. g. Bohn and Bonhoff, 1994, Bohn et al., 1995) offers the opportunity to avoid the use of the film cooling heat transfer boundary conditions and allows a direct calculation of the heat transfer and the wall temperatures. The numerical scheme for the simulation of the fluid flow and heat transfer works on the basis of an implicit finite volume method combined with a multi-block technique. The physical domain is divided into separate blocks for the fluid and solid body regions. Full, compressible, two- or three-dimensional Navier-Stokes equations are solved in the fluid blocks. The closure of the Reynolds averaged equations is provided by the Baldwin-Lomax algebraic eddy-viscosity turbulence model (Baldwin and Lomax, 1978).

The Fourier equation is solved in the solid body blocks. Coupling of fluid blocks and solid body blocks is achieved via a common wall temperature resulting from the equality of the local heat fluxes passing through the contacting cell faces:

$$T_W = \frac{\frac{\lambda_s}{\lambda_{fl}} T_s + T_{fl}}{1 + \frac{\lambda_s}{\lambda_{fl}}} \tag{1}$$

This means that no heat transfer boundary conditions have to be stipulated on the solid surfaces as in conventional numerical simulation without the conjugate technique. This method of calculating the heat fluxes requires a very high grid resolution at the contacting block faces. In particular, the numerical grid for the fluid flow calculation should allow an adequate resolution of the laminar sublayer. The use of a principally identical formulation and solution of the energy equation in the solid body blocks as in the fluid blocks is advantageous for the implementation and stability of the coupling procedure (homogeneous method). Other conjugate calculation approaches have been presented also by several authors (e.g. Kao and Liou , 1996, Han et al., 2000, Montenay et al., 2000, Li and Kassab, 1994, Okita and Yamawaki, 2002, Heidmann et al., 2003, York and Leylek, 2003).

Conjugate Models for the Film-cooled Blade

Due to the complexity of the complete configuration, it has been decided to divide the conjugate calculation into two different tasks in order to reduce the calculation effort. Task 1 deals with the modelling and simulation of the leading edge cooling, whereas task 2 neglects the leading edge ejection and the leading edge supply channel.

a) model for leading edge simulation (task 1)

For the conjugate calculation of the leading edge region, a 3-D numerical grid consisting of nearly 3.1 million grid points in 181 blocks has been generated. The numerical grid consists of the complete blade passage, the radial gap in a simplified model, all cooling holes of the three rows at the leading edge (altogether 42 holes), and the leading edge supply channel. The ribbed walls have been modelled as smooth walls.

With respect to the CCT, additional solid body blocks in the leading edge region have been included in the model. Therefore,

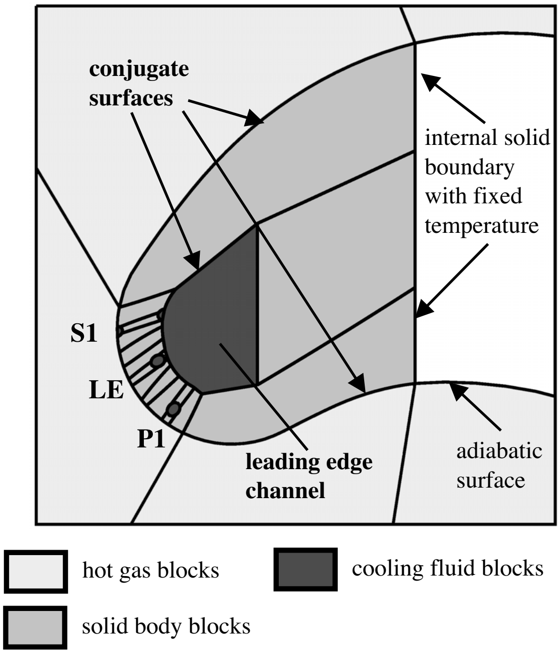

Fig. 3 Illustration of the blade leading edge model for the CCT

direct coupling of the solid body and the fluid flow regions is established in the leading edge region and the internal and external heat transfer are taken into account during the calculation. Figure 3 shows the leading edge and the boundary of the solid body calculation region. At the internal solid body boundary a fixed temperature has been prescribed. Thus, it will be possible to consider the effects of this fixed boundary condition, when the thermal load at the leading edge is investigated. Due to the limitations of the model several effects have to be taken into account in the evaluation of the results:

- Convective cooling effects by other supply channels than the leading edge channel are not part of the calculation.
- Surface temperatures and solid body temperatures in the leading edge region will be affected by the fixed thermal boundary condition.
- Enhanced heat transfer by ribs in the supply channel is not part of the model.

b) model without leading edge cooling (task 2)

For the conjugate calculations, a 3-D numerical grid consisting of nearly 4.4 million grid points in 253 blocks has been generated. The numerical grid consists of the blade internal passages (except leading edge passage), the radial gap (complex model including the tip outlets), all cooling holes of the suction side row with shaped holes, pressure side row with shaped holes, and the trailing edge row of ejection slots. The trailing edge chamber has been modelled without the pin fins and all passage walls have been calculated as smooth walls.

With respect to the CCT, the solid blocks for the blade itself have been limited to the upper part of the blade. Thus, the blade is divided into an adiabatic lower part and a full-conjugate upper part. Due to the neglecting of the of the leading edge cooling, the thermal load in the front part of the blade is calculated by far too high. Therefore, in the evaluation of the results the effect of heat conduction from the hot leading edge to the other parts of the blades has to be taken into account.

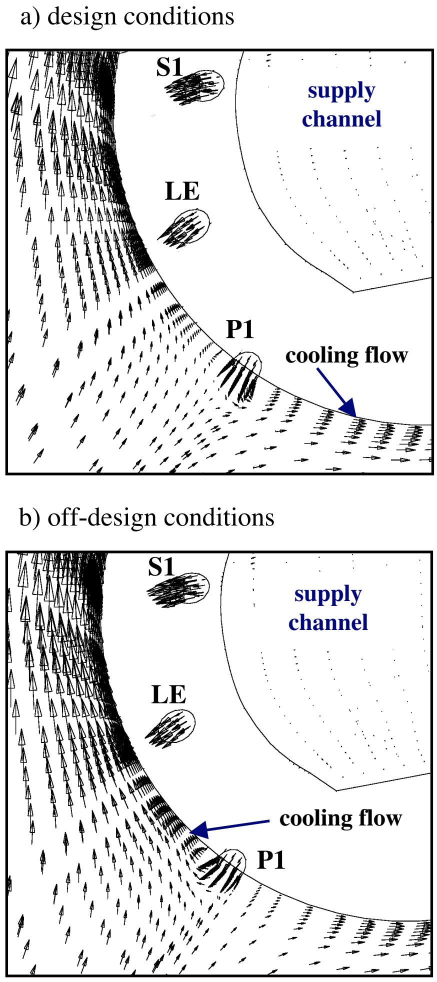

Fig. 4 Zoomed view of leading edge (radial cutting plane in mid-section), flow vectors

RESULTS ON THERMAL LOAD

Task 1: Leading Edge Cooling

Figure 4 shows the vector plots in a radial cutting plane for the leading edge ejection under design and off-design conditions. Figure 4a demonstrates that under design conditions the cooling fluid ejected by row "P1" is distributed along the blade pressure side as supposed. In Fig. 4b, it can be shown that due to the movement of the stagnation line the cooling fluid ejected by the hole of row "P1" flows around the leading edge. Thus, the leading edge part of the pressure side remains widely unprotected resulting in unacceptable high material temperatures in that region. The hot region can clearly be detected by the surface temperature analysis in Fig. 5, which is a result of the Conjugate Calculation Technique. The surface temperature has been normalised by the averaged total temperature at the inlet as a reference temperature and is plotted along the surface coordinate for the leading edge area as indicated in the drawing. For the off-design condition, the temperature peak on the pressure side of the leading edge becomes obvious whereas under design conditions a homogeneous distribution at a lower temperature level is reached.

For the off-design conditions a thermal index experiment has been performed by KHI at Akashi R&D Center. A comparison of the experimental data with the numerical results for pressure side part of the leading edge is given by Fig. 6. Black lines in the numerical result indicate block boundaries of the structured multi-block grid.

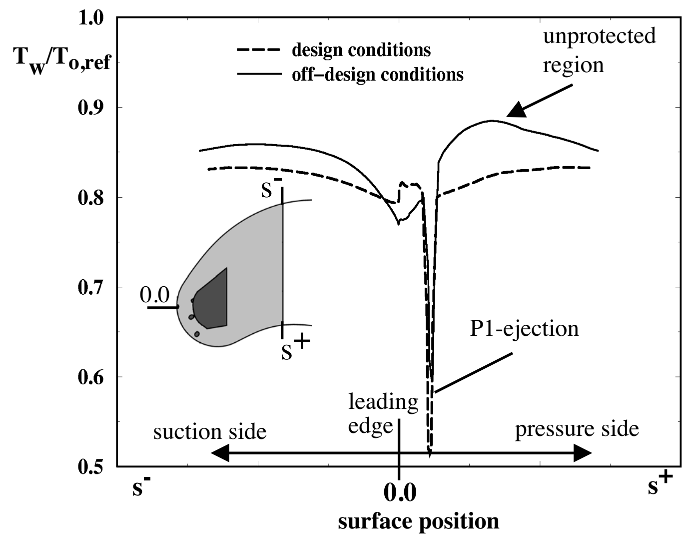

Fig. 5 Surface temperature distribution in leading edge area

Due to the movement of the stagnation line the pressure side shows a region of high thermal load (region A), which is not covered by the cooling fluid. This region can be found in the thermal index paint results as well as in the numerical results, but, in the calculation region A is predicted at a higher radial position. Furthermore, both results show a region of low thermal load (region B) at the blade tip of the leading edge. The streamline distribution of the cooling air reveals that cooling air from some of the upper holes is partly able to flow in this region leading to lower thermal loads by protecting the surface from the hot gas attack. Region B is to be found larger in the experiments. The reason for the differences in region A and region B with the experimental data lies in the boundary conditions, in particular in the off-design flow angle. Based on these results, it has been concluded that the estimated value for the off-design angle of the flow at the leading edge is somewhat too high in the calculation. If the off-design flow angle would be reduced in the calculation, the amount of cooling fluid to be able to flow in region B will be increased leading to an enlargement of region B and a movement of region A.

The predicted low thermal load in region C is not found in the thermal index paint measurement. In the calculation, it has been detected that in this region the convective cooling by the rows LE and S1 is increased because of a higher cooling mass flow in the cooling holes. The conjugate calculation for the design conditions has shown that this region vanishes if the cooling fluid distribution on the cooling holes equalises. Thus, it has been concluded that region C is also a result from the overestimated off-design flow angle. Somewhat higher temperatures for the calculation in the hub region are a result of an adiabatic wall condition for the platform of the blade.

The comparison of the results has shown that the thermal load prediction for the leading edge is very sensitive to the main flow angle. Nevertheless, it has been proven that the main phenomena of the thermal load distribution have been found in the conjugate calculation as well as in the thermal index paint measurements. Neglecting the internal ribs of the leading edge channel seems to have less effect on the main results than expected.

For the conjugate part of the calculations it is possible to determine the specific heat flux rate q from the local gradient in the thermal boundary layer:

$$q = -\lambda \left.\frac{\partial T}{\partial y}\right|_w \tag{2}$$

Figure 7 illustrates the local distribution of the specific heat flux rate for the conjugate wall of the leading edge region of the blade. It can be shown that the distribution of the specific heat flux rate is very inhomogeneous in this region. Regions with heat trans-

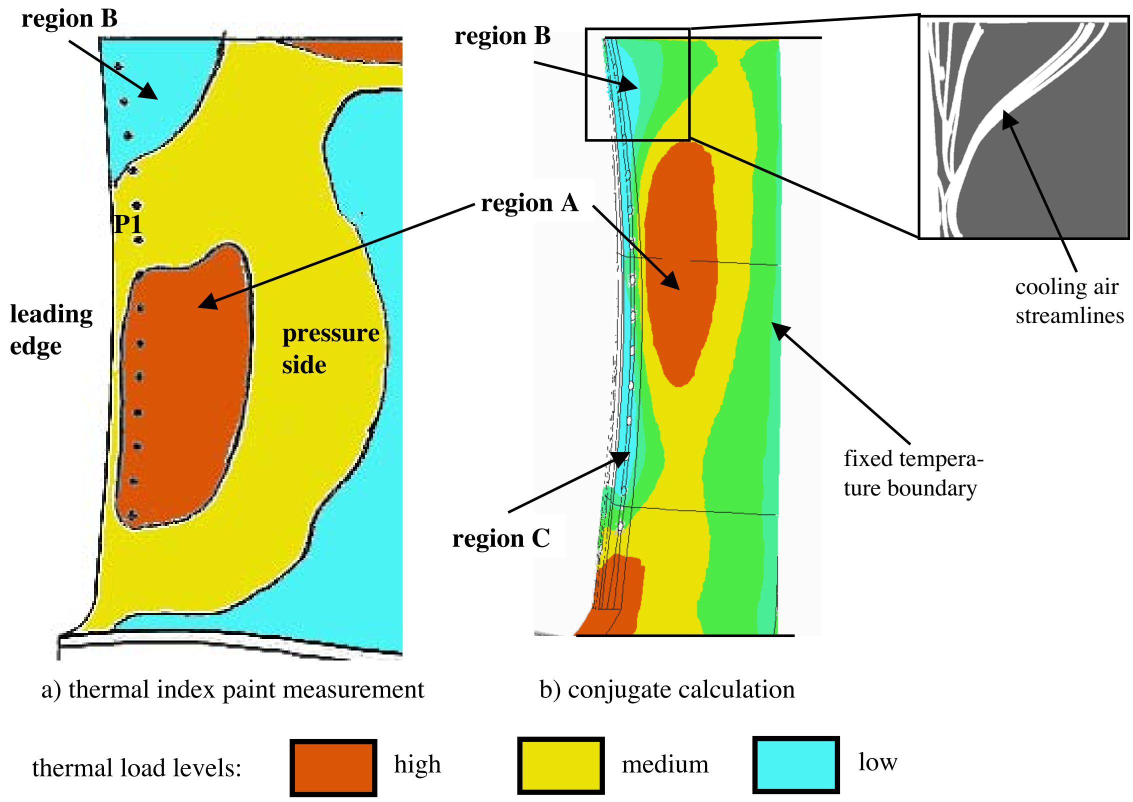

Fig. 6 Comparison of thermal index paint experiment and conjugate calculation of the leading edge region (pressure side)

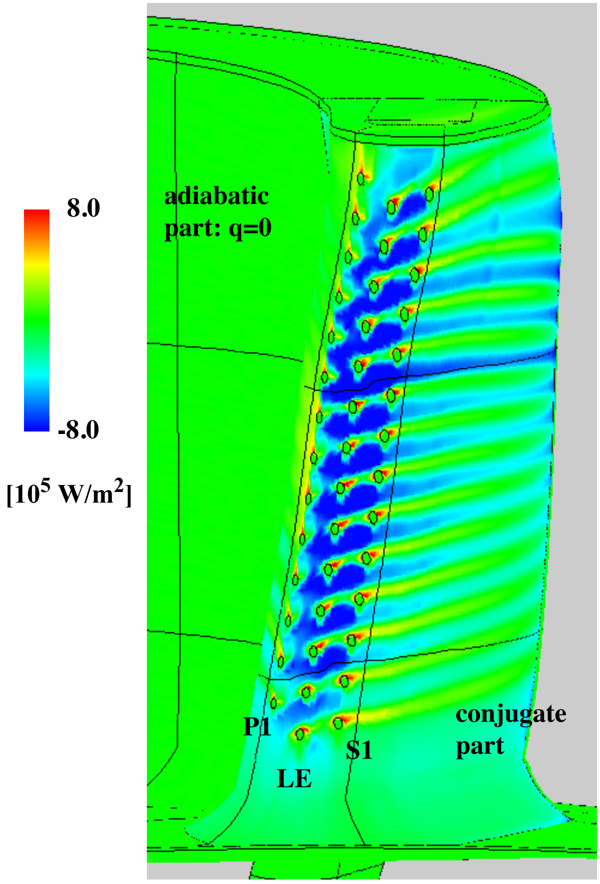

Fig. 7 Specific heat transfer distribution at the leading edge

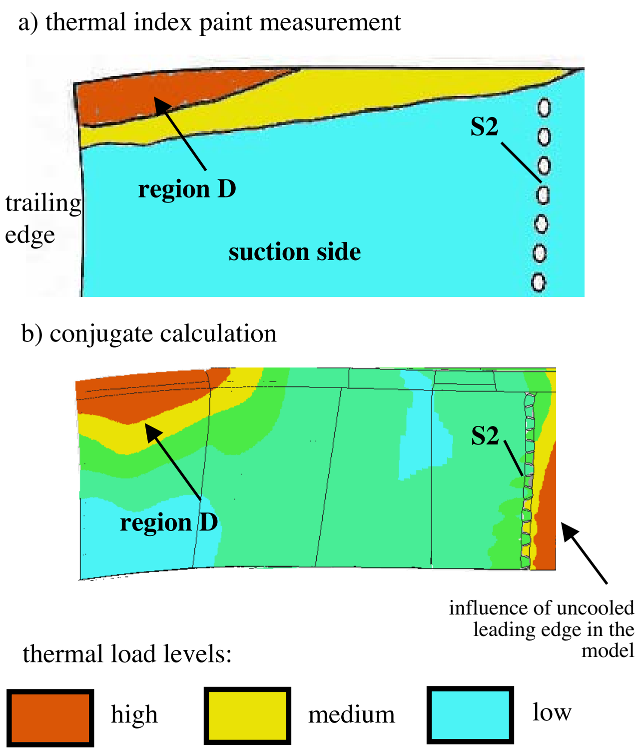

Fig. 8 Comparison of thermal index paint experiment and conjugate calculation of the upper part of the suction side

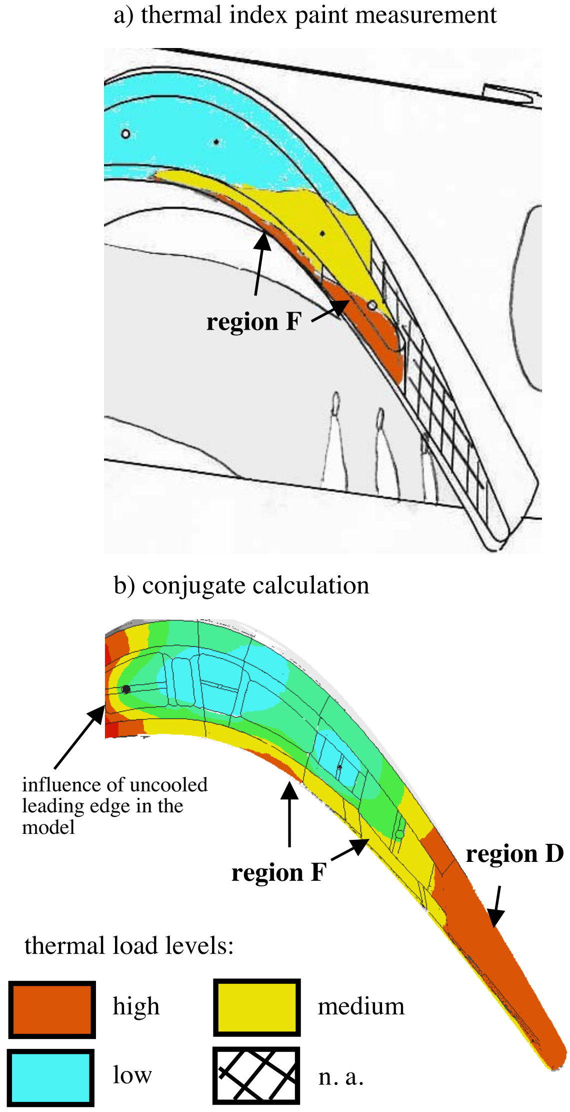

Fig. 9 Comparison of thermal index paint experiment and conjugate calculation of tip region (blade tip view)

fer from the flow into the blade wall (negative ‘"blue" values) exist directly in the stagnation area. Regions with heat transfer from the wall to the external flow (positive "red" values) are to be found downstream the cooling holes where wall contact of the cooling film is established. In particular, in the vicinity of the holes very high positive heat flux rates up to 1.6E6 W/m^2 can occur. The maximum negative values in the stagnation region are up to 1.2E6 W/m^2. As the local conjugate wall temperatures are the result of the coupled calculation of convective internal cooling, the heat conduction in the blade material and the external heat transfer, the distribution of the heat transfer rate is more realistic than calculations by conventional numerical tools based on heat transfer boundary conditions. Other investigations by Bohn et al. (2003) have shown that an improved cooling film leads to a more homogeneous distribution of the heat transfer rate

Task 2: Blade Tip Cooling

The conjugate calculation results for the trailing edge region correspond very well to the measurements. Figure 8 shows that in particular the high thermal load at the blade tip (region D) is predicted as found in the experiments. Due to the limitations of the model (neglecting the leading edge cooling) the influence of the

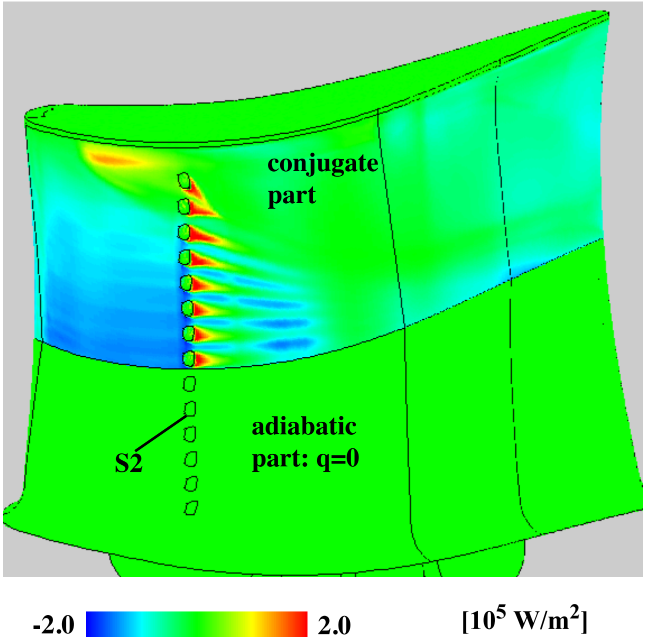

Fig. 10 Specific heat transfer distribution on the suction side

uncooled leading edge is obvious in the front part of the calculation results.

Figure 9 gives the comparison of the results for the blade tip in an onview. Additional high thermal load is found in particular in region F. The evaluation of the flow field in the simulation reveals that the origin of this phenomenon is hot gas from the pressure side that is able to stream into the radial gap and, thus, is also one reason for the high thermal load in the region D. Despite the simplifications of the model the main phenomena of the thermal load distribution have been found in the conjugate calculation in good correspondence to the measurements. Again it can be stated that neglecting the internal ribs of the passages seems to have less effect on the main results than expected.

The visualization of the specific heat transfer rate in Fig. 10 shows that for the shaped-holes on the suction side the distribution of the heat transfer becomes more homogeneous because a direct contact with the cooling fluid is established downstream the holes. Therefore, the level of the positive and negative values of the heat transfer rate is also reduced. The region with a positive heat transfer rate near the tip of the blade is established by cooling fluid ejected to the radial gap of the blade.

Based on the results of the conjugate analysis and the hot gas experiments for the test configuration, the blade cooling configuration has been improved so that even under similar off-design flow conditions a sufficient cooling of the leading edge and the blade tip is guaranteed.

CONCLUSIONS

A cooling configuration with extensive film-cooling for a test blade of a modern gas turbine has been investigated by means of numerical calculation and thermal index paint measurement under hot gas conditions. The numerical results were obtained by the Conjugate Calculation Technique and have been provided in a blind test case. The comparison of the results with respect to regions of high thermal has proven the ability of the code to detect regions of cooling deficiencies in good agreement to the experimental testing. Based on the understanding of the flow situation under design and off-design conditions, improvements of the cooling configuration with respect to the location of cooling hole rows can be obtained. Although the numerical effort is high due to the large number of necessary grid points a reasonable calculation time can be achieved with a parallel computation on LINUX PC-Clusters. Therefore, the Conjugate Calculation Techniques becomes an useful tool in the design process of modern cooling configurations.

REFERENCES

Baldwin B. S., and Lomax, H., 1978, "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," *AIAA-paper 78-257*.

Bohn. D., Ren, J., and Kusterer, K., 2003, "Conjugate Heat Transfer Analysis for Film Cooling Configurations with Different Hole Geometries," *ASME-paper GT2003-38369*.

Bohn, D., Krüger, U., and Kusterer, K., 2001, "Conjugate Heat Transfer: An Advanced Computational Method for the Cooling Design of Modern Gas Turbine Blades and Vanes," *Heat Transfer in Gas Turbines*, eds. B. Sundén and M. Faghri, pp. 58-108, WIT Press, Southampton, UK.

Bohn, D., Bonhoff, B., Schönenborn, H., and Wilhelmi, H., 1995, "Validation of a Numerical Model for the Coupled Simulation of Fluid Flow and Diabatic Walls with Application to Film-cooled Turbine Blades," *VDI-Bericht 1186*, pp. 295-272, Germany.

Bohn, D., and Bonhoff, B., 1994, "Berechnung der Kühl- und Störwirkung eines filmgekühlten transonisch durchströmten Turbinengitters mit diabaten Wänden," *VDI-Bericht 1109*, pp. 261-275, Germany.

Han, Z.-X., Dennis, B. H., and Dulikravich, G. S., 2000, "Simultaneous Prediction of External Flow-Field and Temperature in Internally Cooled 3-D Turbine Blade Material," *ASME-paper 2000-GT-253*.

Kao, K.-H., and Liou, M.-S., 1996, "On the Application of Chimera/Unstructured Hybrid Grids for the Conjugate Heat Transfer," *ASME-paper 96-GT-156*.

Li, H., and Kassab, A. J., 1994, "Numerical Prediction of Fluid Flow and Heat Transfer in Turbine Blades with Internal Cooling," *AIAA-paper 94-2933*.

Montenay, A., Paté, L., and Duboué, J. M., 2000, "Conjugate Heat Transfer Analysis of an Engine Internal Cavity," *ASME-paper 2000-GT-282*.

Okita, Y., and Yamawaki, S., 2002, "Conjugate Heat transfer Analysis of Turbine Rotor-Stator System," *ASME-paper GT-2002-30615*.

Heidmann, J. D., Kassab, A. J., Divo, E. A., Rodriguez, F., and Steinthorsson, E., 2003, "Conjugate Heat Transfer Effects on a Realistic Film-cooled Turbine Vane," *ASME-paper GT2003-38553*.

York, W. D., and Leylek, J. H., 2003, "Three-dimensional Conjugate Heat Transfer Simulation of an Internally-cooled Gas Turbine Vane," *ASME-paper GT2003-38551*.

Sugimoto, T., Nagai, K., Ryu, M., Tanaka, R., Kimura, T. and Nagatomo, T., 2002, "Development of a 20MW-Class High-Efficiency Gas Turbine L20A," *ASME-paper GT-2002-30255*.