Propulsion and Power Research 2013;2(1):1–9

http://ppr.buaa.edu.cn/

Propulsion and Power Research

www.sciencedirect.com

ORIGINAL ARTICLE

Conjugate calculation of a film-cooled blade for improvement of the leading edge cooling configuration

Norbert Moritz[a,*], Karsten Kusterer[a], Dieter Bohn[b], Takao Sugimoto[c], Ryozo Tanaka[c], Tomoki Taniguchi[c]

[a]*B&B-AGEMA GmbH, Juelicher Str. 338, Aachen 52070, Germany*
[b]*RWTH Aachen, Aachen 52056, Germany*
[c]*Kawasaki Heavy Industries, LTD., Gas Turbine & Machinery Company, Akashi 673-8666, Japan*

Received 17 April 2012; accepted 11 September 2012
Available online 1 February 2013

KEYWORDS

Gas turbine;
Rotor blade;
Film-cooling;
Conjugate heat transfer;
Conjugate calculation;
Pyrometer measurement

Abstract 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. The conjugate calculation technique is used for the three-dimensional thermal load prediction of a film-cooled test blade of a modern gas turbine. Thus, it becomes possible to take into account the interaction of internal flows, external flow, and heat transfer without the prescription of heat transfer coefficients. The focus of the investigation is laid on the leading edge part of the blade. The numerical model consists of all internal flow passages and cooling hole rows at the leading edge. Furthermore, the radial gap flow is also part of the model. The comparison with thermal pyrometer measurements shows that with respect to regions with high thermal load a qualitatively and quantitatively good agreement of the conjugate results and the measurements can be found. In particular, the region in the vicinity of the mid-span section is exposed to a higher thermal load, which requires further improvement of the cooling arrangement. Altogether the achieved results

*Corresponding author: Tel.: +49 241 5687840.
E-mail address: moritz@bub-agema.de (Norbert Moritz).

Peer review under responsibility of National Laboratory for Aeronautics and Astronautics, China.

Production and hosting by Elsevier

http://dx.doi.org/10.1016/j.jppr.2012.10.005

demonstrate that the conjugate calculation technique is applicable for reasonable prediction of three-dimensional thermal load of complex cooling configurations for blades.

1. Introduction

The design process of advanced film-cooling configurations for blades of modern gas turbines is one major task in the manufacture's strategy for further improvement of gas turbine efficiency. 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. Thus, 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. In particular, the leading edge cooling fluid ejection is of great interest in the design task. Several authors [1–3] have presented full 3-D numerical simulations of the aerothermal flow conditions including leading edge ejection. Bohn and Kusterer [4,5] have investigated the 3-D cooling jet phenomena for blade leading edge ejection from non-lateral and radially inclined cooling holes. They have shown the development of different vortex systems in the cooling jets, which influence the cooling fluid distribution in the case of leading edge ejection.

With respect to numerical heat transfer calculations based on full 3-D numerical simulations, several studies can be found in the literature [6–8]. As an example, numerical studies on the leading edge film-cooling physics by York and Leylek [9,10] focus on the determination of the adiabatic film-cooling effectiveness and heat transfer coefficients.

Conventional thermal design processes rely on empirical correlation 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. A numerical example is given by Bohn et al. [11], where it has been shown that the vortex structures in cooling jets are significantly affected by the heat transfer. A large number of conjugate calculation approaches have been published in recent years. With respect to application of conjugate calculation methods to blade film-cooling, the work of the following groups can be mentioned as examples: Heidmann et al. [12] have used the Glenn-HT code for calculation of conjugate heat transfer effects on a realistic film-cooled turbine vane. Within this code, the boundary element method (BEM) by Li and Kassab [13,14] has been implemented. The BEM does not require meshing of the solid region. As described by Rigby and Lepicovsky [15], the Glenn-HT code has also been extended to solve the conjugate heat transfer problems by gridding inside the solid and setting the velocities there to zero. Adami et al. [16] have upgraded a finite volume CFD solver by coupling it to a routine solving the Fourier equation in the solid domain. The code has been tested successfully for a transonic NGV including film-cooling of the pressure side.

Numerous applications of other conjugate heat transfer methods, including commercial codes, can be found in literature. Facchini et al. [17] and Luo and Razinsky [18] have analyzed the NASA C3X profile by application of StarCD. A nozzle guide vane of a real engine has been investigated with the same software by Mazur et al. [19]. Several application of the CFX solver to NASA Mark-II case and C3X cases are also to be found in literature [20,21]. Also more realistic or real, fully-cooled examples of conjugate heat transfer analyses with different solvers can be found (e.g. [20,22]).

Within the present paper, numerical investigations on a test configuration for the first stage turbine blade of an industrial gas turbine have been performed. The CHTflow solver [23] has been used and the calculations have been executed before the testing of the configuration started. Based on the numerical results of the initial cooling design an improvement of the leading edge film-cooling configuration has been performed. The comparison of experimental results of this design and the numerical results shows that the conjugate calculation technology implemented to the CHTflow solver is applicable to predict the thermal loading with the necessary accuracy.

2. Test configuration

Kawasaki Heavy Industries (KHI), LTD., has developed an experimental test configuration of a first stage blade of an industrial gas turbine. At the blade leading edge, the configuration consists of three rows of radially inclined cooling holes (indicated as “P1”, “LE” and “S1” in Figure 1), which are supplied with cooling air by a single channel. Furthermore, the test configuration includes two rows of shaped holes, the row “S2” on the suction side and “P2” on the pressure side. They are supplied by further internal, serpentine shaped cooling channels, indicated as “II” and “III” in Figure 1. For augmentation of the convective heat transfer, the internal walls of the passages are equipped with small square ribs. Furthermore, a large

Nomenclature

Ma	Mach number
Re	Reynolds number
T	temperature (unit: K)

Greek letters

η	cooling effectiveness
λ	thermal conductivity (unit: $W/(m \cdot K)$)

Subscripts

c	cooling fluid
h	hot gas
in	inlet
fl	fluid
o	stagnation
s	solid
w	wall

number of pin-fins are applied to the trailing edge chamber. The configuration has been analyzed experimentally under hot gas conditions by KHI at Akashi R&D Center. Measurement data of the thermal load were obtained by pyrometer experiments. The 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.

3. Numerical method and models

3.1. Conjugate heat transfer and flow solver (CHTflow)

For the numerical simulation, the conjugate calculation technique used in the CHTflow code [23] 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 code has been developed in the 1990s at the Institute of Steam and Gas Turbines at RWTH Aachen University,

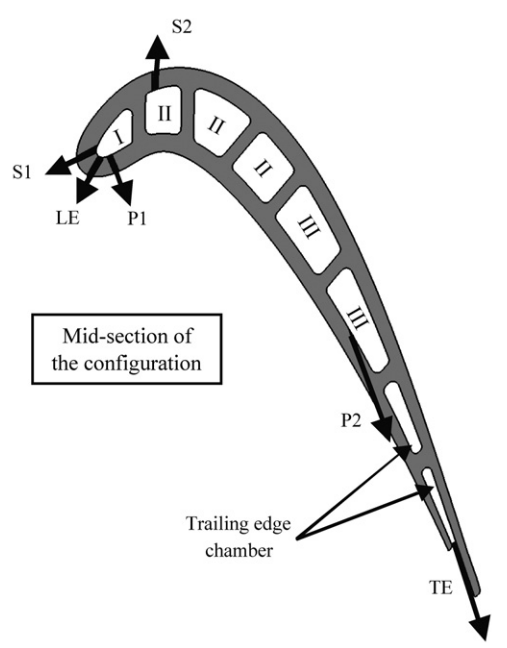

Figure 1 Film-cooled test blade.

focusing on providing a most sophisticated approach for the coupled calculation of fluid flows and heat transfer especially for hot gas components in gas turbines. The numerical group of the institute developed the homogeneous method for the conjugate calculation technique (CCT). The method involves the direct coupling of the fluid flow and the solid body using the same discretization and numerical principle for both zones. This makes it possible to have an interpolation-free crossing of the heat fluxes between the neighboring cell faces. Thus, additional information on the boundary conditions at the blade walls, such as the distribution of the heat transfer coefficient, becomes redundant, and the wall temperatures as well as the temperatures in the blade walls are a direct result of this simulation.

Basic validation of the code has been published by Bohn et al. [24,25]. 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 (RANS) is provided by the Baldwin–Lomax algebraic eddy-viscosity turbulence [26]. This turbulence model is very robust and calculations also of very complex geometry usually run very stable. Despite of its simplicity, the authors have very good experience with the application of the Baldwin–Lomax turbulence model. The solver CHTflow has been used for different film-cooling geometries, which are documented in several publications (e.g. [27,28]), and the comparison to experimental results has shown, which phenomena are predicted with reliable accuracy and which deviations are calculated, if this turbulence model is applied.

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

$$T_w = \frac{(\lambda_s/\lambda_{fl})T_s + T_{fl}}{1 + (\lambda_s/\lambda_{fl})} \tag{1}$$

This means that no heat transfer boundary conditions have to be stipulated on the solid surfaces. This method of calculating the heat fluxes requires a very high grid resolution normal to solid body surfaces. In particular, the numerical grid for the fluid flow calculation should allow an adequate resolution of the laminar sub layer. 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).

3.2. Conjugate model for film-cooling configuration

For the conjugate calculation of the leading edge region of the test blade, a 3-D numerical grid consisting of approximately 7.0 million hexahedral cells in 255 blocks has been generated for both investigated configurations. The numerical grid comprises the complete blade passage, the radial gap at blade tip with a simplification of the surface inside the squealer, all cooling holes of the three rows at the leading edge (in total 54 holes), and the leading edge supply channel. Each cooling hole is discretized with one H-type block consisting of 1024 cells (16 cells on the circumference and 16 cells along the hole axis). In radial direction, the squealer is represented by 14 cells and the radial gap with 6 cells. This discretization leads to 522 radial cells in the blade passage. All geometry data represent the hot conditions of the turbine. The ribbed walls have been modeled as smooth walls.

The two rear cooling channels (supplies II and III in Figure 2) have been removed in the numerical model and the solid region is meshed between the leading edge supply channel (supply I) and the blade passage and in the vicinity of the supply channel in downstream direction. Figure 3 shows the meshed region in a cutting plane close to mid-span of the blade passage.

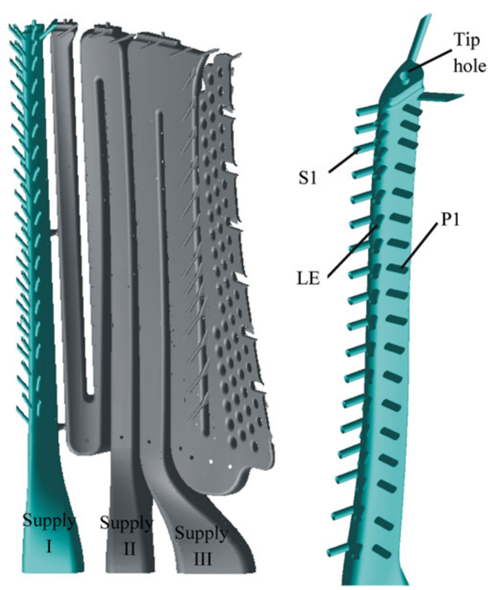

Figure 2 Internal cooling system of test configuration.

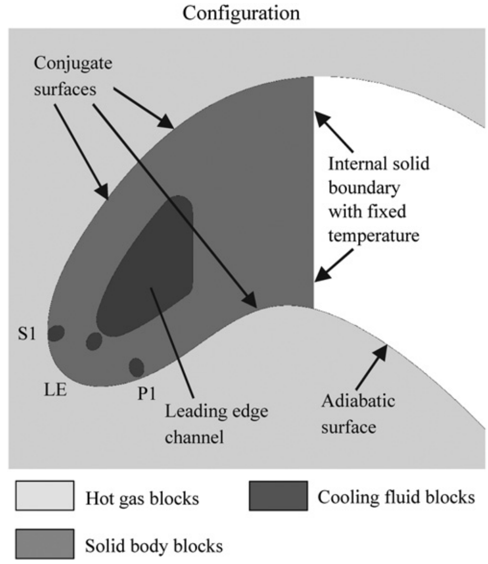

Figure 3 Illustration of the blade leading edge model.

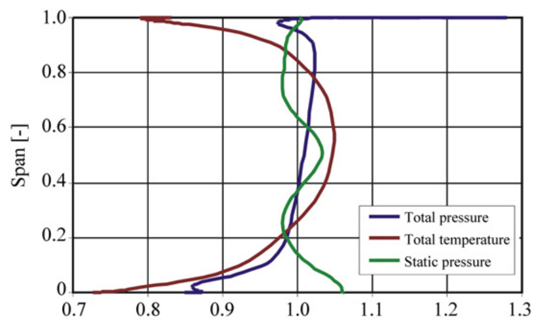

Figure 4 Radial distribution of non-dimensional relative total pressure and temperature at inlet and static pressure at outlet.

3.3. Boundary conditions

The boundary conditions for the blade passage have been derived from 2-D streamline curvature calculations. At the inlet, the relative total pressure, the relative total temperature and the relative swirl angle have been prescribed. The boundary condition for the outlet is the static pressure. The radial distributions of the non-dimensional values are shown in Figure 4. The mass averaged relative Mach number at the inlet is $Ma_{in} = 0.3779$. The inlet Reynolds number, calculated with the channel height and the mass averaged axial velocity, is $Re_{in} = 581{,}902$. The ratio of the mean inlet relative total pressure at passage inlet and supply channel I is 1.27, the ratio of the relative total temperatures is 0.54.

4. Results

4.1. Leading edge flow structures

From the conjugate calculation results, the cooling effectiveness distribution and velocity distribution have

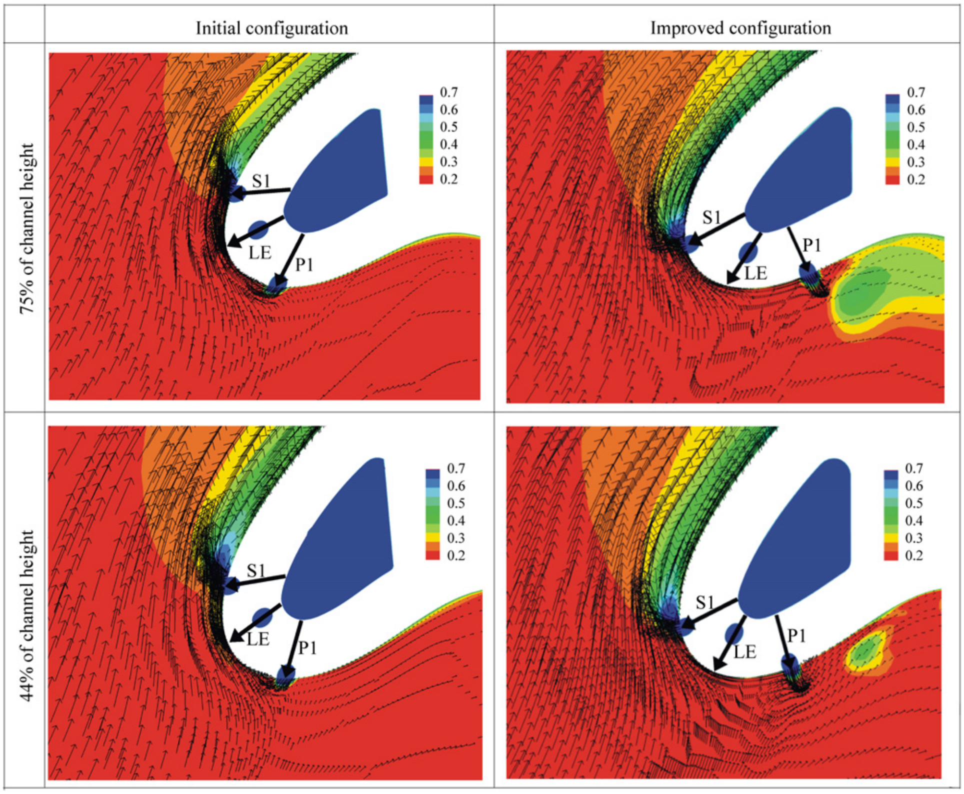

Figure 5 Comparison of cooling effectiveness distribution and flow vectors.

been extracted as an aero-thermal result. Within this scope, the cooling effectiveness η is calculated with stagnation quantities for the hot gas and cooling fluid temperature. Thus, it is defined as

$$\eta = \frac{T_{o,h} - T}{T_{o,h} - T_{o,c}} \tag{2}$$

with T as the local (static) temperature, $T_{o,h}$ as the mass averaged stagnation temperature of the main flow in front of the blade, and $T_{o,c}$ as stagnation temperature of the cooling fluid.

The flow structures in the vicinity of the leading are analyzed in sections of constant radius at positions of 44% and 75% of the blade passage channel height. In Figure 5, the cooling effectiveness distribution and the flow vectors in these sections are shown for the initial (left-hand side) and the improved (right-hand side) leading edge cooling configuration. The large arrows named "P1", "LE" and "S1" indicate the positions and directions of the radially inclined cooling holes, which are intersected by the cutting planes.

Regarding the initial configuration, it can be seen that the positioning especially of the rows "LE" and "P1" has not been fully correct with respect to the stagnation point at the leading. Due to the high turning of the flow, the stagnation point has been moved further to the pressure side than it has been estimated during the design process. The result of this mismatching of cooling hole positions and stagnation point positions is, that nearly the entire cooling air leaving from row "P1" is distributed to the suction side. Thus, in the vicinity of the leading edge the suction side surface has been cooled extensively due to the cooling jets from all three holes protecting the blade surface from hot gas contact, whereas the pressure side lacks of cooling.

On basis of these numerical results, the positions and directions of the cooling holes have been rearranged, so that for the improved configuration the rows "LE" and "P1" are located on both sides of the stagnation point. However, it also can be seen that the rearrangement of the row positions leads to a change of the stagnation point positions. Thus, during the design of leading edge film-cooling configurations the interaction of cooling flow and main flow has to be taken into account to find the correct positions of cooling hole rows. The positions and directions of the holes of row "S1" have also been changed, so that the cooling air leaving this row and the row "LE" should interact to protect the blade surface from hot gas contact.

4.2. Leading edge cooling for improved design

The results of the improved leading edge cooling design are shown in Figure 6 at six sections of constant radii at 10%, 26%, 42%, 58%, 74% and 90% of the channel height. It can be seen that in the sections close to mid-span (42% and 58%), which is the region with the highest temperatures in the main flow; the material is not cooled as efficient as in the other sections. This

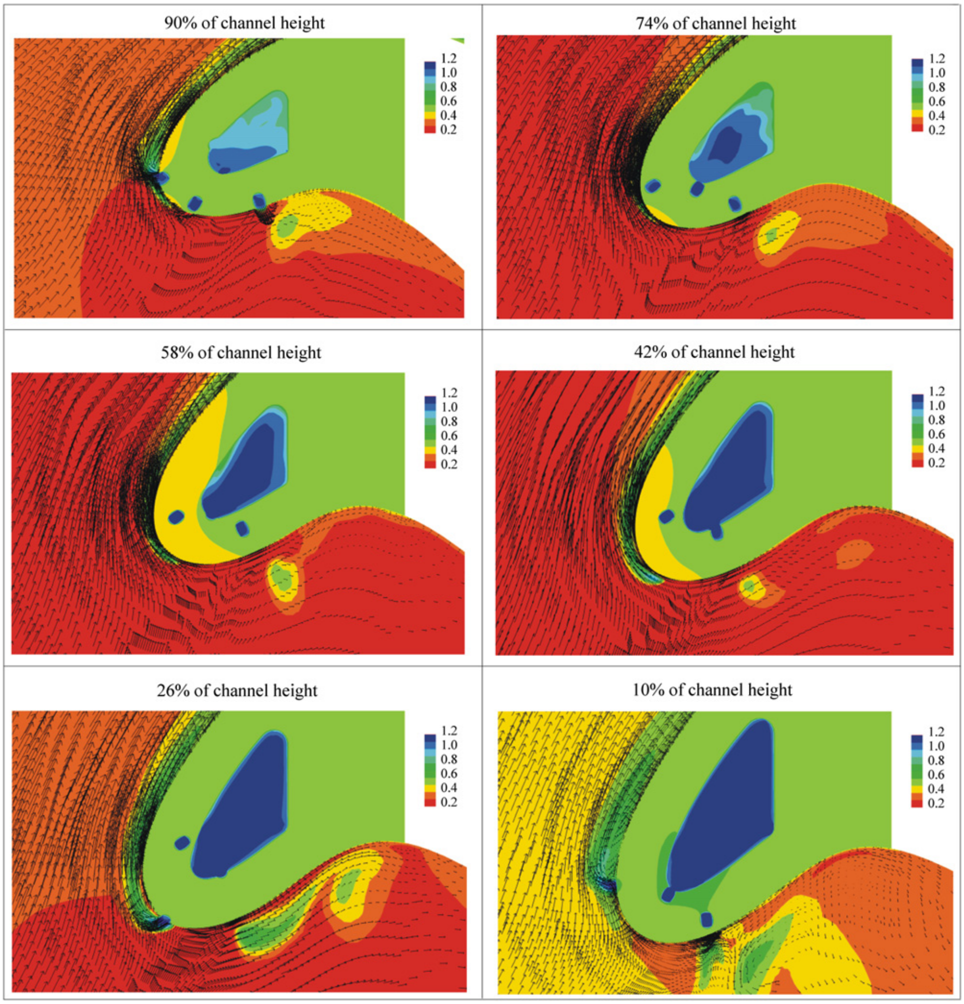

Figure 6 Cooling effectiveness distribution in fluid and solid and flow vectors for improved cooling configuration.

result is also to be found in the experimental results (see Section 4.4). Although the temperatures do not exceed the maximum allowable material temperature, a reduction of the temperature gradient would be preferred. Thus, an improvement of the cooling design in this region would be advantageous with respect to the live time of the blade.

4.3. *Interaction of cooling jets for improved design*

For the improved leading edge cooling configuration, the interaction of the cooling jets of the rows "LE" and "S1" is illustrated for the region at blade tip in Figure 7. The mauve colored streamlines indicate the cooling air from row "LE" and the cyan colored streamlines indicate the cooling air from row "S1". The blade surface in the region of the conjugate wall is colored with the cooling effectiveness. It can be seen clearly that the cooling jets from row "LE" cover the blade surface in between the gaps of the hole exits from row "S1". Thus, an effective protection of the blade surface from hot gas contact is achieved.

Moreover, it can be seen that the cooling air leaving the most top cooling hole from row "LE" is dragged away from the blade surface. This effect is due to the interaction with the tip clearance flow and the vortex generated by this flow. With respect to the film-cooling performance, it could be considered to move this hole to a lower position or to remove it from row "LE". However, the cooling effectiveness distribution at the blade tip indicates that the convective cooling of this hole is important to reduce the material temperatures in this area, so that removing this hole should cause a high thermal load at the leading edge tip.

4.4. *Comparison of numerical and experimental results*

Figure 8 shows the surface cooling effectiveness distribution of the conjugate calculation in comparison to the experimental results for the improved leading edge cooling configuration. In general, a qualitatively and quantitatively good agreement of the numerical and the experimental results can be seen.

Quantitatively, the conjugate calculation results show slightly higher temperatures than the experimental results. The major reason for this deviation is the temperature boundary condition, which is applied to the solid body inside the blade. The mean temperature in the solid body and the level of the blade surface temperature is surely affected by the chosen value. Moreover, the neglecting of the ribs inside the supply channel and the modeling of the smoothed walls has an effect on the outer surface temperature distribution. In comparison to ribbed walls, the smoothed walls show a lower heat transfer potential. Therefore, the amount of heat, which is transferred into to the leading edge support channel, is lower for the conjugate calculation, and, thus, the surface temperatures are increased in comparison to the test conditions.

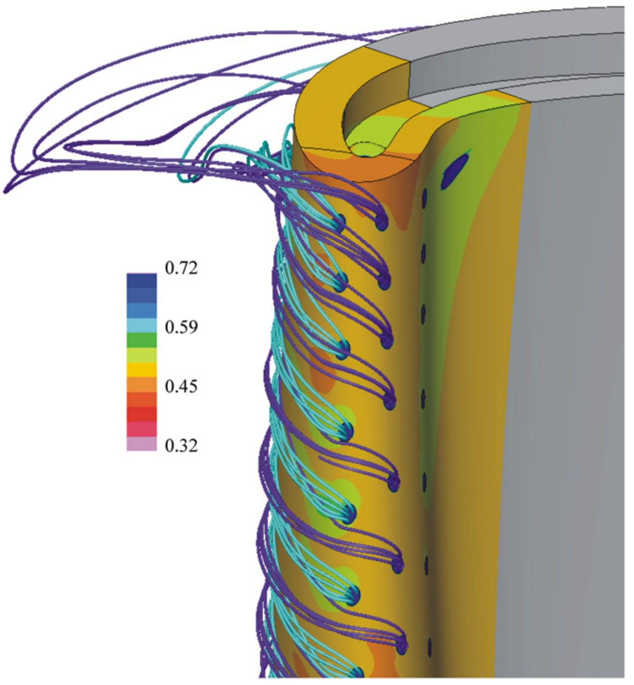

Figure 7 Cooling effectiveness distribution and streamlines of rows "LE" and "S1" for the improved cooling configuration.

With respect to qualitative comparison, it can be seen that the radial position of the higher temperature region in the vicinity of the blade midsection is similar for the conjugate calculation and the experimental results. However, this region is located around row "S1" in numerical results, whereas the measurements indicate this region between row "LE" and "P1". Thus, compared to experiments the numerical results show a shift of the higher temperature region towards the suction side. This deviation should be due to the difficulty to precisely predict the swirl angle behind the first vane of a gas turbine and, thus, the chosen inlet boundary condition might deviate from the test engine conditions. In the test facility the flow conditions in front of the first stage blade cannot be analyzed at different radial positions, so that a validation of the inflow boundary conditions is not possible.

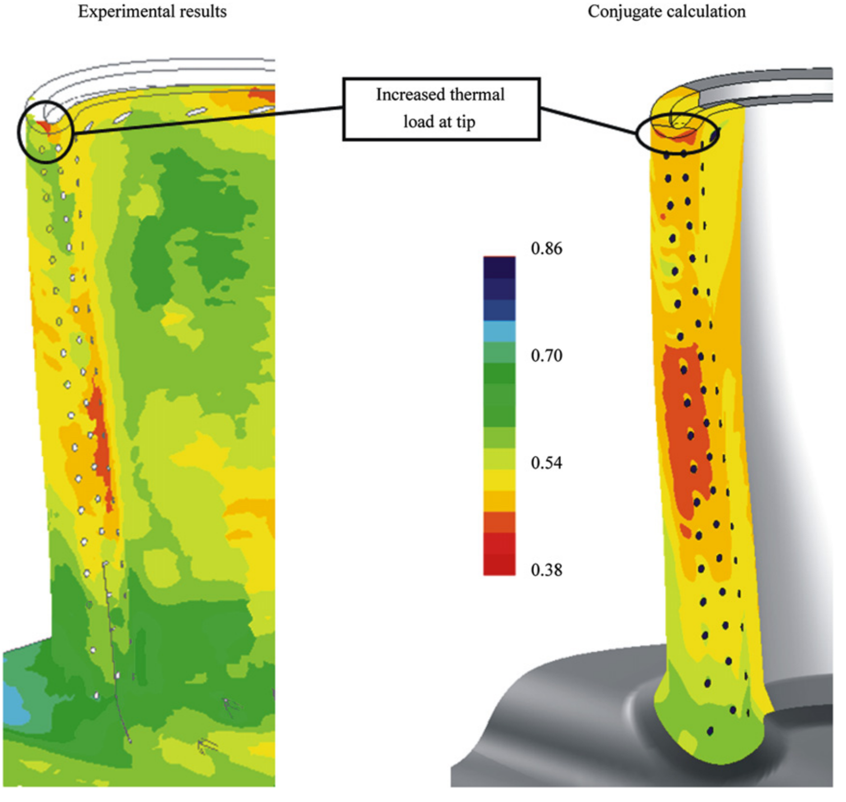

Figure 8 Surface cooling effectiveness distribution-comparison of experimental and numerical results.

Regarding the cooling effectiveness distribution at the blade tip, it can be seen that the experimental results and the conjugate calculation show a rise of thermal load. This region usually is one of the critical point of a cooled turbine blade. The radial gap flow increases the heat transfer at the tip, the deceleration of the flow near stagnation point increases the static temperatures of the hot gas and, additionally, there is some distance to the next cooling hole, which leads in combination to a challenging cooling situation. It becomes clear, that due to its convective cooling effect the tip hole of row "LE" cannot be removed, although the cooling air leaving this hole is detached from the suction side surface due to the interaction with the tip clearance flow and vortex. It could be even advantageous to move it closer to the blade tip.

5. Conclusions

A leading edge film-cooling configuration for a test blade of a modern gas turbine has been investigated by means of numerical conjugate calculation and pyrometer measurement under hot gas conditions. During the numerical investigation, a mismatching of the stagnation point location and the positioning of the leading edge cooling holes has been observed. Based on these results, an improvement of the hole positions has been performed. This improved configuration has been used for the experiments.

The comparison of the conjugate calculation results and the experimental results shows in general a good agreement, so that it can be stated that the numerical approach, which has been applied in this analysis, is useful for the investigation and improvement of leading edge cooling configurations. Especially, the radial distribution of thermally high loaded regions is in good agreement. The deviations in the thermal load are mainly due to uncertainties in the inflow boundary conditions and the application of a fixed temperature boundary condition inside the blade. The later has been used, due to the removing of the rear cooling channels.

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 and that based on such analyses an improvement of cooling configuration is possible.

References

[1] D. Bohn, V. Becker, K. Kusterer, L. Fottner, S. Ardey, Three-dimensional flow analysis of a turbine blade cascade with leading-edge ejection, AIAA Journal of Propulsion and Power 16 (1) (2000) 49–56.

[2] J.D. Heidmann, D.L. Rigby, A.A. Ameri, A 3-D coupled internal/external simulation of a film-cooled gas turbine, ASME Paper 99-GT-186, 1999.

[3] C.A. Martin, K.A. Thole, A CFD benchmark study: leading edge film-cooling with compound angle injection, ASME Paper 97-GT-297, 1997.

[4] D. Bohn, K. Kusterer, Blowing ratio influence on jet mixing flow phenomena at the leading edge, AIAA Paper 99-0670, 1999.

[5] D. Bohn, K. Kusterer, Aerothermal investigations of mixing flow phenomena in case of radially inclined ejection holes at the leading edge, ASME Journal of Turbomachinery 122 (2) (2000) 334–339.

[6] E.J. Hall, D.A. Topp, R.A. Delaney, Aerodynamic/heat transfer analysis of discrete site film-cooled turbine airfoils, AIAA Paper 94-3070, 1994.

[7] V.K. Garg, Heat transfer on a film-cooled rotating blade, ASME Paper 99-GT-44, 1999.

[8] V.K. Garg, D.L. Rigby, Heat transfer on a film-cooled blade-effect of hole physics, International Journal of Heat and Fluid Flow 20 (1) (1999) 10–25.

[9] W.D. York, J.H. Leylek, Leading edge film-cooling physics: Part I – adiabatic effectiveness, ASME Paper GT2002-30166, 2002.

[10] W.D. York, J.H. Leylek, Leading edge film-cooling physics: Part II – heat transfer coefficient, ASME Paper GT2002-30167, 2002.

[11] D. Bohn, J. Ren, K. Kusterer, Conjugate heat transfer analysis for film cooling configurations with different hole geometries, ASME Paper GT2003-38369, 2003.

[12] J.D. Heidmann, A.J. Kassab, E.A. Divo, F. Rodriguez, E. Steinthorsson, Conjugate heat transfer effects on a realistic film-cooled turbine vane, ASME Paper GT2003-38553, 2003.

[13] H.J. Li, A.J. Kassab, Numerical prediction of fluid flow and heat transfer in turbine blades with internal cooling, AIAA Paper 94-2933, 1994.

[14] H.J. Li, A.J. Kassab, A coupled FVM/BEM approach to conjugate heat transfer in turbine blades, AIAA Paper 94-1981, 1994.

[15] D.L. Rigby, J. Lepicovsky, Conjugate heat transfer analysis of internally cooled configurations, ASME Paper 2001-GT405, 2001.

[16] P. Adami, F. Martelli, F. Montomoli, A finite volume method for the conjugate heat transfer in film cooling devices, in: Proceedings of the XVI International Symposium on Air Breathing Engines, Paper No. ISABE-2003-1066, 2003.

[17] B. Facchini, A. Magi, A.S.D. Greco, Conjugate heat transfer simulation of a radially cooled gas turbine vane, ASME Paper GT2004-54213, 2004.

[18] J. Luo, E.H. Razinsky, Conjugate heat transfer analysis of a cooled turbine vane using the V2F turbulence model, ASME Paper GT2006-91109, 2006.

[19] Z. Mazur, A. Hernandez-Rossette, R. Garcia-Illescas, A. Luna-Ramirez, Analysis of conjugate heat transfer of a gas turbine first stage nozzle, ASME Paper GT2005-68004, 2005.

[20] G. Nowak, W. Wroblewski, Application of conjugate heat transfer for cooling optimization of a turbine airfoil, ASME Paper GT2009-59818, 2009.

[21] Z.F. Wang, P.G. Yan, H.Y. Huang, W.J. Han, Conjugate heat transfer analysis of a high pressure air-cooled gas turbine vane, ASME Paper GT2010-23247, 2010.

[22] A. Sipatov, L. Gomzikov, V. Latyshev, N. Gladysheva, Three-dimensional heat transfer analysis of high pressure turbine blade, ASME Paper GT2009-59163, 2009.

[23] K. Kusterer, U. Krüger, D. Bohn, Conjugate heat transfer: an advanced computational method for the cooling design of modern gas turbine blades and vanes, in: B. Sunden, M. Faghri (Eds.), Gas Turbine Heat Transfer, WIT Press, Southampton, UK, 2001, pp. 57–108.

[24] D. Bohn, B. Bonhoff, Berechnung der Kühl- und Störwirkung eines filmgekühlten transsonisch durchströmten Turbinengitters mit diabaten Wänden, VDI-Berichte 1109, Germany, 1994, pp. 261–275.

[25] D. Bohn, B. Bonhoff, H. Schönenborn, H. Wilhelmi, Validation of a numerical model for the coupled simulation of fluid flow and diabatic walls with application to film-cooled turbine blades, VDI-Berichte 1186, Germany, 1995, pp. 259–272.

[26] B.S. Baldwin, H. Lomax, Thin layer approximation and algebraic model for separated turbulent flows, AIAA Paper 78-257, 1978.

[27] D. Bohn, V. Becker, K. Kusterer, S. Ardey, L. Fottner, The influence of slot ejection and showerhead ejection on the 3-D flow field of a film-cooled turbine blade under consideration of side wall effects, in: Proceedings of the 13th International Symposium on Air Breathing Engines, Chattanooga, USA, 7–12 September 1997, pp. 1203–1213.

[28] K. Kusterer, T. Hagedorn, D. Bohn, T. Sugimoto, R. Tanaka, Improvement of a film-cooled blade by application of the conjugate calculation technique, ASME Paper GT2005-68555, 2005.