Turbulent and conjugate heat transfer simulation for gas turbine application

D. Bohn\textsuperscript{1} & K. Kusterer\textsuperscript{2}

\textsuperscript{1}Institute of Steam and Gas Turbines, Aachen University, Templergraben 55, D-52056 Aachen, Germany
\textsuperscript{2}B&B-AGEMA GmbH, Jülicher Str. 338, D-52070 Aachen, Germany

Abstract

Turbulent heat transfer is of significant importance in gas turbine design and applications. High turbine inlet temperatures in modern gas turbines exceed by far the permissible metal temperatures. In particular, the vanes and blades of turbine front stages have to be cooled sufficiently for a safe operation. Internal convective cooling and external film cooling are required. 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.

Most of the flow in a gas turbine is turbulent. The flow is characterized by fluctuations and random variations in space and time. Furthermore, the convective heat transfer is significantly increased in turbulent flows. Conventional design processes rely on empirical correlations for turbulent 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.

An advanced numerical approach predicts the thermal load based on a coupled calculation of the fluid flow and the heat transfer (conjugate calculation technique, CCT) avoiding the heat transfer correlations. It can be shown that the technology is applicable for the simulation of modern gas turbine systems.
Nomenclature

\[ A \text{ area } [m^2] \]
\[ D, d \text{ diameter } [m] \]
\[ H, h \text{ height } [m] \]
\[ L, l \text{ length } [m] \]
\[ P \text{ power } [W] \]
\[ Q \text{ heat flux rate } [W] \]
\[ R \text{ specific gas constant } [J/(kg \cdot K)] \]
\[ S \text{ circumferential length } [m] \]
\[ T \text{ temperature } [K] \]
\[ T_B \text{ combustion temperature } [K] \]
\[ T_F \text{ firing temperature } [K] \]
\[ V \text{ secondary flow velocity } [m/s] \]
\[ a \text{ velocity of sound } [m/s] \]
\[ \dot{c} \text{ velocity } [m/s] \]
\[ c_p \text{ cooling potential } [J/(s \cdot K)] \]
\[ c_P \text{ specific heat at constant pressure } [J/(kg \cdot K)] \]
\[ h \text{ heat transfer coefficient } [W/(m^2 \cdot K)] \]
\[ m \text{ mass flow rate } [kg/s] \]
\[ n \text{ number of revolutions } [1/s] \]
\[ p \text{ pressure } [N/m^2] \]
\[ q \text{ specific heat } [W/m^2] \]
\[ r \text{ recovery factor } [-] \]
\[ s \text{ coordinate along vane surface } [m] \]
\[ s \text{ entropy } [J/(kg \cdot K)] \]
\[ x, y, z \text{ Cartesian coordinate } [m] \]
\[ y^+ \text{ dimensionless wall coordinate } [-] \]

Greek letters
\[ \Delta \text{ difference } [-] \]
\[ \vartheta \text{ temperature } [{^\circ}C] \]
\[ \alpha, \beta \text{ angles of hole geometry } [^\circ] \]
\[ \varepsilon \text{ numerical error } [-] \]
\[ \eta \text{ efficiency, effectiveness } [-] \]
\[ \lambda \text{ conductivity } [W/(m \cdot K)] \]
\[ \mu \text{ dynamic viscosity } [m^2/s] \]
\[ \rho \text{ density } [kg/m^3] \]
\[ \tau_{ij} \text{ components of stress tensor } [kg/m^2] \]
\[ \kappa \text{ isentropic exponent } [-] \]

Subscripts
\[ C \text{ compressor } \]
G, g hot gas
H centre housing
OC oil channel in center housing
PS pressure side
R thermal radiation
S steam
SB solid body
SS suction side
T turbine
W, w wall
a adiabatic
abs absolute
c cooling, cooling gas condition
electric
expe experiment
f film cooling
iso isoline
max maximum
min minimum
nC natural convection
o stagnation
r recovery
ref reference
rel relative
th thermal
ζ partial differentiation in the arbitrary coordinate
1 at inlet
2 at outlet

Abbreviations
CCT conjugate calculation technique
LE leading edge
OP operating point
PS pressure side
SS suction side

Non-dimensional characteristic numbers and parameters
\[ A = \frac{(m_c c_p)}{(h S l)} \] heat transfer capacity
\[ M = \frac{(\rho c)_1}{(\rho c)_c} \] blowing ratio
\[ Ma = \frac{c}{a} \] Mach number
\[ Nu = \frac{(h l)}{\lambda} \] Nusselt number
\[ Pr = \frac{(\mu c_p)}{\lambda} \] Prandtl number
\[ Re = \frac{(\rho c l)}{\mu} \] Reynolds number
\[ \theta = \frac{(T_r-T_{\infty})}{(T_r-T_{wb})} \] non-dimensional temperature
1 Introduction

1.1 Heat transfer in gas turbine systems

Turbulent heat transfer is of significant importance in the design of modern gas turbine systems. Further numerical and experimental investigations are important for understanding and improving the heat transfer processes as well as for developing advanced design tools. In particular, the need of cooling technologies in gas turbines requires a good understanding of heat transfer processes in internal and external flows of the cooled components. Furthermore, the importance of heat exchangers to improve the thermal efficiency and performance of modern gas turbines is closely related to the understanding of turbulent heat transfer processes. A good overview on heat transfer issues in gas turbine systems is given by Sundén and Faghri [1]. Heat exchanger design and manufacturing requirements are discussed in detail by Utriainen and Sundén [2].

Due to temperature differences between the hot and cold parts of a gas turbine system undesired heat transfer by heat conduction through inner and outer casings occurs leading to unexpected influences in other components of the system. As an example, the prediction of the temperature distribution in a gas turbine rotor containing closed, gas-filled cavities, for example in between two discs, has to account for the heat transfer conditions encountered in these cavities [3, 4]. In an entirely closed annulus forced convection is not present, but a strong natural convection flow exists. Investigations of turbulence effects on flows in rotating cavities by application of numerical simulation have been presented by several authors (e.g. Bohn and Gier [5, 6], Iacovides et al. [7]). Another good example of undesired influences of heat transfer from the hot turbine parts to cooler regions of the system is a turbocharger. Despite an insulating material layer with low thermal conduction properties, there is still a countable heat flux from the hot turbine to the compressor leading to negative effects on the aerodynamic performance of the compressor. Therefore, improving turbocharger efficiency requires design processes taking into account the results of a complete heat transfer analysis of the turbocharger system [8].

Within this chapter the focus is on heat transfer design aspects of conventional design methods and the advanced conjugate calculation technology for gas turbine cooling technologies. High cycle efficiencies and high power-weight ratios are two major requirements for the economic operation of both stationary gas turbines and aero engines. This development leads to extremely high turbine inlet temperatures and adjusted pressure ratios. The allowable hot gas temperature is limited by the materials used for the components. Intensive cooling is required to guarantee an economically acceptable life span of the components (e.g. combustor walls, turbine vanes and blades, disks, etc.) that are in contact with the hot gas. At present, the most efficient cooling concept is a combination of internal cooling technologies, i.e. internal forced convection cooling and impingement cooling, and external cooling technologies like film
cooling or effusion cooling. Figure 1 shows that the cooling effectiveness of film cooling and effusion cooling is distinctly higher than that of internal cooling under the same conditions. In other words, less cooling fluid can be used to obtain the required cooling of a component part.

With respect to the internal cooling technologies, Section 3 discusses the basic aspects of convective internal cooling and presents validation results achieved by turbulent and conjugate heat transfer calculations for the internal cooling of air-cooled nozzle guide vanes. Furthermore, results for a steam-cooled vane are presented showing some difficulties of this technology for sufficient internal cooling of the trailing edge region. With respect to the external cooling, basic design aspects for film cooling and the influence of conjugate heat transfer on the cooling jets are discussed in Section 4. Furthermore, the successful application of the conjugate calculation technology on real blade cooling – including internal and external cooling of the blade – is demonstrated. The numerical results are compared to thermal index paint measurements obtained under hot gas conditions. Finally, Section 5 presents a complete system heat transfer analysis for a turbocharger.
1.2 Conventional design strategy of cooled turbine blades

Due to the necessity of cooling technologies in modern gas turbines, turbulent heat transfer is of significant importance in the thermal design process of the cooled components. Tremendous efforts have been put into the determination of empirical correlations for the internal and external heat transfer, which are necessary for the conventional design process. Here, analysis of turbine blade cooling and heat transfer consists of three areas [9]: (a) prediction of the heat transfer coefficients on the external surface of the airfoil [10], (b) prediction of heat transfer in the internal cooling passages [10] and (c) calculation of the temperature distribution in the blade material.

For the internal flow calculation, 1-D flow solvers are often used for modelling the cooling passages and the heat transfer is determined on the correlations for different duct flows. The 1-D solver is coupled to an FEM-solver for the temperature calculation in various radial sections of the blade. Methods of predicting external turbine blade heat transfer often include empirical correlations for a standard flow situation, such as flow around a cylinder for modelling the blade leading edge or flow along a flat plate for modelling external heat transfer at the suction side and pressure side. Additional augmentation factors and correction factors are used based on the experience of the manufacturer for the considered airfoil. Hence, in the current development of the manufacturer’s technical knowledge in this field of complex cooling configuration, the field data of the actual engines and the consideration of the latest heat transfer data play a significant role [11]. The solution of the FEM-solver provides the wall temperatures for the calculation of the quantitative heat transfer in the next design step. In the case of additional film cooling, the cooling fluid properties and mass flows at the hole exits are provided by the 1-D solver, too. For the external heat transfer, corrections of the heat transfer due to the influence of the film cooling on the external flow field are also provided on the basis of correlations for the heat transfer coefficients [12]. Furthermore, the adiabatic cooling efficiency is of significant importance for the determination of the reference temperature in the heat transfer calculation.

The internal view of an intensive cooled turbine blade reveals the complex cooling channel system with serpentine-shaped cooling passages, which is necessary for the intensive internal cooling of the blade. For heat transfer enhancement a large number of ribs, turbulators and pin fins are to be found. In the design of the internal cooling system, the determination of reliable heat transfer correlations is very difficult and requires large efforts. A large number of experimental investigations is to be found in the literature on this topic (e.g. Mochizuki et al. [13], Schabacker et al. [14], Abuaf and Kercher [15], Chanteloup et al. [16], Casarsa et al. [17], Taslim [18, 19], etc.). Numerical studies have also been performed by several authors concerning this subject (e.g. Bonhoff et al. [20], Moore and Moore [21], Murata and Mochizuki [22], Prakash and Zerkle [23], Rigby [24], Shih et al. [25], Wang and Chyu [26], Bohn et al.
An extensive survey of the literature on enhanced internal cooling of gas turbine airfoils is given by Lau [28]. Basic experimental and numerical studies for external heat transfer determination of convection-cooled turbine vanes have been presented by several working groups (e.g. Hylton et al. [29], Wittig et al. [30], Arts and De Rouvroit [31], Ames [32]). Basic knowledge of film cooling technologies for modern gas turbines can be derived from the investigations by Goldstein [33], Goldstein et al. [34] and Ito et al. [35], for example, carried out in the 1970s. As regards the CFD research in this area, a bibliography (1971–1996) of the most important publications during this time can be found in a study by Kercher [36]. A good survey of modern cooling technologies and heat transfer design aspects for gas turbines is given by Han et al. [37].

2 Conjugate calculation technique

2.1 Basic principle of a conjugate calculation

With regard to the inter-relations between the external fluid flow, the internal fluid flow and the heat conduction, it is obvious that a coupled calculation of the fluid, the heat transfer and the heat conductivity in the solid body (Sundén [38]) can lead to a higher accuracy in the design process. This leads to the development of hybrid coupling processes, with which the fluid flow and the structure heat conduction are simulated with different numerical methods (e.g. Heselhaus et al. [39]). The disadvantages of such a hybrid process are the problems associated with numerically dealing with borders between different calculation areas, which are discretised using completely different numerical methods. The development of a stable coupling algorithm must therefore be prepared with great care and caution [40]. Often, however, numerical instabilities occur due to the required interpolation prescriptions.

Similar experiences have been perceived by Imlay et al. [41] when using a coupling procedure that transfers heat flux from the Navier–Stokes module to the heat conduction module and wall temperature from the heat conduction module to the fluid solver. More stability of the procedure is reached by transferring a heat conduction coefficient and a reference temperature from the Navier–Stokes module to the heat conduction module. Another example of a hybrid conjugate method is given by Li and Kassab [42, 43] using the boundary element method (BEM) for the solid body and a finite-volume method for the fluid flow. Recently, the BEM has been implemented to the Glenn-HT code for a film cooling application [44].

Based on the above considerations, a coupling process was developed during the early 1990s by Bohn et al. [45 – 48] at the Institute of Steam and Gas Turbines, Aachen University, which follows a homogeneous method, and that we call the conjugate calculation technique (CCT). The method involves the direct coupling of the fluid flow and the solid body using the same discretisation and
numerical principle for both zones. This makes it possible to have an interpolation-free crossing of the heat fluxes between the neighbouring 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.

In the case of film cooling, the conjugate strategy allows taking into account the heating up of the cooling fluid by convective heat transfer in the internal blade passages and, thus, the influence on the external cooling performance is also part of the calculation. Figure 2 shows the principle of the conjugate calculation technique. An extensive description of the methodology and its validation is given in [49]. Different conjugate calculation approaches have been presented also by other authors in recent years and have been applied to gas turbine cooling (e.g. Kao and Liou [50], Han et al. [51], Montenay et al. [52], Takahashi et al. [53], Rigby and Lepicovsky [54], York and Leylek [55]).

2.2 Mathematical model for the coupling procedure

The coupling of fluid blocks and solid body blocks as shown in Fig. 3 is achieved via a common wall temperature resulting from the equality of the local heat fluxes passing the contacting cell faces:

Figure 2: Motivation and principle of a conjugate calculation.
This means that the discrete heat flux of each wall boundary cell can be determined. The equality of the local heat fluxes leads to a wall temperature:

\[
T_w = \frac{\lambda_{SB} T_{SB} - T_{Fluid}}{1 + \lambda_{SB} \lambda_{Fluid}}.
\]

This wall temperature serves as a boundary condition between the fluid blocks and the solid-body blocks. It is obvious that 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, which means that at least two grid lines should be arranged within the region of \( y^+ < 3 \). A greater number of grid lines would, of course, be even more suitable for the calculation. Due to the high

\[
q_{Fluid} = q_{SB}, \quad (1)
\]

with

\[
q_{Fluid} = -\lambda_{Fluid} \frac{T_{Fluid} - T_w}{0.5 \cdot \sqrt{x_s^2 + y_s^2 + z_s^2}} \quad (2)
\]

\[
q_{SB} = -\lambda_{SB} \frac{T_w - T_{SB}}{0.5 \cdot \sqrt{x_s^2 + y_s^2 + z_s^2}}. \quad (3)
\]

Figure 3: Conditions at cell face between fluid and solid body for calculation of common wall temperature.
resolution of the boundary layer, the local heat fluxes are then determined with sufficient accuracy. For the solid blocks a lower resolution of the grid is sufficient, but in cases of high temperature gradients some attention should be given to the temperature dependency of the material heat conduction. Simulations of additional thermal barrier coatings (TBCs) require higher grid resolution in the solid surface regions. The use of a principally identical formulation and solution of the energy equation in the solid body blocks as well as in the fluid blocks is advantageous for the implementation and stability of the coupling procedure.

With respect to the simulation of turbomachinery flows, which are usually flows with high Reynolds numbers, the Reynolds-averaged Navier-Stokes (RANS) equations have to be solved. Thus, the modelling of the Reynolds stresses that arise from the averaging process becomes necessary. One of the most widely used simple turbulence models in turbomachinery flow simulation with high numerical stability and usefulness even in flows with small separation [56] is the algebraic model of Baldwin and Lomax [57]. Beside its low computation time another advantage of this model is that it is suitable for a conjugate calculation of the heat transfer. Thus, it has been implemented to the conjugate heat transfer and flow simulation code (CHTflow) developed at the Institute of Steam and Gas Turbines, Aachen University [45 – 48].

The numerical scheme of the code works on the basis of an implicit finite – volume method combined with a multi-block technique and structured grids. Upwind discretisation is used for the inviscid fluxes. With respect to numerical diffusion, a Godunov–type flux-differencing is employed. In order to achieve third order accuracy, van Leer’s MUSCL technique is used (Anderson et al., [58]). Since this Godunov flux is not sufficiently diffusive to guarantee stability in regions with complex flow phenomena, it is combined with a modified Steger–Warming flux (flux-vector splitting) (Eberle et al., [59]). The viscous fluxes are approximated using central differences. The coupling of fluid blocks and solid body blocks for the heat transfer task is achieved via the CCT described above. The code has been used in recent years successfully in a wide range of conjugate calculations of turbomachinery flows. A more detailed description of the basic equations, validation and calculation examples can be found in [49].

Instead of the algebraic model, it is also possible to use other and more complex turbulence models (e.g. low reynolds $k–\varepsilon$ models [60] or low reynolds $k–\omega$ models [61]). For these models, additional governing equations for the turbulence energy and the turbulence dissipation rate have to be solved. Craft et al. [62] have developed a cubic eddy-viscosity model, which has successfully been implemented and tested within CHT flow by Bohn et al. [63] for turbine-stage flows and by Bohn and Tümmers [64] for leading edge ejection. For a much more detailed discussion of different turbulence models and their application for turbomachinery flows, the reader should refer to articles by Shih and Sultanian [65] or Amano [66].

All results presented in the following sections have been obtained by application of the CHT flow solver.
3 Heat transfer design aspects for internal cooling

3.1 Validation of the CCT for air-cooled guide vanes

3.1.1 Conventional determination of internal heat transfer

In the case of a lower temperature level in the turbine, e.g. in the 3rd stage, the vanes and blades are usually cooled by air in internal convection cooling configurations. These cooling configurations usually consist of several radial cooling passages, in simple cases the passages are bore holes in the blade material. The internal heat transfer can be predicted on the basis of correlations for turbulent tube flows with acceptable accuracy, if all necessary internal flow conditions are known.

One example for a turbulent heat transfer correlation for a turbulent tube flow is given by the equation of Gnielinski [67]:

\[
Nu = \frac{\zeta / 8(Re - 1000)Pr}{1 + 12.7\sqrt[8]{\frac{\zeta}{8}} \left(Pr^{2/3} - 1\right) \left[1 + \left(\frac{d}{l}\right)^{2/3}\right]}.
\]  

(5)

with

\[
\zeta = \frac{1}{(0.79 \ln Re - 1.64)^2}.
\]  

(6)

The fluid properties have to be taken at the averaged temperature of the inflow and outflow conditions of the cooling passage:

\[
T_{mid} = \frac{T_{in} - T_{out}}{2}.
\]  

(7)

In the case of a strong dependency of the fluid properties on the local temperature, e.g. in the case of a strong thermal boundary layer, a further correction factor can be added to eqn (5) depending on the ratio of the different Pr numbers:

\[
Z = \left(\frac{Pr}{Pr_w}\right)^{0.11}.
\]  

(8)

In the conventional design process the inlet temperature of the cooling fluid is known but the outlet temperature depends on the quantity of the transferred heat. The determination of the quantitative heat transfer becomes an iterative process depending on the temperature difference of the cooling flow bulk temperature and the wall temperature. The wall temperature in every iteration step is a result of the FEM calculation of the blade solid body, which is dependent on the chosen external heat transfer condition. As described for the principle of the CCT, the external heat transfer condition might be affected by the influence of the heat transfer on the external fluid dynamics. Thus, with
respect to the external heat transfer the CCT is able to increase the accuracy of the thermal load prediction also in the case of prescription of the internal heat transfer by the above correlation. This strategy becomes necessary if only 2-D sections are investigated.

3.1.2 Convectively cooled test cases

The two examined configurations have been designed and tested by Hylton et al. [29] and Nealy et al. [68] to provide a data base for testing the predictive capabilities of analytical models and numerical codes. The investigations cover a wide range of operating conditions and geometries at simulated engine conditions. The two airfoils to be discussed are the:

a) Mark–II profile

The Mark–II vane is a high-pressure turbine nozzle guide vane convectively cooled by ten cooling channels supplied with air. The vane material is ASTM 310 stainless steel. The temperature dependency of the thermal conductivity has been taken into account in the calculations. The 2-D grid has a resolution of 11920 grid points in the fluid domain and 3112 grid points in the solid domain making use of a multi-block hybrid grid. The $y^+$-value is less than 1 in order to accurately predict the near-wall flow characteristics such as skin friction and heat transfer. The cooling configuration is shown in Figure 4a.

b) C3X profile

The geometry of the convectively cooled C3X vane is similar to the Mark–II vane, and it is also fabricated of ASTM 310 stainless steel. The main differences are the leading edge shape, the arrangement of the 10 cooling holes, and the circular trailing edge. The 2-D grid consists of 14708 grid points in the fluid domain and of 7506 grid points in the solid domain. The circular shape of the trailing edge is taken into account by discretising the passage between the blades with the help of an H–O–H-type grid. Figure 4b shows the C3X vane and the cooling configuration.

For the 2-D calculation, heat transfer coefficients have to be prescribed for the internal cooling channels, because their flow direction is perpendicular to the main flow. Depending on the measurement results for the internal flow conditions, the internal heat transfer is calculated by a simplified $Nu$ correlation:

$$Nu = Cr \cdot (0.022 \cdot Pr^{0.5} Re^{0.8}) \cdot$$  \hspace{1cm} (9)

Here, $Cr$ is a correction factor mainly depending on the diameter of the holes [29]. The reference temperatures $T_{mid}$ have been obtained from the measured inlet and outlet temperatures for the cooling passages. No heat transfer coefficients are necessary on the outer wall of the vanes due to the CCT. The Baldwin–Lomax model has been used for turbulence modelling. The transition point on the suction side has been set close behind the position of the...
compression shock. Several operating points have been investigated and the main flow boundary conditions can be obtained from Table 1.

3.1.3 Thermal validation results
In Fig. 5a comparison of experimental and numerical data for the different operating points proves the high quality of the results for the Mark–II test case. The pressure distributions and the temperature distributions are given on the

Table 1: Test case conditions.

<table>
<thead>
<tr>
<th>Test case</th>
<th>Mark II</th>
<th>C3X</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>4311</td>
<td>4522</td>
</tr>
<tr>
<td>$p_{l1}$ [bar]</td>
<td>2.765</td>
<td>3.907</td>
</tr>
<tr>
<td>$T_{l1}$ [K]</td>
<td>803</td>
<td>719</td>
</tr>
<tr>
<td>$M_1$ [-]</td>
<td>0.18</td>
<td>0.18</td>
</tr>
<tr>
<td>$Re_1$ [$10^6$]</td>
<td>0.45</td>
<td>0.72</td>
</tr>
<tr>
<td>$M_2$ [-]</td>
<td>0.90</td>
<td>0.91</td>
</tr>
<tr>
<td>$Re_2$ [$10^6$]</td>
<td>1.56</td>
<td>2.52</td>
</tr>
<tr>
<td>$p_{2}$ [bar]</td>
<td>1.635</td>
<td>2.285</td>
</tr>
<tr>
<td>$T_u$ [%]</td>
<td>6.5</td>
<td>8.3</td>
</tr>
<tr>
<td>$T_w/T_g$ [-]</td>
<td>0.71</td>
<td>0.83</td>
</tr>
</tbody>
</table>
surface of the vane. On the suction side the rapid acceleration of the flow leads to
a strong shock. Further downstream the flow accelerates again resulting in a
weak shock in run 5411.

Considering the temperature distribution along the vane surface, a local
temperature maximum can be observed in the vicinity of the stagnation point. Downstream, the following surface temperature distribution develops on the
suction side. The surface temperature reaches a local minimum up to the point where the profile shape changes from convex to concave. This is due to the fact that in a flow along a convex surface the heat transfer decreases and along a concave surface the heat transfer increases. Similar to the pressure side, a local temperature minimum is reached. The interaction of the compression shock with the boundary layer leads to a narrow flow separation and reattachment. Due to this disturbance, the flow along the suction side is now considered to be turbulent. Behind the compression shock, the heat transfer rises strongly, leading to a rapid temperature increase. On both the pressure side and suction side a local surface temperature maximum always exists in between two hole positions, and a local surface temperature minimum in the proximity of a hole. Due to the small cooling holes and the thin trailing edge, the surface temperatures are very high in the trailing edge region. According to its ratio $T_w/T_g$, run 4522 shows a flat temperature distribution compared to the other two runs.

Figure 6 serves to validate the CCT for the C3X vane by the comparison of data from experiments and from the conjugate calculation. On the pressure side, the laminar flow calculation provides excellent results. On the suction side, the

Figure 6: C3X test case results for surface pressure and temperature.
more complicated flow behaviour leads to some deviations. Whenever shocks occur and a small transition zone appears (e.g. for Mark–II test cases, Fig.5) the pressure distribution and the temperature distribution show excellent results. But, for the C3X test case, in particular in run 4422, the flow seems to form a laminar separation bubble, followed by transition and reattachment. Although the algebraic turbulence model is not able to consider all flow phenomena occurring in this region, the deviation from the experimental temperature data is still acceptable. Improved turbulence modelling with special focus on the modelling of the transition region instead of transition points is under investigation. A more detailed discussion of the simulation results is given by Bohn and Heuer [69].

3.1.4 Accuracy of the results
According to eqn (10) the relative error is defined as the difference between the experimental data and the data from the CCT, divided by the experimental value at the corresponding location \( x/L \).

\[
\varepsilon_{rel} = \frac{T_{exp} - T_{CCT}}{T_{exp}}. \tag{10}
\]

To achieve the mean relative error, the total of all relative errors is summed up and divided by the number of control points:

\[
\varepsilon_{rel,mean} = \frac{1}{n} \sum_{i=1}^{n} \frac{|T_{exp} - T_{CCT}|}{T_{exp}}. \tag{11}
\]

In Table 2 the maximum and the mean relative errors for all investigated test cases are summarized. Since the maximum relative error lies between 2.21\% and 3.22\% and, furthermore, the mean relative error is between 0.710\% and 1.136\%, even the quantitative results are excellent.

<table>
<thead>
<tr>
<th>Test case</th>
<th>Mark–II</th>
<th>C3X</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>4311</td>
<td>4522</td>
</tr>
<tr>
<td>( \varepsilon_{rel, max} ) [%]</td>
<td>2.40</td>
<td>2.47</td>
</tr>
<tr>
<td>( \varepsilon_{rel, mean} ) [%]</td>
<td>0.710</td>
<td>0.831</td>
</tr>
</tbody>
</table>
3.2 Simulation of internal steam cooling

3.2.1 Efficiency potential of steam cooling

Conventionally cooled gas turbines use compressor air for cooling purposes. Optimisation of the cycle process demands higher firing temperatures and reduced cooling efforts. Beside its favourable fluid properties, process steam used as cooling fluid provides several benefits in comparison to air-cooling. In a combined-cycle power plant, steam can be provided ‘inexpensively’ at favourable supplying conditions of pressure and temperature levels.

Due to the improved cooling conditions, convective cooling in closed circuits can be established. This means that convective cooling might be sufficient for cooling purposes without any additional film cooling. Thus, mixing losses are avoided and, in particular, the temperature drop across the first-stage nozzle is reduced significantly leading to a higher firing temperature in front of the first blade while the combustion temperature can remain as low as necessary for low-NOx purposes. Additionally, the closed-circuit steam cooling (CCSC) permits the recovery of the thermal energy taken up in the gas turbine by expansion in the steam turbine.

Attracted by these benefits, several manufacturers of gas turbines have significantly increased their efforts on developing machines for commercial operation (e.g. Corman [70, 71], Maekawa et al. [72], Gaul et al. [73]).

Alderson et al. [74] have performed an extensive parametric study within a cycle analysis over a range of firing temperatures for a closed-circuit steam cooling configuration, which has achieved better performance in comparison to open-circuit steam cooling. They were also able to show that the net plant efficiency (combined cycle, CC) of the investigated CCSC cycle is about 1.5 percentage points better than the cycle with an air-cooled turbine at the same firing temperature. Recent cycle analysis studies of Jordal and Torisson [75] have also shown that the benefits of replacing the air-cooling system of the vanes with steam in a CCSC cycle are around 1.5 percentage points. For a detailed analysis of different blade cooling solutions by using air and steam as cooling fluids the reader is referred to the extensive study by Facchini et al. [76].

By application of a process analysis tool, the thermal efficiency of the steam-cooled cycle can be compared to a conventional combined cycle with air cooling of all vanes and blades [77]. The conventional process is used as a reference and achieves a thermal efficiency of $\eta_{th}=58\%$ (Table 3). For comparison, steam-cooling of the first vane row has been investigated by using a closed-circuit steam cycle.

In a first calculation with steam-cooling, the combustion temperature $T_B$ was kept constant. As a result, the firing temperature (in front of the first blade row) increases significantly ($T_F=1392\, K$) as mixing with cooling air in the vane row is avoided. The thermal efficiency rises significantly to approximately $\eta_{th}=59.4\%$ (Table 3). This value has to be stated as the maximum possible increase in efficiency of the investigated system, because it is assumed that additional
cooling with air of the first blade row is not necessary although there is an increase in the firing temperature. In fact, additional cooling will be necessary, reducing the benefit generated by the steam-cooled vane row.

A second calculation has been executed in order to show the benefit by steam-cooling of the first vane row in the case of a constant firing temperature at the first blade row. As a result, the combustion temperature has to be decreased. This might lead to instabilities in the combustion, which have to be taken into account in a re-design of the combustion chamber. The effect on the thermal efficiency of the cycle is an increase of 0.8 percentage points (Table 3). These results underline the potential of steam-cooling even in the case of cooling only the first vane row with steam in a closed-circuit cycle.

### 3.2.2 Test configuration for steam-cooling

Figure 7 shows the geometrical configuration, which consists of a rectangular duct containing a three-vane cascade [77, 78]. The central vane can be convectively cooled by supplying steam or compressor air to 22 straight radial cooling passages. The arrangement of the cooling passages within the cooled vane is illustrated in a mid-cross-sectional cut on the bottom left side of Fig. 7, while the corresponding diameters of the cooling passages are listed in the table on the bottom right side of Fig. 7. There are two different cooling passage diameters with respect to the different heat transfer characteristics expected on the pressure side and the suction side of the vane. As the heat transfer rate is intended to be lower on the pressure side, the diameter of the cooling passages is only 1 mm for the cooling passages located near the pressure side (PS1 to PS8). In the area of the trailing edge, where minimum wall thicknesses must be considered, the cooling passage (TE1) also has a diameter of just 1 mm.

Based on the geometry, 3-D numerical investigations using the conjugate calculation technique (CCT) have been performed, providing simulation data in a wide range of cooling steam supply pressures. Material properties for IN738LC (conventionally casted) for the vane material have been used.

<table>
<thead>
<tr>
<th></th>
<th>Thermal efficiency $\eta_{th}$ [%]</th>
<th>Combustion temperature $T_B$ [K]</th>
<th>Firing temperature $T_F$ [K]</th>
<th>El. cycle output $P_{el}$ [MW]</th>
</tr>
</thead>
<tbody>
<tr>
<td>CC with cooling air (1st vane row &amp; other rows)</td>
<td>58</td>
<td>1421</td>
<td>1326</td>
<td>262</td>
</tr>
<tr>
<td>CC with CCSC (1st vane row, $T_B$ constant)</td>
<td>59.4</td>
<td>1421</td>
<td>1392</td>
<td>292</td>
</tr>
<tr>
<td>CC with CCSC (1st vane row, $T_F$ constant)</td>
<td>58.8</td>
<td>1353</td>
<td>1326</td>
<td>267</td>
</tr>
</tbody>
</table>

Table 3: Results of cycle analysis.
Figure 8 shows the 3-D numerical grid used to solve the discretised governing equations. Due to the demand of the CHTflow solver the hot gas path, the cooling passage flow, and the solid body of the central vane have to be covered by a fully structured numerical multi-block grid. Using the CCT, there is no need for any boundary information about heat transfer characteristics at fluid/solid interfaces. The precise calculation of the temperature gradients at all solid/fluid interfaces requires that the non-dimensional wall distance $y^+$ of the first cell row is kept at an acceptable level for a conjugate calculation. This demandingly high resolution of the thermal boundary layer plus the need for including the hot gas path, cooling flow, and the solid body of the central vane raises the number of grid points up to 2.8 million. Divided into 172 structured blocks, the whole computation domain is separated into several tasks, which are calculated separately by ordinary PCs making use of the parallel structure of the numerical code.

Figure 7: Geometry of steam-cooled vane.
3.2.3 Parametric study

In order to show the influence of realistic cooling steam supply conditions on the cooling effectiveness (eqn (12)) of the examined test configuration, a parametric study is carried out.

\[ \eta_c = \frac{T_G - T_g}{T_G - T_e} \cdot \frac{\dot{m}_c \cdot c_p}{h \cdot A}. \] (12)

Table 4 shows the boundary conditions for the hot gas flow, which are constant over the whole study. The aim of the parametric study is to provide information about the influence of the non-dimensional cooling mass-flow on the cooling effectiveness of the configuration, to quantify the influence of cooling steam supply conditions (in particular the cooling steam pressure level) and to compare the reachable cooling effectiveness in the case that compressor air is used as a cooling fluid. Therefore, several 3-D calculations have been carried out in the study. Table 5 lists the boundary conditions of the cooling fluid flow for these different cases. To reduce the number of influences, the temperature of the supplied cooling fluid is held constant at 435 °C. At the entrance of the internal cooling passages, a constant velocity profile has been prescribed. Thus, some inaccuracies in the conjugate calculation of the hole entrances have to be taken into account.

The influence of the non-dimensional cooling mass-flow can be easily
shown by varying the numerator of eqn (13) (see cases 1, 3 and 5 in Table 5), which represents the offered cooling potential \( \text{c.p.} = m_c \cdot c_p \). While the geometry and boundary conditions of the hot gas flow are invariable over the whole study, the averaged external heat transfer coefficient on the vane surface can also be assumed as almost constant in order to simplify the evaluation for the dependency of the global cooling effectiveness of the cooling potential. In fact, there is, of course, a slight influence of the changed wall temperatures due to the different internal cooling conditions on the external flow conditions in the boundary layer. For the conjugate calculation itself, this is taken into account as the local external heat transfer is calculated on the local flow conditions and not on the basis of a heat transfer coefficient distribution.

To point out the influence of cooling steam supply conditions, the pressure level of the cooling steam is varied for the 3 different values of the cooling potential (see cases 2, 4 and 6 in Table 5). As the cooling effectiveness of a convectively cooled vane depends strongly on the heat flux into the cooling fluid, the physical model of turbulent heat transfer in tubes is used to specify appropriate boundary conditions for the cooling steam supply. The Nusselt number, which can be recognised as an indicator of heat transfer quality within the cooling passages depends on the Reynolds and Prandtl Number of the fluid flow. Therefore, because of the different pressure dependency of superheated steam properties, it is expected to reach a different cooling effectiveness for the

### Table 4: Hot-gas boundary conditions and characteristic numbers.

<table>
<thead>
<tr>
<th>Velocity (inlet) [m/s]</th>
<th>Density (inlet) [kg/m³]</th>
<th>Stat. pressure (outlet) [bar]</th>
<th>Inlet ( Re_l )-Number</th>
<th>Inlet ( Ma_l )-Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>208</td>
<td>4.641</td>
<td>13.37</td>
<td>890000</td>
<td>0.3</td>
</tr>
</tbody>
</table>

### Table 5: Supply conditions of the cooling fluid.

<table>
<thead>
<tr>
<th>Case #</th>
<th>Cooling fluid</th>
<th>Pressure (inlet) [bar]</th>
<th>Cooling potential ( m_c \cdot c_p ) [kJ/(s.K)]</th>
<th>( Re_{D_{a1.5}} ) [-]</th>
<th>( Pr_c ) [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>steam</td>
<td>100</td>
<td>0.02347</td>
<td>270000</td>
<td>1.054</td>
</tr>
<tr>
<td>2</td>
<td>steam</td>
<td>150</td>
<td>0.02347</td>
<td>218000</td>
<td>1.157</td>
</tr>
<tr>
<td>3</td>
<td>steam</td>
<td>50</td>
<td>0.00928</td>
<td>127000</td>
<td>0.974</td>
</tr>
<tr>
<td>4</td>
<td>steam</td>
<td>100</td>
<td>0.00928</td>
<td>107000</td>
<td>1.054</td>
</tr>
<tr>
<td>5</td>
<td>steam</td>
<td>20</td>
<td>0.003288</td>
<td>49000</td>
<td>0.938</td>
</tr>
<tr>
<td>6</td>
<td>steam</td>
<td>50</td>
<td>0.003288</td>
<td>45000</td>
<td>0.974</td>
</tr>
<tr>
<td>7</td>
<td>compr. air</td>
<td>20</td>
<td>0.003271</td>
<td>77000</td>
<td>0.745</td>
</tr>
</tbody>
</table>
examined configuration when applying the same offered cooling potential but with different cooling steam pressure levels.

The application of compressor air as cooling fluid (see case 7 in Table 5) leads to lower cooling potential values, which are comparable to those realised in cases 5 and 6 for steam as cooling fluid. Because of the fluid properties of compressor air compared to those of superheated steam, the high cooling potential – easily realised when using steam – are impossible to reach with air as cooling fluid for realistic pressure levels of the compressor air. Due to low density and low specific heat, the limiting factor when using air is the velocity in the cooling passage. As shown later, the effort of compressor air to reach a comparable cooling effectiveness of the configuration is much higher than the corresponding steam mass-flow.

3.2.4 Thermal results of conjugate heat transfer simulation

Figure 9 shows the pressure distribution of the central vane (cooled vane) of the cascade at 50% vane height (mid-plane). Due to the linear shape of the vane and the plane cascade, there are only minor differences between the pressure distributions near the side walls and the pressure distribution shown for the mid-plane. Furthermore, there are minor pressure differences to be found as a result of the different internal cooling conditions and their consequences on the vane surface temperature and, thus, on the boundary layer development. These differences are very small and, therefore, the external flow field can be assumed as almost independent of the internal cooling conditions.

Figure 10 shows the external surface temperature distributions for two selected cases of the study. Case no. 1 is with high cooling potential (1st operating point, c.p.=0.0234 kJ s⁻¹ K⁻¹) and a high Reynolds number of the passage flow ($Re_{D=1,s}=270000$). Thus, surface temperatures are significantly lower.
lower in comparison to case no. 6 with low cooling potential (0.0033 \text{ kJ s}^{-1} \text{ K}^{-1}) and relatively low Reynolds number ($Re_{D_{L.5}}=45000$). The temperature distributions are given in three different radial planes. The first one is located near the inner side wall (cooling steam inflow), the second plane is the mid-span and the third plane is near the outer side wall (cooling steam outflow). With the cooling steam taking up heat on its internal way from the inner side wall to the outer side wall, the external surface temperatures rise. A local temperature peak is always to be found at the leading edge. Local minima and maxima in the temperature distributions along the suction side and the pressure side correspond to the positions of the cooling passages. Very high temperatures are to be found at the trailing edge, where no sufficient cooling can be established by the simple cooling passage configuration. Here, temperatures are only a few degrees lower than the external hot gas temperature.

Figure 11 gives a comparison of the internal vane temperatures in the cutting plane for both cases at 50\% height. The solid body temperature field shows a nearly homogeneous distribution inside the cooling passage arrangement. Here, temperatures are about 150 K lower for case no. 1 than in case no. 6. In the trailing edge region, the temperature increase leads to very high temperature gradients. This has to be taken into account in a succeeding thermal stress analysis.

Figure 12 gives a good impression of the conjugate calculation principle. The temperature field and the numerical grid of the leading edge region are shown in Fig. 12a. Due to the conjugate coupling of fluid flow and solid body, grid resolution is very high at all contact faces of fluid flow and solid body. The temperature distribution along a section line – as marked in Fig. 12a – is given in
Fig. 12b. Starting in the external hot gas region, the temperature is nearly constant until the very thin external temperature boundary layer is reached, where temperature gradients are extremely high. In the solid body region, the temperature decrease is nearly linear. The gradient of the curve in combination with the thermal conductivity of the vane material gives the local specific heat flux, which rises when approaching the cooling passage surface. For the internal fluid flow, typical temperature boundary layers for turbulent tube flow are visible. As expected, a higher temperature gradient at the internal wall is to be found in the boundary layer for more intensive convective cooling (case 1).

3.2.5 Cooling effectiveness analysis

A global cooling effectiveness can be given, when $T_w$ is calculated as an averaged surface temperature of the vane. Figure 13 shows the global cooling effectiveness of all calculated cases plotted against their corresponding cooling potentials. Due to the insufficient cooling of the trailing edge, the global values are relatively low. For establishing the same cooling potential for the 100 bar pressure level in case no. 1 as for the 150-bar pressure level in case no. 2, it is necessary to increase the cooling passage flow velocities and the steam mass flow, because the pressure dependency of $c_p$ has also to be taken into account ($T_c$ is kept constant!). Thus, $Re_D$ in case no. 1 is somewhat higher than in case
no. 2 so that a higher cooling effectiveness is expected although the reduction of the Pr-number has an opposite – but smaller – effect than the increase of $Re_D$.

For a defined cooling potential, the amount of compressor air has to be significantly higher than the amount of cooling steam. Figure 14 shows this in respect to the low cooling potential condition (compare cases no. 5, 6 and 7, same cooling potential). For the presented cooling configuration, the amount of cooling air is about $\Delta\eta_c = 94\%$ higher than the amount of cooling steam in case no. 5, which has the same pressure level ($p_c = 20$ bar). The global cooling effectiveness for the air-cooled case is about $\Delta\eta_c = 10\%$ lower than the linear approximated cooling effectiveness for the same cooling mass flow when using steam. It is obvious that the very high cooling potentials can not be realised with compressor air as this will lead to an extremely high cooling mass flow that cannot be established in the fixed cooling geometry of the radial cooling channels.

Figure 12: Temperatures along section line at the leading edge.
Figure 13: Global cooling effectiveness plotted against cooling potential.

Figure 14: Global cooling effectiveness plotted against cooling mass flow.
4 Heat transfer design aspects for film cooling

4.1 Conventional heat transfer determination

With respect to most of the experimental studies (e.g. Gritsch et al. [79], Lutum et al. [80], Yuen et al. [81], Reiss and Bölcs [82]) the determination of the adiabatic film cooling effectiveness is of main importance. The adiabatic film cooling effectiveness is one of the two important parameters of the conventional approach for the determination of the heat transfer rate:

\[ q = h_f(T_w - T_{aw}). \]  \hspace{1cm} (14)

Here, \( h_f \) is the film heat transfer coefficient, \( T_w \) is the wall temperature, and \( T_{aw} \) is the adiabatic wall temperature in the case of film-cooling as a reference temperature. The heat transfer coefficient \( h_f \) considers the influence of the film-cooling on the local heat transfer due to the modified flow field. Due to the typical characteristic numbers for turbine flows, the heat transfer for film cooling configurations will be dominated by turbulent phenomena.

Without film cooling \( T_{aw} \) will be the recovery temperature \( T_r \) of the hot gas, which is given by:

\[ T_r = T_g \left( 1 + r \frac{\gamma - 1}{2} (Ma_g)^2 \right). \]  \hspace{1cm} (15)

The performance of cooling can be expressed as the adiabatic film cooling effectiveness:

\[ \eta_f = \frac{(T_{aw} - T_r)}{(T_{oc} - T_r)}. \]  \hspace{1cm} (16)

Here, \( T_{oc} \) is the stagnation temperature of the cooling fluid near entry. If the adiabatic cooling effectiveness is known, the adiabatic wall temperature can be determined. For the solution of eqn (14) the knowledge of the film heat transfer coefficient is necessary, too. Experiments with constant and defined heat transfer rates (e.g. Lutum et al. [80]) give the possibility to calculate these coefficients, if wall temperatures are measured accurately (e.g. by an infrared thermography system [79]). Determination of the local heat transfer coefficients is also possible by a thermochromic liquid crystal (TLC) layer (e.g. Brandt et al. [83], Yuen et al. [81], Mayhew et al. [84]). Depending on the local heat transfer the TLC displays different colours, representing the local surface temperatures. Visual scanning of the surface in the case of an alternating heating input allows the local heat transfer coefficients to be obtained from the resulting colour pictures.

An alternative approach is given by the linear superposition principle (e.g. Choe et al. [85]):

\[ h(\theta) = h_f (1 - \eta_f \theta). \]  \hspace{1cm} (17)
All effects of the cooling film are described with the value of the heat transfer coefficient \( h(\theta) \) as a function of the non-dimensional wall temperature

\[
\theta = \frac{(T_r - T_{oc})}{(T_r - T_w)},
\]

for the calculation of heat transfer rate based on

\[
q = h(\theta) \cdot (T_r - T_w).
\]

Due to this principle, two different sets of measurements of the wall temperature are sufficient to determine the two basic parameter \( \eta_f \) and \( h_f \) (e.g. Dittmar et al. [86], Saumweber et al. [87]).

Whereas for most of the above mentioned studies flow along a flat plate is considered, the leading edge film cooling phenomena of a blade are also of great interest. Here, experimental investigations on the adiabatic film cooling effectiveness have been provided by Cruse [88]. The results have been used as a test case for numerical simulations [e.g. 89, 90]. Experiments by Yuki [91] are used as validation data in extensive numerical studies on the leading edge film-cooling physics by York and Leylek [92, 93]. Numerical studies on the 3-D cooling jet phenomena for blade leading edge ejection have been performed by Bohn and Kusterer [94, 95].

4.2 Conjugate influence on turbulent heat transfer

It is obvious that both 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 of importance, too. 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, it has an influence on the external cooling performance.

The following results of a numerical study on film cooling of a duct wall show the influence of the conjugate heat transfer on the flow field, in particular on the secondary flows in the cooling jets.

4.2.1 Geometric configuration for combined adiabatic and conjugate study

In this study, a hot gas duct flow with cooling fluid injection through one row of eight cooling holes is investigated. The hole geometry comprises different configurations with cylindrical holes and shaped exits (diffusor and fan shaped). The duct wall with cooling fluid ejection is calculated with two different heat
transfer boundary conditions within the same calculation. The first part (holes no. 1 to no. 4) is calculated with the conjugate heat transfer condition. Thus, heat fluxes between the fluid flow and the solid body and vice versa are calculated directly. The second half of the wall (holes no. 5 to no. 8) is calculated with the adiabatic wall condition. The cooling fluid is supplied to the holes by a rectangular duct from one side with a 90° angle to the streamwise direction (x-coordinate, \(x=0\) at hole outlet). Therefore, the situation of the hole inflow is similar to the supply of cooling holes in real-blade configurations. At the end of the rectangular cooling duct a small exit hole similar to a blade tip hole is part of the calculation. The geometry of the duct and the solution domain are illustrated in Fig. 15.

Figure 16 shows the hole configurations and their basic parameters. The hole diameter is \(D=1\) mm and the hole spacing is \(P/D=3\). The film cooling phenomena of each configuration are calculated with the boundary conditions listed in Table 6. The inlet mass flows for the hot gas channel and the cooling supply are fixed precisely by the boundary conditions. Therefore, the blowing ratio

\[
M = \frac{(\rho_e c_v)}{(\rho_s c_g)},
\] 

(20)
The ratio of density and velocity values for the cooling flow and the hot gas flow, has been set to 1.5 for the holes of all configurations.

4.2.2 Secondary flow analysis

Figure 17 is an illustration of the secondary flow vectors for the two cooling jets in the middle of the duct in axial cutting planes at two downstream positions ($x/D=5$ and $x/D=10$). Whereas cooling jet no. 4 is the last one of the conjugate part (left hand side) of the wall, cooling jet no. 5 is the first one of the adiabatic part. The secondary flow vectors for the cylindrical hole configuration reveal the counter-rotating vortices of the kidney-vortex system. It can be shown that the vortex system is not perfectly symmetrical, as is usually the case for a non-lateral (streamwise) ejection. The reason can be found in the cooling fluid supply, which is from one side only and not in the streamwise direction. As a result, a strong vortex in the cooling hole is induced from the inflow into the hole. The vortex in the cooling hole supports one part of the vortex system in the cooling jet leading to an asymmetric appearance of the kidney-vortex. The secondary flow vectors show that the intensity of the secondary flows is reduced at the downstream position ($x/D=10$). Furthermore, the size of the flow vectors indicates that secondary flows for the cooling jet in the adiabatic part might be a little larger than for the cooling jet close to the conjugate heat transfer wall. This will be investigated in more detail below.

For the diffusor hole configuration, it can be shown that the secondary flows are significantly reduced. The length scaling of the vectors has been nearly doubled (0.08 to 0.15) in order to show appropriate details of the secondary flows. A slightly asymmetric kidney-vortex can also be observed. The velocity vectors indicate a difference in the intensity of the secondary flow. In the case of the fan shaped hole, the momentum of the injected cooling fluid is decreased and, thus, the interaction of the main flow and the cooling jet is reduced leading to weak secondary flow structures. Furthermore, the cooling fluid spreads out into the lateral direction. Thus, little hot gas is able to flow between the cooling jets.

As stated above, the secondary flow vectors give hints on the different intensity of the vortices for the conjugate and adiabatic part of the calculation. Figure 18 shows a quantitative comparison of the secondary flow velocity. The results show for the cylindrical hole configuration that the left part of the vortex-system in the jet is intensified (e.g. for $x/D=5$: 18 m/s vs. 10 m/s for jet no. 4 (conjugate), 22 m/s vs. 14 m/s for jet no. 5 (adiabatic)). The peak values of the secondary velocities also prove that the induced secondary flows for the part

<table>
<thead>
<tr>
<th>Table 6: Boundary conditions.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>parameter</strong></td>
</tr>
<tr>
<td>Inlet density (kg/m$^3$)</td>
</tr>
<tr>
<td>Inlet velocity (m/s)</td>
</tr>
<tr>
<td>Outlet pressure (Pa)</td>
</tr>
</tbody>
</table>
with the adiabatic wall are higher than for the conjugate part. This becomes even more obvious for the positions further downstream (e.g. $x/D = 10$), where the peak values of the adiabatic part are nearly twice as high as for the conjugate part. For the diffusor hole ejection, the analysis shows secondary velocities, which are significantly smaller than for the cylindrical hole by a factor of two to three. Here, it also becomes obvious that the magnitude of the induced secondary flow velocities is distinctly higher in the adiabatic part. Significant values of secondary flow velocities for the fan shaped hole can only be observed directly behind the ejection in the adiabatic part. Further downstream, all secondary flows are very small and no significant differences between the adiabatic and the conjugate part can be detected.
The secondary flow analysis shows that the different treatment of the wall leads to reduced secondary flows and deviations in the jet flow structures for the conjugate heat transfer calculation, which is more realistic with respect to the real blade flow with heat transfer.

4.2.3 Thermal flow analysis

The second part of the results deals with the temperature distribution in the cooling jets. Figure 19 gives an illustration of the dimensionless flow temperature in different axial cutting planes for the two cooling jets in the middle of the duct (no. 4 and no. 5). The cylindrical hole ejection shows a lift-off of the cooling jet and, thus, the core of the jet is not close to the wall and hot gas
contact occurs between the jets. For the conjugate part, heat fluxes from the hot main flow into the wall in front of the ejection and between the jets lead to heating-up of the solid body. At the position of the cooling jets, heat transfer from the hot wall into the cooling jets occurs, which leads to an additional temperature increase of the cooling jet in comparison to the adiabatic case.

The ejection with diffusor-shaped holes shows that the cores of the cooling jets are now close to the wall and the lift-off of the jets is avoided. Furthermore, the cooling jets spread out slightly into the lateral direction leading to reduced hot gas contact between the jets. Due to the conjugate wall condition the cooling jet no. 4 heats up more quickly than jet no. 5 of the adiabatic side. The fan shaped configuration leads to a significantly improved cooling film as the lateral
extension of the single jets is further increased.

For the adiabatic surface of the duct wall, Fig. 20 shows the adiabatic cooling effectiveness. In the case of the conjugate part of the duct wall, the local wall temperatures instead of adiabatic wall temperatures have been used. Thus, the obtained values indicate the complete cooling effectiveness for the film cooling and the convective cooling of the holes in the conjugate part.

The adiabatic effectiveness shows the significant improvement of the cooling performance for the shaped configurations whereas the lift-off in the cylindrical case leads to a very poor performance. Furthermore, the improved lateral extension of the single jets for the fan shaped configuration keeps only a very narrow region of reduced performance between the jets.

For the conjugate part, the effectiveness value of 20% for the cylindrical hole ejection means that only 20% of the cooling potential (temperature difference between the main flow adiabatic wall temperature and the stagnation temperature of cooling fluid) is used whereas for the diffusor-shaped holes it is up to 40%. The fan shaped configuration leads to effectiveness values, that are over 60% close to the hole outlets.

Figure 20: Cooling effectiveness on wall surface ($\Delta \eta_{iso}=0.2$).
4.3 Modern cooling technology: Full-coverage cooling simulation

Increasing the efficiency of modern gas turbines is still an important objective of scientific research. One of the most important issues is the improvement of cooling technology both in the combustion chamber and in the vanes and blades of the turbine. State-of-the-art cooling technology are film cooling configurations that reduce the thermal load of gas turbine components both by convective cooling and protection of the material from hot gas contact. Due to secondary flow phenomena, in particular the well-known kidney vortex, the interaction of cooling fluid and hot gas flow locally leads to a thermal load that can not be borne by the material.

The complete protection of the material from hot gas contact leads to a modern cooling technology called full coverage cooling. Here, the cooled walls of gas turbine components are repleted with small cooling holes that lead to the development of a homogeneous cooling film on the surface. The geometric design has to take into account the interaction of the vortex systems that are generated by each hole so that the cooled surface is not penetrated by hot gas.

Film cooling configurations have been investigated for several years. Many numerical investigations focus on the development of the kidney vortices (Bergeles et al. [96]). As an example, a detailed numerical analysis of the film cooling physics in the case of a flat plate with one row of cooling holes has been executed by McGovern and Leylek [97]. Recently, the physical phenomena of leading edge film cooling were investigated by York and Leylek [92, 93]. Experimental investigation of plates cooled by one or two rows of cylindrical and shaped holes were performed by several authors (e.g. Bunker [98], Dittmar et al. [86] and Saumweber et al. [87]). Aerothermal numerical analyses of film cooling for gas turbine blades were presented by Garg [99] and Heidmann et al. [100].

The entire flow field and heat transfer analysis of film cooled gas turbine components can be accomplished numerically by application of the conjugate technology (e.g. Bohn and Kusterer [94]). Here, this technology has been used to design the full coverage cooling for a multi-layer plate representing the walls of a gas turbine combustion chamber [101–104].

4.3.1 Geometric configuration

The numerical analysis of a full coverage cooling configuration has been performed with a flat plate composed of three materials. The substrate layer is of the super alloy CMSX-4 and has a thickness of 2.0 mm (Fig. 21). The bondcoat consists of a MCrAlY layer with a thickness of 0.15 mm. The thermal barrier coating (TBC) is a yttrium-stabilized ZrO2 layer with 0.25 mm thickness.

The plates are perforated with seven rows of cooling holes with a diameter of $D=0.2$ mm. Altogether, five cooling configurations with cylindrical and shaped holes are investigated. As shown in Table 7, the cylindrical configurations vary the distance between two rows of holes and the inclination...
angle $\alpha$ of the holes. The configurations with shaped holes are derived from configuration C30-06. The outer and inner shaping is applied in the region of the TBC, the main parameters can be found in Table 7.

### 4.3.2 Computational grid and boundary conditions

Figure 22 shows, as an example, the numerical grid in the solid region of configuration S30-06-F. It consists of 180 blocks in total (solid region and fluid flow) with approximately 1,200,000 points. To reduce the numerical effort only two half lines of rows are investigated. At the $z$-planes through the middle axis of the holes symmetry boundary conditions are used.

At the main flow channel the inlet and outlet boundary conditions are specified as shown in Fig. 23. In a distance of 5 mm from the plate surface an adiabatic wall boundary condition is set. The plenum for cooling fluid supply is not part of this investigation, so that at each hole inlet the same boundary condition is set. The values for temperature and pressure are derived from the flow conditions in modern gas turbine combustion chambers. The velocity of the hot gas stream yields to approximately 75 m/s, which leads to a $Ma$ number of 0.1 in the main flow.

### 4.3.3 Results

In this section the numerical results for the five cooling configurations are discussed. Both the cooling effectiveness distribution and vector plots are shown at three $x$-positions that are placed downstream of the fourth row of cooling

---

Table 7: Parameters of the investigated configurations.

<table>
<thead>
<tr>
<th>configuration</th>
<th>$L$</th>
<th>$\alpha$</th>
<th>$\beta_1$</th>
<th>$\beta_2$</th>
<th>$B$</th>
</tr>
</thead>
<tbody>
<tr>
<td>C45-10</td>
<td>10D</td>
<td>45°</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>C45-06</td>
<td>6D</td>
<td>45°</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>C30-06</td>
<td>6D</td>
<td>30°</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>S30-06-F</td>
<td>6D</td>
<td>30°</td>
<td>8°</td>
<td>0°</td>
<td>0.6088 D</td>
</tr>
<tr>
<td>S30-06-FL</td>
<td>6D</td>
<td>30°</td>
<td>8°</td>
<td>10°</td>
<td>0.6088 D</td>
</tr>
</tbody>
</table>

---

Figure 21: Geometry of the investigated plate.
holes. The first cutting plane is set at a distance of 0.2 mm from the center of the outlet square of the cooling hole. In the case of the shaped holes, the outlet shape is considered to be elliptical to obtain the cutting plane position. The following cutting planes are placed in a distance of 0.2 mm to the preceding plane. They are numbered from 1 to 3.

a) Cooling effectiveness distribution

Figure 24 shows plots of the isolines of the cooling effectiveness $\eta$ in the region of the cooling jets on the flat plate. The numerical domain is mirrored at the symmetry boundary ($z$-plane) so that the row #4 is displayed as a complete hole. In the middle of each picture one can find the cooling jet downstream of the fourth hole. At the left and the right border, the half parts of the jets of the staggered hole lines can be found. Here, the cooling effectiveness is defined as

$$\eta = \frac{T_{g,1} - T}{T_{g,1} - T_c},$$  \hspace{1cm} (21)$$

with the static cooling fluid inlet temperature $T_c=723.15$ K and the static hot gas inlet temperature $T_g=1523.15$ K. In the lower part of the cutting plane the grey
The coloured area indicates the region of the thermal barrier coating (TBC). In Fig. 24a the results in the three cutting planes for configuration C45-10 are shown. Here, clearly the lift-off of the cooling jets can be seen. There is no development of a cooling film, but the jets of the particular lines of holes can be distinguished. Thus, a strong hot gas underflow of the cooling jets has to be stated. The reduction of the distance of the rows of cooling rows from 10 to 6 hole diameters leads to a lower lift-off of the cooling jets (Fig. 24b). Due to the shorter distance, the mixing of the jets of row #3 has developed less in configuration C45-06 than in configuration C45-10, which can be seen in the increased cooling effectiveness in the core of the jet. Beyond that, the hot gas underflow is reduced, too. Nevertheless, single cooling jets can be attached to the lines of holes and a full cooling film coverage is not existent.

For configuration C30-06 (Fig. 24c), the reduction of the inclination angle \( \alpha \) leads to a significant reduction of the hot gas underflow of the cooling jet. This is caused by the reduced penetration of the main flow by the cooling gas. Hence, the cooling jet expands more in the lateral direction (\( z \)-direction). The hot gas contact of the surface is reduced and, thus, lower thermal loads of the TBC exist.

The configurations S30-06-F and S30-06-FL with shaped holes show a completely different structure of the cooling film. Here, the development of a homogeneous cooling film on the flat plate surface can be stated. Thus, the thermal load of the TBC is reduced by approximately 40%. In configuration S30-06-F (Fig. 24d), nearly no lift-off of the cooling jets is found. The isolines for \( \eta = 0.0471 \) are very smooth and the penetration of the hot gas is reduced significantly. The concentration of the isolines directly upon the cooling jet of hole #4 shows that the mixing of the cooling gas and hot gas takes place at some distance to the flat plate surface. The expansion of the cooling jet in the lateral direction is increased in comparison to configuration C30-06 so that there is nearly no contact of hot gas to the TBC surface. Thus, higher cooling effectiveness in the solid can be achieved with a lower cooling gas mass flow.

In configuration S30-06-FL (Fig. 24e), the expansion of the outlet shape in the lateral direction downstream of the hole leads to an additional expansion of the cooling jet in comparison to configuration S30-06-F. The penetration of the hot gas by cooling jets is smaller than in configuration S30-06-F.

b) Secondary flow analysis

In Fig. 25, the secondary flow vectors in the region of the cooling film are shown. The three cutting planes for each configuration are placed at the same positions as the plots of the cooling effectiveness distribution. The size of the vectors is scaled in all configurations by the same value, so that the same vector length represents the same velocity in every cut.

In configuration C45-10, the well-known \( \Omega_1 \) vortex system (kidney vortex) that is always found in the case of jets in cross-flow can be clearly detected (Fig. 25a). The sequence of the three cutting planes shows the displacement of the cooling jet in the normal direction and, thus, the hot gas underflow. The core of
Figure 24: Cooling effectiveness distribution in cooling film.
the cooling jets of hole #3, which should be found at the left and right borders of the picture, are placed out of sight. Therefore, a strong cross flow in the lateral direction can be found adjacent to the $\Omega_1$ structure of hole #4.

Configuration C45-06 (Fig. 25b) shows in the first cut a smaller expansion of the vortex system in the normal direction. Due to the reduced hole distance, the lift-off of the $\Omega_1$ system of hole #2 is reduced in comparison to configuration C45-10 so that the $\Omega_1$ structure of hole #4 is strengthened and a significant additional lift-off of the jet is found in the cuts downstream. On the other hand, the displacement of the cooling jet of hole #3 is reduced so that the core of the kidney vortices is found at a height of 70% at position #1 to 85% at position #3, respectively. Due to the expansion of the cooling jets in the lateral direction, the cross-flow of the hot gas is reduced, which leads to a decreasing jet lift-off.

For configuration C30-06 (Fig. 25c), both the increased expansion of the cooling jets in the lateral direction and the reduced lift-off can be clearly detected. In the lateral direction, the cores of the kidney vortex of hole #4 are displaced to a greater distance from the middle axis of the hole. This leads to a reduced hot gas underflow. Beyond, the intensity of the $\Omega_1$ vortex of hole #4 is reduced in comparison to configuration C45-06. This is caused by the reduced inclination angle $\alpha$ in this configuration and thus the reduced component of the cooling gas velocity in the normal direction. Therefore, the displacement of the cooling jet in the normal direction is reduced and a better protection of the plate surface is achieved.

The configurations with shaped holes, again, show a completely different flow structure. Due to the shaping, cut #1 is placed in the outlet area of hole #4. Therefore, kidney vortices can only be found in the cutting plane positions further downstream. For configuration S30-06-F (Fig. 25d), only a small $\Omega_1$ vortex system can be detected. The increase of the outlet area leads to a significant reduction of the cooling gas velocity component in the normal direction. Due to the lateral expansion of the hole, the centres of the kidney vortex are displaced far from the midplane of the holes (cut #2, Fig. 25d). Only a small region of the flow channel is penetrated by the cooling gas. A reduced mixing of cooling fluid and hot gas can be stated. In cut #3 almost no vortex system can be found, but a small velocity component in the normal direction has to be stated.

In Fig. 25e, the vector plots for configuration S30-06-FL are shown. In cut #1, the additional velocity component of the cooling gas in the lateral direction can be seen clearly. This is caused by the angle $\beta_2$ for this shaped outlet region (see Table 7). Due to this lateral expansion of the hole shape, in cut #2 the kidney vortices can only be detected with difficulty. The legs of the $\Omega_1$ vortex systems of the preceding rows of holes are positioned nearly at the same place as the vortices induced by hole #4. Therefore, the counter-rotating vortices eliminate each other. This leads to an additional reduction of the vortex intensity. The displacement of the main stream by the cooling film is reduced, too. Thus, a very good coverage of the flat plate with cooling fluid can be stated.
cutting plane 1 2 3

a) configuration C45-10

b) configuration C45-06

c) configuration C30-06

d) configuration S30-06-F

e) configuration S30-06-FL

Figure 25: Secondary flow vectors in cooling film.
4.4 Simulation of real blade cooling

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 of detecting possible deficiencies in the cooling design, e.g. hot spots, as early as possible.

As has been shown, conventional design processes rely on empirical 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. It has also been shown that the conjugate calculation technology offers a significant potential for improving the cooling design process. 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 [49] is applicable for modern gas turbine cooling simulation and can be used for the successful numerical testing of real cooling configurations.

4.4.1 Test configuration for blade with film cooling

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 [105]. 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. 26), 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. 26. The leading edge is supplied directly by a separate cooling channel (no. I) whereas the other two channels are typically serpentine shaped. 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
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.

### 4.4.2 Numerical models for conjugate cooling simulations

In the fluid flow regions, the Navier–Stokes equations have been solved for these numerical models including rotational effects and turbulence modelling by application of the Baldwin–Lomax model. In the solid blocks, heat conduction is calculated by solving the Fourier heat conduction equation numerically. Direct coupling of the solid body and the fluid flow regions is established by the conjugate procedure using the balance of the local heat fluxes.

![Figure 26: Test configuration for real blade cooling.](image)
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, 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 27 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

Figure 27: Illustration of the blade leading edge model for the CCT.
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 supply channels other 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 neglecting the leading edge cooling, the thermal load in the front part of the blade is calculated as 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.

4.4.3 Results for cooling simulation of the leading edge

Figure 28 shows the vector plots in a radial cutting plane for the leading edge ejection under design and off-design conditions. Figure 28a demonstrates that under design conditions the cooling fluid ejected by row “P1” is distributed along the blade pressure side as supposed. In Figure 28b, it can be seen 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 Figure 29, 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 the Akashi R&D Center. A comparison of the experimental data with the numerical results for the pressure-side part of the leading edge is...
Black lines in the numerical result indicate block boundaries of the structured multi-block grid.

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 found to be 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 were 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

---

Figure 30: Comparison of thermal index paint experiment and conjugate calculation of the leading edge region (pressure side).
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 of the overestimated off-design flow angle. Somewhat higher temperatures in the hub region of the calculation 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.

4.4.4 Results for cooling simulation of the blade tip region

The conjugate calculation results for the trailing edge region correspond very well to the measurements. Figure 31 shows that in particular the high thermal load levels:

![Figure 31: Comparison of thermal index paint experiment and conjugate calculation of the upper part of the suction side.](image)

- high
- medium
- low
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 uncooled leading edge is obvious in the front part of the calculation
results. With respect to the small region E at the tip of the leading edge, this
region has also been found in the calculations for task 1. The origin of region E is
hot gas from the main flow that is able to flow into the radial gap at this position.

Figure 32 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.

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.
5 Heat transfer analysis of a complete system: Turbocharger

In recent years the turbocharging of diesel engines has gained increased importance. Considering mainly performance perspectives the present research and development efforts also focus on economic and environmental aspects. The specific fuel consumption and the exhaust emissions need to be reduced and the dynamic behaviour of the engine needs to be improved.

For these considerations the unsteady behaviour of the whole engine is of interest, i.e. the unsteady interactions between the individual engine components (core engine, compressor and turbine of the turbocharger, exhaust gas recycling, control systems). Common design tools are computer programs for unsteady cycle-analysis computations, which receive information about the components' operating behaviour from maps.

Bulaty [106] published a method for the extrapolation of component maps that is still used in programs for cycle analysis computations. Since the heat fluxes are neglected, the results are inaccurate and the understanding and the quantification of the heat fluxes becomes crucial. Thus, the quantification of the heat fluxes occurring is one condition for the adoption of various component efficiencies. The designer has to distinguish the adiabatic from the diabatic change of state and efficiency, respectively. Since this has already been stated by various researchers [107–109], the scope of this chapter is a contribution to overcome these deficiencies by a conjugate heat transfer calculation of the turbulent flow in a turbocharger.

5.1 Heat transfer model

The first step is the setup of an adequate model that describes the heat fluxes in a turbocharger. Figure 33 shows the developed model that has been divided into the turbine, the center housing, and the compressor. Balancing these components clarifies the paths of heat and power.

One part of the enthalpy drop in the turbine is the power output transferred to the single shaft that drives the compressor wheel. The mechanical efficiency describes the amount of power that is converted into heat and taken away by the lubricating oil that also has a cooling function. The other part of the enthalpy drop in the turbine is heat. It divides into one amount transferred to the
surrounding area due to natural convection and thermal radiation and one flowing into the centre housing where it splits again. Additional to the portion leaving the system due to natural convection and radiation one portion is transferred to the oil channel. The rest of the heat enters the compressor where the amount, which is not delivered to the surrounding area, can heat up the compressor fluid depending on the operating point. The latter is the main objective of this investigation.

5.2 Computational model

5.2.1 Geometry and grid

Since the modelling of the complete turbocharger would require approximately 40 million grid points, a less time-expensive geometric model has been developed (Fig. 34). First, the compressor and the turbine are divided into periodic segments. Secondly, due to the necessary periodic boundary conditions at the cutting planes, the volutes are substituted by annular rings. The 60° compressor segment consists of 2.9 million grid points including 2 splitter blades and the 32.7° turbine segment is composed of 1.45 million grid points and 1 blade. The intermediate central housing is not segmented because of its asymmetry. Thus, the inner oil-cooling system could be considered in a simplified manner. In total these three parts of the turbocharger required 4.5 million grid points distributed among 151 blocks.
5.2.2 Boundary conditions

Two types of conjugate calculations have been conducted. One configuration is the so-called full model including the compressor, the center housing, and the turbine. The parametric study required a time saving model. Hence, the single compressor was taken from the complete model reducing the computational grid and with that the computation time.

The same car turbocharger has been investigated experimentally. From these experiments the boundary conditions for the calculations have been deduced for various operating points. This includes the aerothermal boundary conditions for the fluid at the inlet and at the outlet of compressor and turbine as well as the thermal boundary conditions on the surface of the casing.

In order to achieve realistic boundary conditions on the surface of the casing, pictures have been taken with a thermography camera. Combined with the information from resistance thermometers a surface temperature distribution has been determined. This procedure implicitly contains the influence of natural convection as well as the influence of thermal radiation.

Since the turbocharger consists of different materials for the casings, blades, and shaft, their different thermal behaviour is taken into account by setting the corresponding material properties, like density and thermal conductivity.

Figure 34: Geometric model.
a) Full model (compressor, casing, turbine)

One operating point has been simulated, which serves as a reference point \( \left( \frac{m_c}{m_{C,ref}} = 1, \frac{\vartheta_{oT,1}}{\vartheta_{oT,ref}} = 1, \frac{n}{n_{ref}} = 1 \right) \) for comparison of the different calculations. The number of revolutions is identical for turbine and compressor but also the mass flows have been adjusted to the same value in experiment and calculation.

b) Compressor model

The single compressor needs further information at the boundary towards the center housing. Here, a special temperature boundary condition has been applied that takes into account the actual operating point. A parametric study for the compressor model with variation in the turbine inlet temperatures and increased mass flows has been carried out.

The solid body temperature distribution at the surface of the casing, the blades, and the shaft is given in Fig. 35. At the outer walls of the casing this temperature distribution is identical with the boundary condition received from the experiment as explained in the preceding section. Furthermore, the visualisation of the multiplied compressor and turbine segments illustrates the shape of the geometric model with the modified volutes.

5.3 Results

5.3.1 Heat flux distribution

Figure 36 shows the temperature distribution of the full model computation in a
planar cut along the rotor axis. At the same time it represents the heat flux from
the turbine through the center housing into the compressor. Furthermore, at this
operating point the temperature difference between the fluid and the solid body
of the compressor comes out clearly. The detail with a different temperature
scale reveals this issue.

In order to evaluate the heat fluxes between the fluid and the solid body
(casing, blades, and shaft) of the compressor, 60 cutting planes have been
positioned equidistantly according to Fig. 37. After that, the local heat fluxes
have been determined and assigned to the individual cutting planes.

Figures 38–40 show the summation of the local heat transfer between the
fluid and the solid body along the planes specified in Fig. 37. The local values
are added subsequently, so that a particular value stands for the amount of heat
transferred from the compressor inlet to the respective position. If the gradient of
the curve is positive the heat flux is directed by definition from the solid body
into the fluid and vice versa if the gradient is negative.

For all investigated cases the heat transfer is directed into the fluid in the first
half of the compressor. Further downstream, the fluid heats up as a result of the
compression process to such a high extent that its temperature exceeds the solid
body temperature. Consequently, heat flows from the fluid into the solid body in
the rear part of the compressor. This phenomenon is intensified with rising mass flow because, on the one hand, the compression and with that the fluid temperature rise increases. On the other hand, the expansion in the turbine grows, so that less heat is transferred to the compressor. Increasing the turbine inlet temperature at constant mass flow also leads to an intensified heat transfer from the fluid to the solid body. The reason comes from a sequence of changes. First, the enthalpy drop in the turbine increases and consequently the number of revolutions, too. With the same mass flow as in the turbine, the compressor-pressure ratio increases with its growing enthalpy rise. In the end the higher exit pressure implies a higher exit temperature.
5.3.2 Heat transfer number

Dependent on the operating point the heat transfer inside the compressor may be represented by:

\[ Q_c = f(\mu_{C,1}, \lambda, \dot{m}_c, T_{f,1}, D_{C,1}, D_{C,2}, H_{C,1}, H_{C,2}) \]  

This is a function of the working fluid, the material, the aerothermodynamics, and the geometry. Further magnitudes of influence are dependent magnitudes and have therefore to be neglected. In non-dimensional notation the equation is:

\[ Nu_c = f(Re_c, T_{o,c,1}) \]  

Based on the definition for the Reynolds number and taking into account the compressor mass flow and blading geometry, the following equation for the Reynolds number can be obtained.

\[ Re_c = \frac{\rho_{C,1} \cdot c_{C,1} \cdot l_c}{\mu_{C,1}} \cdot \frac{\dot{m}_c}{\dot{m}_{C,1}} \cdot \frac{D_{C,2} - D_{C,1} + 2 \cdot H_{C,1}}{4 \cdot D_{C,2} \cdot H_{C,2}} \]  

The measures for the geometry can be found in Fig. 37. The computer code calculates the heat fluxes and subsequently a representative heat transfer coefficient for the global heat transfer into the compressor flow of the turbocharger, which is needed for the calculation of the Nusselt number:

\[ Q_c = \bar{h}_c \cdot \Delta T_c \cdot A_c \]  
\[ \Delta T_c = T_{o,c,1} - T_{o,C,1} \]  
\[ A_c = \frac{\pi}{2} \left( D_{C,2} - D_{C,1} + 2 \cdot H_{C,1} \right)^2 \]  
\[ Nu_c = \frac{\bar{h}_c \cdot l_c}{\lambda} \]  
\[ \lambda = \lambda \left( T = 0.5 \cdot (T_{C,1} + T_{f,1}) \right) \]  
\[ l_c = \frac{\pi}{4} \left( D_{C,2} - D_{C,1} + 2 \cdot H_{C,1} \right) \]

This compressor Nusselt number has been calculated for each operating point and is plotted versus the compressor Reynolds number as shown in the diagram (Fig. 41). To enable the Nusselt number prediction for additional operating points, the following calculation specification has been developed:

\[ Nu_c = Nu_0 - \sqrt{p^2 - \left( \frac{Re_c^2}{1 - \varepsilon^2} \right)} \]

This function best describes the curve through the points of constant turbine inlet temperature. It is derived from the vertex equation describing conical sections. It
Table 8: Constants and coefficients of Nusselt law.

<table>
<thead>
<tr>
<th>coefficient</th>
<th>equation for coefficient</th>
<th>constant</th>
<th>value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$e^2$</td>
<td>$e_0 \cdot e^{-\epsilon_1\theta_T'/\theta_{T,ref}} + e_2 \cdot \theta_T/$</td>
<td>$e_0$</td>
<td>$5 \times 10^5$</td>
</tr>
<tr>
<td></td>
<td>$\epsilon_1$</td>
<td>$143$ K</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\epsilon_2$</td>
<td>$1123$ K$^{-1}$</td>
<td></td>
</tr>
<tr>
<td>$Nu_T$</td>
<td>$p$</td>
<td>$170$</td>
<td>$100$</td>
</tr>
</tbody>
</table>

Figure 41: $Re_C - Nu_C$ – diagram.

Figure 42: Compressor: $m_C - Q_C$ – diagram.
contains the Reynolds number and a temperature-dependent coefficient, as well as several constants as listed in Table 8. Inserting \( \varepsilon \) and eqn (24), the notation for the Nusselt number is:

\[
Nu_C = Nu_0 - \left( p^2 - \frac{\left( \frac{\dot{m}_c}{\mu_{c,1}} \cdot D_{c,2} - D_{c,2} + 2 \cdot H_{c,2} \right)^2}{1 - \left( e_0 \cdot e^{-\varepsilon_{a,1}/\varepsilon_1} + e_2 \cdot T_{a,1} \right) } \right). \tag{32}
\]

For all temperatures, the curves in the \( Re_C - Nu_C \) diagram start from a common starting point and drop with rising Reynolds numbers and with rising mass flows, respectively. According to the explanations in the preceding paragraph the drop becomes larger with higher turbine inlet temperatures.

To show the relation between the compressor Nusselt number and the heat flux the following function for the non-dimensional heat flux is derived from eqn (25), (28), and (31) and is plotted versus the compressor mass flow in Fig. 42:

\[
Q_C = \frac{\bar{F} \cdot A_c \cdot \Delta T_p}{l_c} \cdot \left( Nu_{c0} - \left( p^2 - \frac{\left( \frac{\dot{m}_c}{\mu_{c,1}} \cdot D_{c,2} - D_{c,2} + 2 \cdot h_{c,2} \right)^2}{1 - \left( e_0 \cdot e^{-\varepsilon_{a,1}/\varepsilon_1} + e_2 \cdot T_{a,1} \right) } \right) \right). \tag{33}
\]

The Nusselt number inside the curly brackets is multiplied by a factor. This factor only depends on the turbine inlet temperature, so that it is constant for each curve. With a rising turbine inlet temperature different phenomena occur. At operating points with small compressor mass flows the heat flux into the compressor fluid increases, but at high mass flows, the heat flux out of the compressor fluid grows. This results in an operating point with a heat flux independent of the turbine inlet temperature.

Finally, discussing the special case of \( Nu_c = 0 \), which almost coincides with one, the simulated operating point for high temperature at \( Re_c = 1.78 \times 10^5 \) in Fig. 41, helps to understand the heat transfer processes in the compressor. Referring to Fig. 33, it does not mean that no heat is transferred and the change of state is identical with the adiabatic case. Instead, the correct interpretation is that the same amount of heat is transferred to the fluid as it is discharged off the fluid. The change of state is not adiabatic because of the diverging isobars as illustrated in Fig. 43. For clarity reasons, the velocities have been neglected in this diagram. Furthermore, Fig. 42 reveals this fact for the same operating point. In the front part of the compressor, a significant amount of heat is transferred into the fluid and in the rear part the heat is transferred back into the solid body. Altogether, the total amount of heat transferred to the compressor flow for this operation point is only slightly positive and close to zero as shown in Fig. 42.
6 Conclusions

Turbulent heat transfer is of significant importance for the design of convectively cooled and film-cooled components in gas turbines. To improve the design processes and to develop advanced design tools a better understanding of the complex heat transfer phenomena in flow transition regions and in internal cooling passages is still necessary. Furthermore, the inter-relating effects of heat transfer and fluid flow have also to be taken into account for a more accurate prediction of the thermal load. Here, the conjugate calculation technique offers a great opportunity for a revolutionary design process instead of using the conventional heat transfer strategies. It has been shown that this numerical prediction technology is today applicable to the simulation of real-blade cooling configurations for modern gas turbines. Due to the early and reliable detection of design deficiencies, the conjugate strategy helps to reduce time and costs of the design process.

Furthermore, unforeseen influences of heat transfer from hot turbine parts to cold regions can lead to reduced aerodynamic efficiencies or might cause problems in rotating cavities. Therefore, a complete heat transfer analysis by the conjugate calculation technology of a system becomes helpful for the understanding of local heat transfer phenomena and their effects on the complete system. By application of the conjugate calculation technology a complete turbocharger for a passenger car has been analysed extensively for a variety of operating points. As a result, it has been possible to deduce a one-dimensional heat transfer relationship, which enables the prediction of the turbulent heat transfer even for additional operating points and similar geometries.
Acknowledgements

The authors are very thankful for the strong support by Mr. Takao Sugimoto and Mr. Ryozo Tanaka from Kawasaki Heavy Industries, LTD., for the study presented in Section 4.4. The support of this study by Kawasaki Heavy Industries, LTD., is gratefully acknowledged.

With respect to their contributions of numerical results in Section 4.3 and Section 5 the authors wishes to thank Mr. Norbert Moritz and Mr. Tom Heuer from the Institute of Steam and Gas Turbines, RWTH Aachen (Aachen University).

The research project upon which Section 3.2 is based is sponsored by the German Bundesministerium für Wirtschaft und Technologie (grant no. 0326855). The responsibility for the content of the section lies with the authors.

With respect to Section 4.3 the authors gratefully acknowledge the financial support of the Deutsche Forschungsgemeinschaft (DFG) within the Collaborative Research Center (SFB) 561 “Thermally Highly Loaded, Porous and Cooled Multilayer systems for Combined Cycle Power Plants”. The responsibility for the content of the section lies with the authors.

Section 5 of this chapter is the scientific result of a research project, which was initiated and funded by the Forschungsvereinigung Verbrennungskraftmaschinen e.V. (FVV, Frankfurt, Germany) and carried out at the Institute of Steam and Gas Turbines at the RWTH Aachen headed by Professor Dr.-Ing. D. Bohn and accompanied by a counseling circle headed by Dr. P. Prenninger, AVL List, Graz, Austria.

References


