Film-Cooling from Holes with Expanded Exits: 
A Comparison of Computational Results with Experiments

D. Giebert, M. Gritsch, A. Schulz, and S. Wittig
Lehrstuhl und Institut für Thermische Strömungsmaschinen
University of Karlsruhe (TH)
76128 Karlsruhe, Germany

Abstract
A 3D Navier-Stokes code, together with the standard k-ε model with wall function approach, was used to investigate the flowfield in the vicinity of three different single scaled-up film-cooling holes. The hole geometries include a cylindrical hole, a hole with laterally expanded exit, and a hole with forward-laterally expanded exit.

Comparisons of numerical results with detailed flowfield measurements of mean velocity and turbulent quantities are presented for a blowing ratio and density ratio of unity. Additionally, experimental data for different blowing ratios and a density ratio of about two are taken to perform validation of the code for adiabatic film-cooling effectiveness prediction.

Results show that for both the round and the expanded hole geometries the code is able to capture all dominating flow structures of this jet in crossflow problem. However, discrepancies are found when comparing the flowfield inside the hole and at the hole exit. In particular, jet location at the hole exit differs significantly from measurement for the expanded hole geometries. For the adiabatic film-cooling effectiveness, it is shown that for round and expanded hole exits the intensity of the shear regions and the source of turbulence, respectively, have a strong influence on the predictability of the numerical code.

1 Introduction
Nozzle guide vanes and turbine blades are subjected to very high thermal loads in modern gas turbines operating at high turbine inlet pressures and temperatures. Accordingly, an efficient cooling of these components and the endwalls is required to prevent an early distress. Film cooling is widely used to maintain vane and blade temperatures at acceptable levels. Due to its big potential for improvements this cooling technique is subject of numerous experimental as well as computational research.

Various investigations on discrete-jet film-cooling are found in the literature which help to understand the major phenomena. However, in terms of numerical computations, many of these studies concentrate on film-cooling fluid mechanical parameters, such as jet-to-mainstream blowing ratio and momentum flux ratio, for round film-cooling holes inclined at both nominally 30° and 90° (Baldauf and Scheurlem (1996), Walters and Leytek (1996). He et al. (1995), Garg and Gaugler (1993), Jansson and Davidson (1994), Demuren (1993), Alvarez and Jones (1993), Leytek and Zerkle (1994)). It is well known that significant improvement can be achieved in terms of better cooling characteristics of the film by using cooling holes with expanded exits (Goldstein et al. (1974), Makkii and Jakubowski (1986)). Recent investigations have given some physical insight into the film-cooling qualities when applying contoured holes. But the results reported in the open literature are gained only by experimental studies. Therefore it is still a need for a numerical approach to the problem of describing film-cooling characteristic using holes with expanded exits. This paper presents a comparison of numerical calculations with flowfield measurements for three single, scaled-up, cooling hole geometries. The holes under consideration are a baseline round hole, a hole with a laterally expanded exit, and a hole with forward-laterally expanded exit. In addition, for the round hole geometry and the hole with forward-laterally expanded exit, a more detailed investigation of the flowfield and adiabatic film-cooling effectiveness is taken into consideration. Thus far, there are no reported detailed comparison between calculation and experiment for hole geometries with expanded exits using a 3D Navier-Stokes solver with standard k-ε turbulence model. The computations enable an assessment of the numerical method for practical design calculations.

2 Experimental setup
Experiments were conducted in a continuous flow wind tunnel at the Institut für Thermische Strömungsmaschinen (ITS) at the University of Karlsruhe. In the following, a brief description of this film-cooling test section will be given. Details of the experimental setup, the measuring techniques applied, and measurement uncertainties are described in Wittig et al. (1996), Thole et al. (1996), and Thole et al. (1997).

2.1 Test section and hole geometries
A sketch of the test section is shown in Fig. 1. The test section was specially designed for measurements of mean and turbulence characteristics of the flow in the nearfield region of a scaled-up film-cooling hole on a flat test plate. Mainflow and coolant flow channel are forming the test rig to simulate the flowfields associated with external and internal blade cooling.

The measurements allow detailed investigations of the effects of flowfield and geometrical parameters on discharge coefficients, near hole velocity field, adiabatic wall effectiveness, and heat transfer coefficients. Flowfield measurements were conducted by means of a two-component laser Doppler velocimeter. A two-dimensional distribution of the tem-
perature on the plate surface was provided using an infrared camera system.

Tests were carried out using single scaled-up film-cooling holes. The hole geometries (one cylindrical and two shaped holes, subsequently denoted as fanshaped and laidback fanshaped hole) referred to in this paper are shown in Fig. 2. These hole geometries were obtained in cooperation with several European gas turbine companies. The diameter of all three holes at the inlet is 10mm. The holes are inclined at 30° giving a length-to-hole diameter ratio of L/D = 6. The lateral expansion angle, which starts two hole diameters downstream of the hole inlet for the both contoured holes, was 14° resulting in a hole width of 30mm at the hole exit. The forward expansion angle of the laidback fanshaped hole was 15°. The length of this hole extended four hole diameters at the hole exit.

3 Numerical procedure

3.1 Governing equations and turbulence modelling

The equations used to model the flow are the steady Reynolds-averaged conservation equation of mass, momentum, and total energy together with the equation of state. In this study a thermodynamic ideal gas with the equation of state. In this study a thermodynamic ideal gas is assumed. The turbulence is taken into account by the standard $k$-$\varepsilon$ turbulence model (Lauder and Spalding (1974)) which allows the calculation of the eddy viscosity $\mu_t$. Throughout this study, a constant turbulent Prandtl number ($Pr_t = 0.9$) is employed limiting the closure model on the specification of the turbulent viscosity.

3.2 Numerical method

The current numerical study was performed using a three-dimensional Navier-Stokes code. The capabilities and performance of this kind of code have been demonstrated recently for a wide range of technical flow configurations (Benz et al. (1993), Kurreck and Wittig (1994) including heat transfer phenomena (Griebert et al. (1996)). In this CFD code, the governing equations are formulated in a body-fitted non-orthogonal curvilinear co-ordinate system. A fully conservative finite volume method is applied for the spatial discretization. The transport equations for the Cartesian velocity components and other scalars are solved using a non-staggered structured grid. All flow variables are stored at the same nodes ("cell-centered"). The solution procedure is iterative and the pressure-based SIMPLE algorithm of Patankar and Spalding (1972) is used to derive a pressure-correction equation. To avoid checkerboard pressure oscillations a similar momentum interpolation to that proposed by Rhie and Chow (1983) is applied. For the discretization of the diffusive terms, a second order central difference scheme is used, whereas the convective terms of all transport equations are discretized by the second order accurate Monotonized-Linear-Upwind (MLU) scheme (Noll (1992)). The system of the algebraic equations is solved by a generalized conjugate gradient iterative procedure (CG) in conjunction with an incomplete lower-upper decomposition (Noll and Wittig (1992)).

3.3 Computational grid

For the numerical calculation the solution domain was extended to the whole test facility (see Fig. 1) including coolant channel with its inlet nozzle and outlet diffuser as well as the mainflow channel with the connecting hole. The inlet and outlet boundaries of the mainflow channel are located 42 diameter upstream and 14 diameter downstream of the hole exit, respectively. At these positions boundary conditions (see paragraph 3.4 below) from experiment were available. For the calculation, the coolant channel was taken into consideration because previous experimental investigations (Thole et al. (1997)) revealed that the flow field inside the hole is strongly affected by the way the coolant is supplied to the film-cooling hole. Computational H-type grids were generated for the cylindrical, fanshaped and laidback fanshaped hole geometry using the multi-block technique. A total of $\approx 158000$ grid points was used for each grid (mainstream channel $\approx 125000$, cooling hole $\approx 8000$ and coolant channel $\approx 25000$ grid points).

Fig. 3 shows a cross-sectional view along the longitudinal symmetrical axis of the computational grid for the cylindrical hole together with an enlarged view of the hole exit. The symmetry of the problem was exploited to reduce computing time and memory.

3.4 Boundary conditions

At the inlet boundaries total pressure, total temperature and the flow angle are specified. These values are derived from experiment. The inlet turbulent kinetic energy $k$ is given from the experimental inlet turbulence intensity ($2\%$ for the mainstream channel and $1\%$ for the coolant channel, respectively). Its dissipation rate $\varepsilon$ is calculated assuming a turbulent length scale of about $5\%$ of the channel height.

At the outlet the measured static pressure is prescribed and the gradient of the other flow variables in flow direction is set to zero. Symmetrical boundary conditions were specified at the lateral plane of symmetry.

Along the wall, the velocities satisfy the no-slip condition. The wall function approach (Lauder and Spalding (1974)) is used to bridge across the near wall region. The first grid nodes were well placed in the fully turbulent region with an average dimensionless distance of $y^+ \approx 40-90$.

A zero heat flux normal to the wall boundary is specified as thermal wall condition in the energy equation.

4 Results and discussion

Comparison of calculated results with experiments for all three hole geometries is presented. All flow cases shown correspond to a Mach number of $Ma_c = 0.3$ in the coolant channel and $Ma_m = 0.25$ in the mainstream channel, respectively. For the calculations, the flow through the film-cooling hole was not taken from the experiment, but resulted from the calculated pressure distribution across the hole. The computed flow rates in the mainstream and coolant channel, as well as through the film-cooling hole, were very close to the measured data. The calculated blowing ratios showed a maximum deviation of about $5\%$ from the measured values.

The following sections present the mean and turbulent flowfield results and some results for the adiabatic film-cooling effectiveness. Unless otherwise noted, the origin of the co-ordinate system is located at the center of the ellipse of the cylindrical hole exit in the mainstream channel (see Fig. 2).
4.1 Flowfield comparison

For the mean and turbulent flowfield results presented in this section, the blowing ratio M and density ratio DR are of unity.

Streamwise and vertical velocity components are normalized by the reference velocity $u_{ref}$ and $v_{ref}$, respectively. $u_{ref}$ corresponds to the value at a position $x/D = -2$ and $y/D = 2$ in the mainflow channel while $v_{ref}$ is a total jet velocity based on inlet hole diameter and coolant mass flux.

4.1.1 Flowfield in the jet centerline plane

Figs. 4 - 6 show the comparison of the streamwise (left) and vertical (right) velocity component with the measured data along the jet centerline plane in the mainstream channel for the cylindrical, fanshaped, and laidback fanshaped hole geometry, respectively.

As can be seen in Fig. 4 for the cylindrical hole geometry, both the $u$- and $v$-velocity are well predicted by the calculation in the up- and downstream region of the hole exit. Moreover, the location of the calculated jet exit in the hole is in good agreement with the experimental result, which shows a jet detached from the bottom wall of the mainstream channel. This detail of the flowfield is also picked up by the simulation. However, the penetration height of the jet is slightly underestimated by the calculation.

In contrast to the result for the cylindrical hole, the contoured hole geometries show a weak interaction of the exiting jet with the crossflow. The jet remains attached at the bottom of the mainstream channel (Figs. 5 and 6). As expected from the results presented above for the cylindrical hole, the calculations for the expanded hole geometries agree very well with the measurement for the $u$- and $v$-velocity in the farfield region of the cooling hole. However, differences occur between calculated and measured data in the nearfield of the hole exit for both the fanshaped and the laidback fanshaped hole: as can be seen from Figs. 5 and 6, the streamwise velocity is overestimated by the calculation in the leading edge region of the cooling hole ($-0.5 < x/D < 0$), whereas at the downstream portion of the hole exits, the position is reversed. Also, an overestimation was found for the normal velocity component. This concludes that, in contrast to the experiment, the jet exiting the hole is shifted towards the upstream edge in the computation. As a consequence, fluid motion from the mainstream channel into the cooling hole at its leading edge, which is indicated by the measurement for the laidback fanshaped hole in the jet centerline plane (Fig. 6), is not predicted by the calculation.

The differences in the location of the exiting jet between calculation and measurement for the contoured hole geometries show large dissimilarities in fluid motion inside the contoured portion of the cooling hole. A more detailed analysis on this phenomenon will be given below.

Figs. 7 - 9 show the comparison of the calculated $(Tu = 100 \times \sqrt{2k/3(u_{ref})})$ and measured $(Tu = 100 \times 0.5(u^2 + v^2)/u_{ref})$ turbulence intensities along the jet centerline plane for all three hole geometries.

For the cylindrical hole (Fig. 7), good agreement is obtained in the upstream region of the hole as well as for axial locations $x/D > 6$. At the leading edge of the hole exit the quantitative distribution of the intensity is also well captured. However, the high turbulence levels downstream the hole exit are significantly underestimated by the calculation by two orders of magnitude. This deviation is partly rooted in a too coarse computational grid in this region so that a finer grid may probably improve the computed results. But even for a very fine grid resolution a significant underprediction of turbulence will be revealed when applying the standard $k-c$ model as shown e.g. in the studies of Alvarez and Jones (1993) and Claus and Vanka (1990). They investigated a perpendicular jet in crossflow situation with a round hole geometry using very fine grids and found that the $k-c$ model is not appropriate for highly anisotropic mean strain field downstream of the jet exit.

In Figs. 8 and 9 the calculated contour lines of turbulence intensity are compared with experimental data for the fanshaped and the laidback fanshaped hole, respectively. Due to the weak interaction of the exiting jet with the crossflow, the turbulence levels are much less as for the round hole geometry. This leads to reasonable overall agreement between calculation and experiment except for the region at the hole exit. In this area, the contour levels are seriously underpredicted by the calculation by approximately a factor of two. Unlike the cylindrical hole, the concentration of high turbulence at the hole exit indicated by the measurement and the moderate turbulence level downstream of the hole exit suggest that the flow inside the hole is the dominant source for turbulence production for the contoured holes rather than the jet-crossflow interaction (Thole et al. (1996)). This production is not adequately picked up by the simulation. Reasons for this may be attributed to an insufficient resolution of the computational grid in the cooling hole, especially for the contoured portion of the hole and the application of the standard $k-c$ turbulence model.

The results shown before indicate, that the fanshaped and laidback fanshaped holes are quite similar in terms of mean velocity field and turbulence intensities for both the experiment (see Thole et al. (1996)) and calculation. Moreover, deviations between calculation and measured data show the same trend for these hole geometries caused by the same mechanisms. Therefore, in the following only the results for the cylindrical and laidback fanshaped hole geometry will be shown.

4.1.2 Details of the flow at the cooling hole

In the following, results will be presented which refer to cross-sectional cuts located at $y/D = const$ in both the mainstream channel and the coolant channel. Unless otherwise noted, the direction of the fluid flow in the channel where the cut is located is from bottom to top. For vertical-spanwise cuts inside the cooling hole, the crossflow direction in the mainstream channel is normal to the plane shown.

The calculated $u$-velocity component for the round hole geometry at the hole exit is shown in Fig. 10. The considered cross-sectional cut is located at $y/D = 0.1$ in the mainflow channel. The contour of the hole exit is indicated by the black ellipses. As indicated by this figure, reasonable agreement is obtained for this velocity component. The maximum of the velocity is quantitatively well captured by the calculation but differences occur in the shape of the velocity profile in the centre of the hole exit. However, like the measurements, the computation reveals a jet exiting the hole at its trailing edge ($z/D = 1$).

In order to analyse the mechanisms causing the deviations between calculation and experiment for the round hole geometry the predicted $u$-velocity component is compared with the measured data at the hole inlet (Fig. 11). The cross-sectional plane is located 1.5mm beneath the ceiling of the cooling channel. Remarkable similarity is obtained, which concludes that the jet entering the hole in the cooling channel is very well predicted by the numerical code. Therefore, a wrong prediction of the jet entering the cooling hole in the coolant channel can be excluded as source for the deviations observed at the hole exit. The differences between measured and calculated velocities observed at the hole exit are attributed to an inadequate capturing of the secondary flow pattern inside the cooling hole, whose calculation is significantly influenced by both the $k-c$ turbulence model applied and the resolution of the computational grid. A finer grid resolution of the film-cooling hole will be examined in the near future.

As shown in section 4.1.1 for the contoured hole geometries, the calculation reveals a jet penetrating into the mainstream channel at the leading edge region of the hole exit, which is in contrast to the experimental results. This upstream shift of the calculated jet is confirmed by the flow pattern of the streamwise velocity at the hole exit ($y/D = 0.1$) for the laidback fanshaped hole (see Fig. 12). This figure clearly demonstrates that large differences between calculation and measurement occur for the
development of the jet inside the hole. At the hole exit, the calculation mimics a jet concentrated along the symmetrical plane $z/D = 0$ with its maximum at the leading edge, whereas the experiment indicates the jet exiting over a large portion in the downstream region of the hole.

Details of the complex flow pattern inside the hole ($z/D = -0.75$) are given in Figs. 13 and 14 for the streamwise and lateral velocity component, respectively. As can be seen in Fig. 13, the predicted and measured $u$-velocity reveal a quite contrasting distribution. In the experiment the jet inside the hole is located towards the bottom region of the hole ($y/D \approx -1$), whereas the simulation indicates the jet lifted up towards the top side. In contrast to this, the secondary motion inside the hole (Fig. 14) is qualitatively captured by the calculation in the upper region of this cross-sectional cut. For both, simulation and experiment an increased fluid motion is observed from the symmetrical plane $z/D = 0$ towards the outer region of the cross-section. However, the amount of the lateral velocity component is overpredicted by the numerical code, indicating a much stronger vortex motion inside the hole. The reverse flow in the bottom region shown by the simulation cannot be adequately judged because of lack of experimental data in this area.

In order to identify the mechanism causing a lift-off of the calculated jet for the contoured hole geometry, a more detailed analysis of the velocity vectors at the hole inlet was performed which revealed that no separation occurs at the downstream edge of the hole inlet in the coolant channel. Thus, in contrast to the results reported by Leyte and Zerkle (1994), a separation at the leeward region of the hole entrance, which may be a possible mechanism for a jet lifted towards the windward edge of the hole, can be excluded in this study. The jet lift-off indicated by the present computations can be addressed to the overestimation of the vortex rotation (see Fig. 14). This difference between calculation and measurement in terms of vortex strength is addressed to the $k-\varepsilon$ turbulence model which is unsuited to model the complex anisotropic flow situation in the contoured hole passage: it seems that due to the underprediction of the turbulence shown by the calculation at the contoured hole exit (see Fig. 9) and, therefore, also inside the hole, the predicted eddy viscosity is too low to dampen the vortex rotation to values revealed by the experiment.

4.1.3 Flowfield downstream of the hole exit

In Fig. 15 the predicted vertical velocity component is compared with the measured data for the cylindrical hole geometry. This cross-sectional cut is located downstream of the hole exit at an axial location of $z/D = 4$. As can be seen, very good agreement with the experiment is found. The secondary flow pattern in the wake region downstream of the hole exit, which contributes to the kidney shape of the jet field, is adequately picked up by the simulation.

Fig. 16 shows the result for this velocity component for the contoured hole geometry. It should be noted that the position of the cut corresponds to a location one diameter downstream of the trailing edge of the hole exit. Also very good agreement is found for this normal velocity. But a more detailed analysis shows an insufficient rate of spreading. This deviation between calculation and experiment has its origin in the location of the exiting jet relative to the hole exit; the computation reveals a more concentrated jet along the symmetrical plane (Fig. 12), which is clearly reflected in this figure.

The turbulence intensity distributions at the same plane for the round and expanded hole geometry are shown in Figs. 17 and 18. As can be seen for the cylindrical hole in Fig. 17, the experiment reveals high turbulence levels in the shear layer between jet and wake region due to strong interaction of the penetrating jet with the crossflow. The maximum in turbulence generated downstream of the hole exit is significantly underestimated by the calculation. The reasons for this are the same as described in section 4.1.1.

For the contoured hole geometry, remarkable similarity is obtained for the calculated turbulence contours (Fig. 18). Even the maximum of turbulence intensity in the more moderate shear-layer is adequately captured by the calculation. This result gives rise to the assumption, that the application of the standard $k-\varepsilon$ turbulence model is, relative to the round hole, more adequate in this flow situation due to the weak interaction between jet/crossflow and jet/wake region downstream of the hole exit.

4.2 Adiabatic Film Cooling Effectiveness

In this section computational results are presented for the adiabatic film cooling effectiveness $\eta$. Subsequently, for all flow cases and geometries considered the total temperature in the mainstream and the cooling channels are $T_{cool, m} = 535\, ^\circ F$ and $T_{cool, e} = 290\, ^\circ F$, respectively, which leads to a density ratio $DR = \rho_{j}/\rho_{m} = 1.85$. The blowing ratio for the round hole geometry is $M = 1.0$, whereas the blowing ratio for the contoured hole is $M = 1.5$. Details of the film-cooling effectiveness study can be found in Gritsch et al. (1997).

Results for the centreline effectiveness versus downstream distance as well as the lateral variation of effectiveness at several downstream positions are compared with experimental data for both the cylindrical hole and the laidback-fanshaped hole geometry. The definition of the film cooling effectiveness which is used for the comparison is given as follows:

$$\eta = \frac{T_{cool, m} - T_{wall}}{T_{cool, m} - T_{wall}}.$$

Here, $T_{cool, m}$ is the recovery temperature at the mainstream channel wall and $T_{wall}$ is the wall temperature, respectively. The position which is not affected by the coolant ejection. $T_{cool, m}$ and $T_{wall}$ are the total temperature in the cooling channel and the local adiabatic wall-temperature, respectively.

The origin of the co-ordinate system is now placed at the trailing edge of the hole exit.

4.2.1 Effectiveness along jet centerline plane

Computed and measured distribution of the adiabatic film cooling effectiveness along the jet centreline for the cylindrical hole is shown in Fig. 19. As can be seen from this figure, the measured distribution shows a pronounced decrease and a following 'bump' which indicates that the jet is detached from the bottom wall and reattaches downstream. This phenomenon is not adequately picked up by the calculation. Only a slight increase of $\eta$ at the location $z/D = 2$ suggests that the jet lift-off and reattachment is recognized but not sufficiently captured. A better prediction of this effect may be achieved if a finer grid in the near wall region together with a low-Reynolds number extension of the standard $k-\varepsilon$ turbulence model is applied. This is confirmed by the results of Walters and Leyte (1996) who found indeed the jet lift-off and reattachment when using a high grid density near the wall downstream of the hole exit. Jansson and Davidson (1994) also used a very fine grid resolution near the wall in conjunction with a one-equation turbulence model and they could pick up lift-off and reattachment of the jet in their investigation, too. Due to the increase of $\eta$ in the experiment caused by the reattachment of the cooling jet an underprediction of the experimental result in the downstream region ($z/D > 2$) is indicated by the calculation. Because of the lack of experimental data at $z/D > 8$ it is not clear if the measured effectiveness approaches the predicted distribution.

Fig. 20 shows the comparison of the measured and predicted adiabatic film cooling effectiveness along the jet centreline plane for the laidback fanshaped geometry. In contrast to the round hole the measurement exhibits a jet attached to the wall of the mainstream channel even for this higher blowing ratio. This is also captured by the calculation. Moreover, the calculation shows a qualitative good agreement with the experimental data right downstream of the hole exit ($z/D > 0$) but overpredicts the measurements by approximately 15%. For this higher blowing ratio the calculation shows a consistent trend relative to the experimental data with a correct decay rate of $\eta$ for downstream locations $0 < z/D < 5$. 


distribution of the jet exiting the hole has a strong influence upon the simulation also picked up an inward shift of the maxima. This result leads to the assumption that the application of the standard k-ε turbulence model, which cannot predict the correct turbulence levels and thus diffusions rates for the round hole geometry. As pointed out by Andreopoulos and Rodi (1984), the gradient $\partial \psi / \partial y$ may have a significant contribution in a jet-crossflow situation to the production of the shear stress $\psi'$ which represents the lateral turbulent mixing and governs the lateral spreading. However, this gradient is not taken into account by a eddy-viscosity model based on the Boussinesq approximation.

Fig. 22 shows the lateral distribution of the adiabatic effectiveness for the hole with expanded exit. Remarkable agreement between calculation and measurement is found for the lateral spreading of the effectiveness. This result leads to the assumption that the application of the standard k-ε model is more suitable for this flow situation compared to the cylindrical hole. As already shown in Figs. 7-9 and Figs. 17-18 for $M = 1.0$, the turbulence intensity shows a better agreement between calculation and measurement for the contoured hole geometry rather than for the cylindrical hole. This suggests that the turbulent diffusion rates are adequately captured by the simulation for this higher blowing ratio flowcase. Moreover, it seems that due to reduced gradients in the downstream region of the hole exit the gradient $\partial \psi / \partial y$ plays a minor role for the $\psi'$-stress so that the lateral distribution of the effectiveness can be predicted with the standard k-ε model.

Qualitative agreement is displayed for the $\eta$-distribution in the region near the jet centerline where $\eta < 1.7$. The calculation shows a similar variation of $\eta$ but overpredicts the experimental data. Unlike the measured distribution the simulation also picks up an inward shift of the maxima for the hole with expanded exit. Remarkable agreement between calculation and measurement is achieved for the contoured hole. This suggests that the turbulent diffusion rates are adequately captured by the simulation for this higher blowing ratio flowcase. Moreover, it seems that due to reduced gradients in the downstream region of the hole exit the gradient $\partial \psi / \partial y$ plays a minor role for the $\psi'$-stress so that the lateral distribution of the effectiveness can be predicted with the standard k-ε model.

5 Conclusions
A computational study on film-cooling for three different holes geometries was presented. Calculations were performed with a 3D Navier-Stokes code. For turbulence closure, the standard k-ε model with wall function approach was applied.

In terms of flow structure the calculations are in good agreement with experiments in the farfield region of the hole exit independent of the hole geometry used. However, deviations are found for the mean velocity inside the hole and at the hole exit. These effects are assumed to be attributed to both a too coarse computational grid and the application of the standard k-ε model which cannot cope with the highly anisotropic turbulent flow leading to a significant underprediction of peak turbulence levels. As expected, the agreement between calculation and experiment for the mean velocities is directly linked to the degree of accordance in turbulence. Since the source of turbulence for the round hole is located downstream of the hole exit, where the velocity gradients are large, the computation agrees reasonable good with the measurement inside the hole and at the exit for this geometry. The situation is worse for the contoured holes with expanded exits where the peak turbulence levels appear inside the hole and at the hole exit, respectively. Here, the location of the exiting jet is strongly shifted compared to the experiment.

Good results are achieved for the hole with forward-laterally expanded exit for the adiabatic film-cooling effectiveness in terms of distribution of $\eta$ along the jet centerline and its rate of lateral spreading. The quite low turbulence production downstream of the expanded holes and the absence of a jet lift-off and subsequent reattachment are the main reasons for these results. An underprediction of lateral spreading of the effectiveness for the round hole is caused by high turbulence downstream of the hole exit which cannot adequately picked up by the calculation.

Further improvements of the computational results may be possible if the computational grid is refined in those flow regions where high turbulence levels are generated. Additionally, a turbulence model should be applied which takes anisotropic effects into account in order to give a better prediction of turbulent diffusion rates. Moreover, since the use of the wall function approach is questionable for the calculation of detaching and reattaching flows a low-Reynolds number extension of the applied turbulence model would be appropriate.

6 Acknowledgements
This study was partly funded by the European Union through grant by the Brite EuraM program "Investigation of the Aerodynamics and Cooling of Advanced Engine Turbine Components" under Contract AER2-CT92-0044. The authors wish to express their gratitude to the partners involved in the program for the permission to publish this paper.

References


Fig. 4: u- and v-velocity contour lines in the mainstream channel ($z/D=0$), cylindrical hole

Fig. 5: u- and v-velocity contour lines in the mainstream channel ($z/D=0$), fanshaped hole

Fig. 6: u- and v-velocity contour lines in the mainstream channel ($z/D=0$), laidback fanshaped hole
Fig. 7: Turbulence contour lines at z/D=0, cylindrical hole

Fig. 8: Turbulence contour lines at z/D=0, fanshaped hole

Fig. 9: Turbulence contour lines at z/D=0, laidback fanshaped hole

Fig. 10: u-velocity at hole exit (y/D=0.1), cylindrical hole

Fig. 11: u-velocity at hole inlet (y/D=-3.15), cylindrical hole

Fig. 12: u-velocity at hole exit (y/D=0.1), laidback fanshaped hole

Fig. 13: Near-exit u-velocity inside the hole (x/D=-0.75), laidback fanshaped hole

Fig. 14: Near-exit w-velocity inside the hole (x/D=-0.75), laidback fanshaped hole
Fig. 15: v-velocity at axial location x/D=4, cylindrical hole

Fig. 16: v-velocity at axial location x/D=4, laidback fanshaped hole

Fig. 17: turbulence level at axial location x/D=4, cylindrical hole

Fig. 18: turbulence level at axial location x/D=4, laidback fanshaped hole

Fig. 19: streamwise distribution of $\eta$, cylindrical hole

Fig. 20: streamwise distribution of $\eta$, laidback fanshaped hole

Fig. 21: lateral distribution of $\eta$, cylindrical hole

Fig. 22: lateral distribution of $\eta$, laidback fanshaped hole