

Article ID: 1000 8055(2007) 04- 0521- 10

LES of film cooling for different jet fluids

P. Renze, W. Schröder, M. Meinke

(Aerodynamisches Institut, RWTH Aachen, 52062 Aachen, Germany)

Abstract: The present paper investigates the impact of the velocity and density ratio on the turbulent mixing process in gas turbine blade film cooling. A cooling fluid is injected from an inclined pipe at $\alpha = 30^\circ$ into a turbulent boundary layer profile at a freestream Reynolds number of $Re_\infty = 400000$. This jet in a crossflow (JICF) problem is investigated using large eddy simulations (LES). The governing equations comprise the Navier-Stokes equations plus additional transport equations for several species to simulate a non reacting gas mixture. A variation of the density ratio is simulated by the heat mass transfer analogy, i. e., gases of different density are effused into an air crossflow at a constant temperature. An efficient large eddy simulation method for low subsonic flows based on an implicit dual time stepping scheme combined with low Mach number preconditioning is applied. The numerical results and experimental velocity data measured using two component particle image velocimetry (PIV) are in excellent agreement. The results show the dynamics of the flow field in the vicinity of the jet hole, i. e., the recirculation region and the inclination of the shear layers, to be mainly determined by the velocity ratio. However, evaluating the cooling efficiency downstream of the jet hole the mass flux ratio proves to be the dominant similarity parameter, i. e., the density ratio between the fluids and the velocity ratio have to be considered.

Key words: aerospace propulsion system; large eddy simulations (LES); jet in a crossflow; film cooling; density ratio; velocity ratio; cooling efficiency

CLC number: V231.1

Document code: A

Introduction

In modern gas turbines film cooling mechanisms protect the turbine components from the thermal stresses resulting from the exposure to the hot gas stream. Since the technical design process of film cooling systems depends on the exact knowledge of the generated flow field, a detailed understanding of the flow physics is a must to improve existing cooling techniques.

In the present case, the cooling film is generated by an injection of a cooling fluid through a row of staggered holes. The flow field resulting from the interaction of the inclined cooling jet and the turbulent boundary layer is governed by complex vortex dynamics. The outer field is dominated by a counter-rotating vortex pair (CVP), which is the leading mechanism in the

mixing process between the hot gas and the coolant. Due to the high impact of film cooling on the thermal efficiency of turbines a wide range of experimental investigations on this matter can be found in the literature, e. g., Andreopoulos and Rodi^[1] or the recent study Plesniak and Cusano^[2]. An extensive study of the effect of the density ratio has been presented by Goldstein and Eckert^[3]. In a following investigation Pedersen et al.^[4] studied the impact of this parameter applying the heat mass transfer analogy. They reported a strong influence of the density ratio on the local film cooling effectiveness by measuring the concentration of a high density cooling gas along a plate. Measurements of cryogenically cooled jets with thermocouple arrangements have been performed by Pietrzyk et al.^[5] and Sinha et al.^[6]. The authors of the former paper measured

the turbulent intensities at different density ratios and in the latter article the cooling effectiveness was found to be improved with increasing density ratio.

Most numerical investigations of the JICF problem have been based on Reynolds-averaged Navier-Stokes equations, e. g., Hoda and Acharya^[7] or the systematic study of film cooling physics by Walters and Leylek^[8]. However, since the JICF problem is influenced by effects of wall-bounded and free turbulence, most standard turbulence models like zero-, one- or two-equation models fail to correctly predict the resulting flow field. For this reason, it is necessary to apply a more general numerical ansatz such as large-eddy simulation (LES) to investigate JICF problems. Such an analysis was performed, for instance, by Tyagi and Acharya^[9]. In this work random perturbations have been introduced at the inlet of the turbulent boundary layer. An LES study that includes an accurate treatment of the incoming turbulent boundary layer and a plenum area was performed by Guo et al.^[10], who investigated among other issues the effects of inclination angle and blowing ratio on the flow field. In [11] a similar study has been reported based on likewise ideas as far as the geometry and the numerical set-up is concerned as in the analysis discussed in the prior publications by Guo et al.^[10, 12-14].

In the present paper the impact of the density ratio between coolant and crossflow is analyzed by LES and compared with experimental data. According to Pietrzyk et al.^[5] there are three different ways to vary the density ratio parameter: a heated freestream flow, cryogenically cooled injectant flow, and foreign gas injection. Since the first two options require very complex and expensive wind tunnel measurements and such experiments are performed in [15] for comparison reasons, the density ratio parameter is studied by the heat-mass transfer analogy, i. e., a foreign gas (CO₂) jet is injected into an air crossflow.

The paper is organized as follows. After a

succinct description of the governing equations and the numerical method the flow configuration and the boundary conditions are presented. In the results section the flow structures of a round jet in a crossflow are analyzed and the impact of velocity ratio and the density ratio will be studied in detail. The findings of the numerical simulations will be compared with two-component particle image velocimetry (PIV) measurements of Jessen et al.^[15]. Finally, the cooling efficiency will be evaluated.

1 Governing equations

The simulation of a turbulent and nonreacting multicomponent flow is considered. The governing equations are the Navier-Stokes equations including conservation equations for the partial densities ρ_n of $N-1$ species, where N is the total number of different species. In tensor notation and in terms of dimensionless conservative variables the equations read in a Cartesian coordinate system

$$\frac{\partial \mathbf{Q}}{\partial t} + (\mathbf{F}_\beta^c - \mathbf{F}_\beta^d)_{\beta=0}, \quad \mathbf{Q} = [\rho n, \rho, \rho u_\alpha, \rho E]^T, \quad (1)$$

where $\mathbf{Q}$ is the vector of the conservative variables and $\mathbf{F}_\beta^c$ denotes the vector of the convective and $\mathbf{F}_\beta^d$ of the diffusive fluxes

$$\mathbf{F}_\beta^c - \mathbf{F}_\beta^d = \begin{pmatrix} \rho u_\beta \\ \rho u_\beta \\ \rho u_\alpha u_\beta + p \delta_{\alpha\beta} \\ u_\beta (E + p) \end{pmatrix} + \frac{1}{Re} \begin{pmatrix} j_{n\beta} \\ 0 \\ \alpha_{n\beta} \\ u_\alpha \sigma_{\alpha\beta} + q_\beta \end{pmatrix} \quad (2)$$

The stress tensor $\sigma_{\alpha\beta}$ is written as a function of the strain rate tensor $S_{\alpha\beta}$

$$\sigma_{\alpha\beta} = -\nu_2 (S_{\alpha\beta} - \frac{1}{3} S_{\gamma\gamma} \delta_{\alpha\beta}) \quad \text{with} \quad S_{\alpha\beta} = \frac{1}{2} (u_{\alpha,\beta} + u_{\beta,\alpha}). \quad (3)$$

The mass diffusion $j_{n\beta}$ is described by Fick's law

$$j_{n\beta} = - \frac{\mathcal{D}_n}{S_{c0}} Y_{n,\beta} \quad (4)$$

where Y_n is the mass fraction of the species n , $\mathcal{D}_n$ is the diffusion coefficient computed with mixing rules from the binary diffusion coefficients, $S_{c0} = \mathcal{W}_0/\mathcal{D}_0$ is the Schmidt number, and the subscript 0

indicates the reference state of the mixture. Fourier's law of heat conduction is used to compute the heat flux q^β

$$q^\beta = - \frac{k}{Pr_0(\gamma_0 - 1)} T_{,\beta} \quad (5)$$

where Pr is the Prandtl number. The system is closed with the equation of state for a mixture of ideal gases:

$$p = \sum_n p^n \quad \text{with} \quad p^n = \frac{T}{\gamma_0} \rho R^n \quad (6)$$

where γ is the ratio of the specific heats, T the temperature, p the pressure, and R the gas constant. All transport coefficients are assumed to be a polynomial function of 4th order of the temperature, e. g., for the coefficient of heat conductivity k

$$\ln k_n^* = \sum_i a_i^* (\ln T^*)^{i-1}. \quad (7)$$

The superscript $*$ denotes dimensional values. The values of the transport coefficients for the mixture of the species are determined according to the mixing rules of Wilke^[16] and Bird^[17], for the diffusion coefficient and the viscosity, respectively, and according to Mathur^[18] for the heat conductivity. In the mixing rule for the diffusion coefficient D_n the pressure dependence is taken into account.

2 Numerical Method

The discretization of the governing equations is based on a mixed central-upwind AUSM (advective upstream splitting method) scheme using a centered 5 point low dissipation stencil to compute the pressure derivative in the convective fluxes. This scheme is described and validated in Meinke et al.^[19]. Similar to the MILES approach as reported by Fureby and Grinstein^[20,21], the inherent dissipation of the numerical scheme serves as an implicit SGS model.

The temporal approximation is based on an implicit dual time-stepping scheme. A convergence acceleration of the low Mach number problem is achieved using multigrid methods and preconditioning. This dual time stepping algorithm is described in Alkishriwi et al.^[22]. Therefore, a derivative with respect to the pseudo-time τ is

introduced in Eqn. (1)

$$\Gamma^{-1} \frac{\partial \mathbf{Q}}{\partial \tau} + \frac{\partial \mathbf{Q}}{\partial t} + \mathbf{R} = 0 \quad (8)$$

The matrix of the preconditioned scheme Γ has been introduced by Turkel et al.^[23] and is modified to account for the additional mass fraction equations. $\mathbf{R}$ represents the convective and viscous fluxes. A 5-step Runge-Kutta method is applied to propagate the vector of solution $\mathbf{Q}$ in Eqn. (1) from pseudo-time level n to $n+1$.

3 Boundary conditions

In large eddy simulations of spatially developing boundary layers the treatment of the inflow boundary of the computational domain is one of the most difficult problems. In general and in particular in the JICE problem, the flow depends highly on the solution prescribed at the inlet.

The most physical ansatz to meet this challenge is to start to compute the flow right at the leading edge where the boundary layer emanates. However, the computational costs would be too extensive for the present problem. Tyagi and Acharya^[9] prescribed a turbulent profile and superimposed random fluctuations, which on the one hand, is based on minimum effort, on the other hand, however, generates a slightly discontinuous solution, e. g., of the wall shear stress. The following detailed analysis of the vortical structures in the vicinity of the jet hole does suffer from such unphysical inlet conditions of the boundary layer.

Therefore, in the present work an independent spatially developing boundary layer simulation is performed and based on a slicing technique the inflow distribution is prescribed using the velocity profile possessing the boundary layer parameters necessary at the inflow boundary of the jet in a crossflow problem. This method is indicated in Fig. 1.

The auxiliary flat plate flow simulation generates its own turbulent inflow data using the compressible rescaling method proposed by El-Askary et al.^[24,25]. The rescaling method is a

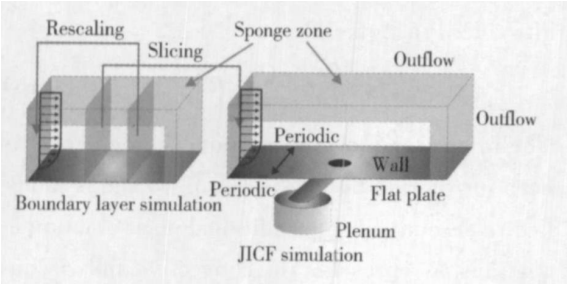


Fig. 1 Schematic of the computational domains and the boundary conditions of the JICF simulation

means of approximating the properties at the inlet via a similarity approach applied to the downstream solution. A detailed validation of this inflow generation method and its application to the JICF problem is given by Guo et al.^[10].

A sketch of the whole computational domain, which consists of the auxiliary domain, the JICF domain, a plenum, and a pipe, from which the cooling jet is injected into the crossflow, is shown in Fig. 1. In the spanwise direction full periodic boundary conditions are applied. Thus, an infinite row of staggered holes at a distance of $S=3D$ between the centerlines is simulated. At the inlet of the plenum area the stagnation pressure is prescribed. The cooling fluid is driven through the pipe by the pressure difference between crossflow and plenum. The stagnation pressure is controlled by the calculation of the mass flux through the pipe to enforce a correct mass flux ratio. At the outer circumferential surface of the plenum the mass flux is assumed to vanish asymptotically.

The outflow boundary conditions are based on the conservation equations written in characteristic variables. To damp numerical reflections at the outflow boundaries sponge zones are introduced into the computational domain, as indicated in Fig. 1. In these regions source terms are superposed onto the right-hand side of the governing equations to drive the instantaneous solution of Q to a desired target solution Q_t , e. g., the logarithmic boundary layer profile. The source terms are formulated as

$$S = \sigma [Q(t, \mathbf{x}) - Q_t(\mathbf{x})] \tag{9}$$

where the parameter σ is a polynomial function of the distance to the outflow boundary and de-

creases from $q_{\max}$ to 0 within the boundary layer.

On the wall the no-slip conditions are prescribed, the pressure gradient is assumed to be negligible, and adiabatic conditions are imposed.

4 Flow configuration

The domain of integration of the JICF simulation is shown in Fig. 2. The dashed lines mark the outer boundaries of the computational domain. The cooling fluid is driven by a pressure gradient from a plenum into the boundary layer flow through a 30° streamwise inclined pipe. The origin of the frame of reference is located at the jet hole center. The coordinates x, y, z represent the streamwise, normal, and spanwise direction.

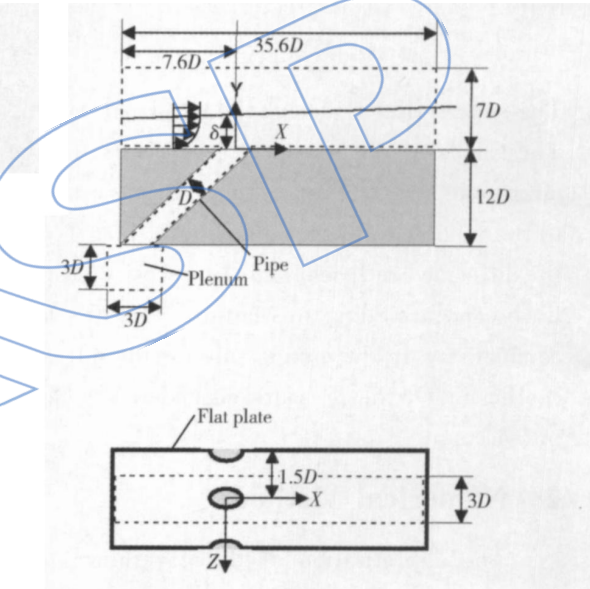


Fig. 2 Schematic of the flow configuration in the $x-y$ plane(top) and the $x-z$ plane(bottom)

To mimic the flow parameters in a gas turbine a cooling fluid is injected from a complete row of jets into a turbulent flat plate boundary layer at a Mach number $Ma=0.2$ and a local Reynolds number of $Re_\infty=400\,000$ based on the length from the leading edge of the flat plate to the hole. The ratio of the local boundary layer thickness to the hole diameter is $\delta/D=2$. The relevant flow parameters and the geometrical parameters are summarized in table 1. The velocity, density, and mass flux ratio are defined as

$$VR = \frac{u_j}{u_\infty}, DR = \frac{\rho}{\rho_\infty}, \text{ and } MR = \frac{\rho u_j}{\rho_\infty u_\infty} \tag{10}$$

where the subscripts j and ∞ denote the jet and the crossflow fluid. The mass flux is varied as three different velocity ratios are investigated $VR=0.1, 0.28$ and 0.48 . Two different values of the density ratio DR , an air-to-air injection with $DR=1$ and a CO_2 -to-air injection with $DR=1.53$, are considered. All cases are listed in table 1.

Table 1 Flow and geometry parameters

case	Re_∞	Ma	$\alpha/(^\circ)$	D/mm	δ_0/D	VR	DR	MR
1	4×10^5	0.2	30	10	2	0.1	1	0.1
2	4×10^5	0.2	30	10	2	0.28	1	0.28
3	4×10^5	0.2	30	10	2	0.48	1	0.48
4	4×10^5	0.2	30	10	2	0.1	1.53	0.153
5	4×10^5	0.2	30	10	2	0.28	1.53	0.43

5 Results and discussion

The result section possesses the following structure. First, some computational details are explained. Then, the impact of the velocity and the density ratio on the flow field is discussed. Subsequently, the findings are compared with experimental data to show the quality of the computational method. Next, the instantaneous mixing process is analyzed and the characteristic properties of the jet-crossflow interaction are explained. Finally, the cooling efficiency is studied by means of the mixture fraction distribution of the denser gas.

5.1 Computational details

All simulations have been performed on a block-structured mesh shown in Fig. 3. It consists of 5.65 million cells distributed over 24 blocks. The purpose of the blocks upstream of the jet hole is to deliver an independent boundary layer simulation as mentioned above. The grid points are clustered near the solid walls of the flat plate and the pipe. The wall-normal distance of the control volume at the flat plate has a dimension of $\Delta y = 0.004D$, which corresponds to $\Delta y^+ = 1.0$.

The physical time step is $\Delta t = 0.02D/U_\infty$. To obtain the time-averaged statistics the flow field has been sampled over 8 time periods. Here, one period is the time the crossflow fluid needs to pass over the length of the plate.

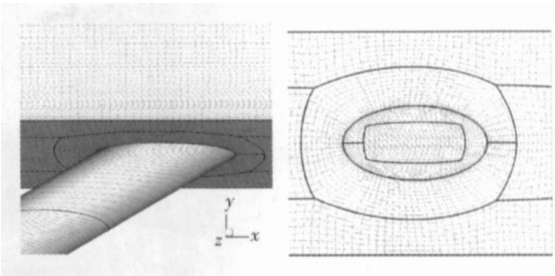


Fig. 3 Block structured mesh used for the JICF simulation; left: 3d view of the pipe-to-plate junction; right: cross section at $y/D=0$; every second node shown.

5.2 Time-averaged quantities

The interaction of the turbulent boundary layer with a high density cooling jet is shown in Fig. 4. The velocity ratio is $VR=0.28$ and the density ratio is $DR=1.53$, which resembles the CO_2 -air mass ratio. The upper left view displays the JICF part of the computational domain. The velocity field is emphasized by streamlines to show the deflection of the crossflow by the jet and the swirling motion of the jet fluid being entrained by the crossflow. In several cross sections at different x/D locations contours of the mean mixture fraction f of the jet fluid are shown. The cross sections are enlarged and vectors to evidence the secondary velocity field are added. At $x/D=0$ the high density fluid starts to penetrate into the crossflow and the shear layers start to roll up at the jet hole edges. At $x/D=1.5$ the counter-rotating vortex pair (CVP) resulting from the shear of the crossflow fluid by the jet^[26] is already fully developed. The mixing between both fluids follows the development of the vortex pair. The maximum values of the mixture fraction f mark the center of the unmixed jet fluid. This is lifted off the wall as low density fluid from the crossflow is entrained between the wall and the jet by the CVP. The cross section at $x/D=3$ shows the streamwise development of the CVP that continuously grows. The vorticity magnitude is decreased, the centers of the CVP approach each other and drift away from the surface.

To figure out the impact of the velocity ratio and the density ratio on the cooling efficiency the parameters VR and DR are varied in

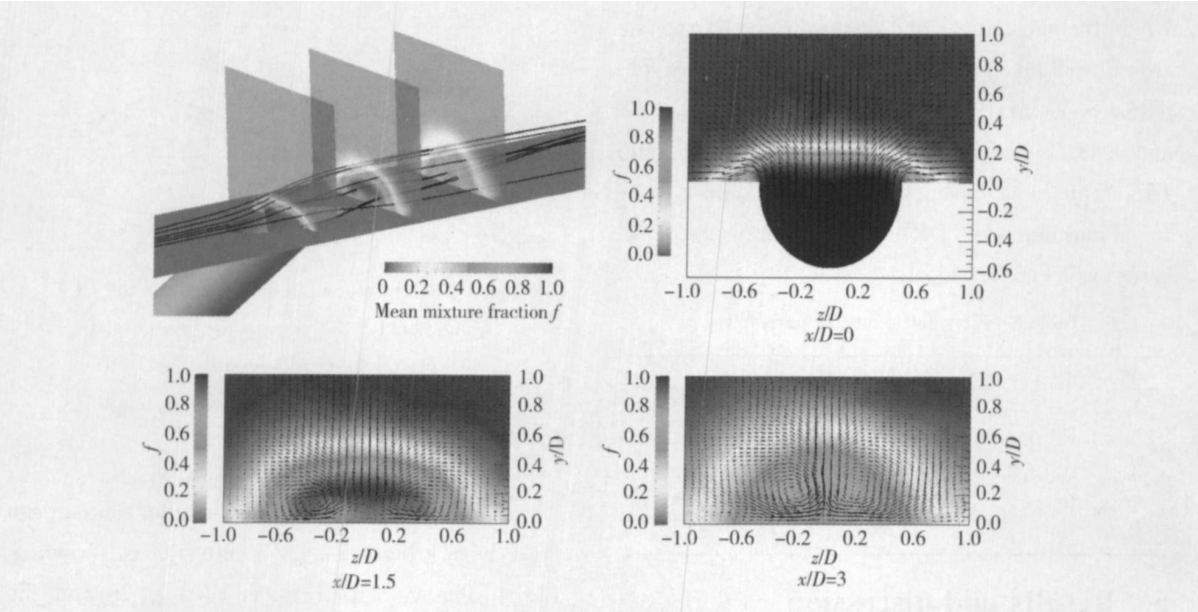


Fig.4 $VR=0.28$ and $DR=1.53$, streamlines and contours of the mixture fraction f at different cross sections(upper left), contours of f and velocity vectors in the cross section at $x/D=0$ (upper right), cross section at $x/D=1.5$ (lower left), and cross section at $x/D=3$ (lower right).

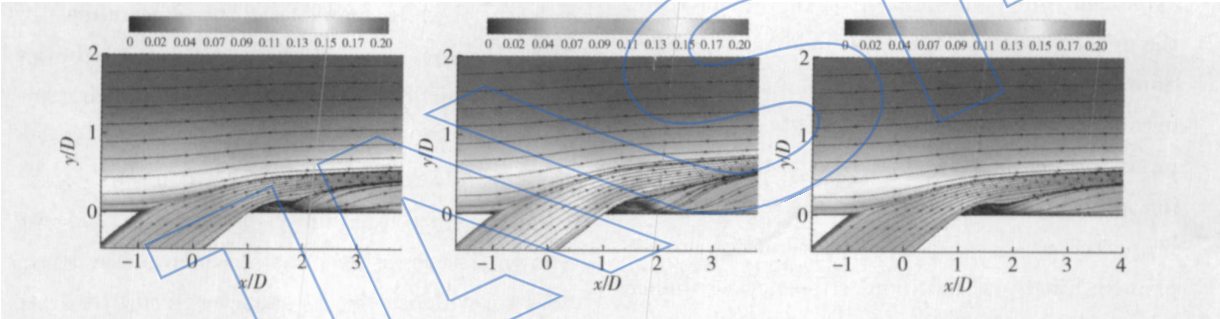


Fig.5 Mach number contours in the JICF symmetry plane and streamlines at $VR=0.28$ and $DR=1$ (left), at $VR=0.48$ and $DR=1$ (center), at $VR=0.28$ and $DR=1.53$ (right).

the large eddy simulations. In Fig.5 Mach number contours and streamlines are extracted in the JICF symmetry plane for three different cases. It is obvious from the left and center illustrations that a variation of the velocity ratio has a large impact on separation and reattachment behavior as well as the strength of the penetration into the crossflow. However, a variation of the density ratio at a constant velocity ratio, which is given in the left and right illustrations, has only a minor effect on the dynamics of the flow field in the vicinity of the jet hole. The size of the recirculation region, the wall normal velocity gradients, and the penetration depth show only small differences if the velocity ratio is kept constant. Just the lateral spreading of the jet fluid is slightly increased at a higher jet density. This re-

sult agrees with the conclusion reported in [5].

The same behavior becomes evident when turbulent quantities are considered. In Fig.6 the contours of the time averaged streamwise velocity fluctuation $(\overline{u'^2})^{1/2}/U_\infty$ are shown in the symmetry plane at $DR=1$ and $VR=0.28$. The highest turbulence levels are clearly located at the windward side of the jet exit, i.e., just upstream of the jet hole, and in the shear layer between the recirculation region and the jet. In the latter zone the major part of the mixing between both fluids takes place. The high shear between the jet and the wake region also possesses a high density gradient since crossflow fluid is entrained between the wall and the jet.

Profiles of the streamwise velocity fluctua-

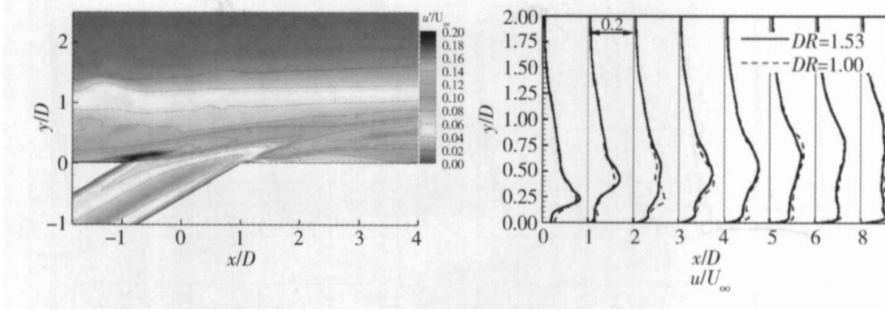


Fig. 6 Contours of the streamwise velocity fluctuation $(\overline{u'^2})^{1/2}/U_\infty$ in the symmetry plane at $DR=1$, $VR=0.28$ (left) and profiles of the streamwise velocity fluctuation for $DR=1$ and $DR=1.53$ at different x/D locations

tion at different x/D locations also are shown in Fig. 6 at two different density ratios $DR=1$ and $DR=1.53$. The wall normal distribution and the peak levels almost match at both density ratios. This is expected as the velocity gradients in the mean flow field are almost identical.

5.3 Comparisons with PIV measurements

The LES technique of the present study is validated by comparing the time-averaged flow field with the particle image velocimetry (PIV) measurements of Jessen et al.^[15]. In Fig. 6, 3 profiles of the streamwise velocity are compared for two different density ratios ($DR=1$, $DR=1.53$) and at two different velocity ratios $VR=0.1$ and $VR=0.28$. The profiles are located in the spanwise symmetry plane ($z/D=0$) at different streamwise locations ($x/D=-1, 0, 1, 1.5$, and 2). The location $x/D=-1$ corresponds to the upstream edge of the jet hole. The boundary layer flow is nearly undisturbed at this location and the velocity profile matches the fully developed turbulent profile at the corresponding Reynolds number. At $x/D=0$ and $x/D=1$ the boundary layer is lifted by the jet. In the case of the higher velocity ratio the jet is separated at $x/D=1.5$ and reattached at $x/D=2$. In all four cases the predicted flow field is in excellent agreement with the PIV measurements. As mentioned before, the velocity field in the vicinity of the jet injection is very similar for both density ratios.

5.4 Characteristics of the jet crossflow interaction

The coupling of the flow dynamics to the

turbulent mixing process of the different species is closely connected to the formation of the counter-rotating vortex pair (CVP) downstream of the jet hole. There are different explanations for the generation of the CVP, e. g., those from Kelso and Smits^[27] or Morton and Ibbetson^[28]. The present study agrees with the hypothesis discussed in detail by Peterson and Plesniak^[26]. They state the reason for the initial formation of the CVP to be the shear of the boundary layer fluid by the jet.

Due to the high resolution as well as the time accuracy of the present numerical scheme and the possibility to track the passive mixture fraction scalar a detailed analysis of the formation of the CVP and the initial mixing process can be performed. In Fig. 8 two cross sections at $x/D=1$, i. e., directly at the trailing edge of the hole, and further downstream at $x/D=1.5$ display contours of the mixture fraction f and vectors of the secondary velocity field at the same instantaneous time level. The velocity ratio is $VR=0.28$ and the density ratio is $DR=1.53$. The vorticity, which originates from the spanwise edges of the jet hole, causes the initial roll-up in the shear zone. The resulting eddies are already lifted off the plate at $x/D=1$. The mixture fraction on the plate wall is reduced as crossflow fluid is entrained under the jet by the large-scale vortices. Further downstream, in the zone that is governed by oscillating separation and reattachment the vortical structures possess a smaller scale, are more intense and intricate, and as such the mixing process is accelerated.

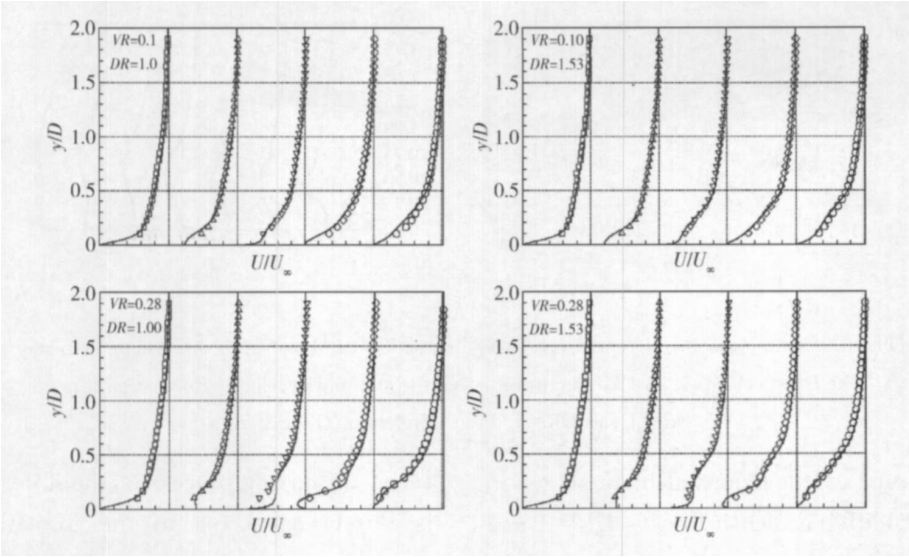


Fig. 7 Profiles of streamwise velocity in the symmetry plane at different streamwise locations, top left: $VR=0.1$, $DR=1$, bottom left: $VR=0.28$, $DR=1$, top right: $VR=0.1$, $DR=1.53$, bottom right: $VR=0.28$, $DR=1.53$, experimental data $\square$: $x/D=-1$, $\triangle$: $x/D=0$, ∇ : $x/D=1$, $\diamond$: $x/D=1.5$, $\circ$: $x/D=2$, solid lines: LES data.

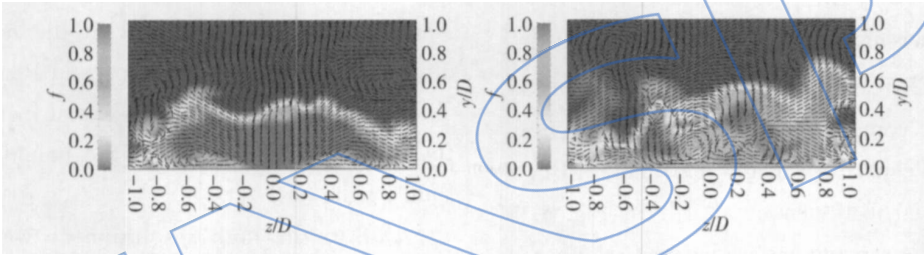


Fig. 8 Instantaneous mixture fraction contours and velocity vectors at $X/D=1$ (left) and $X/D=1.5$ (right), $VR=0.28$, $DR=1.53$

The separation/reattachment zone can be identified in Fig. 9. The mixture fraction f is shown at an instantaneous time level in the streamwise symmetry plane. On the right, coherent structures indicated by the λ_2 criterion^[29] with mapped on mixture fraction distribution f are depicted. The selected views in Fig. 9 visualize the interaction of the inherent hairpin vortices of the turbulent boundary layer with the effusing

jet fluid. The hairpin like structures in the upper part of the boundary layer are lifted and entrain jet flow fluid only after they passed the trailing edge. The lower part of the boundary layer is deflected by the blockage effect of the jet. The small scale structures lead to the initial mixing in the shear zones. Further downstream the mixing process is significantly accelerated in the separation region.

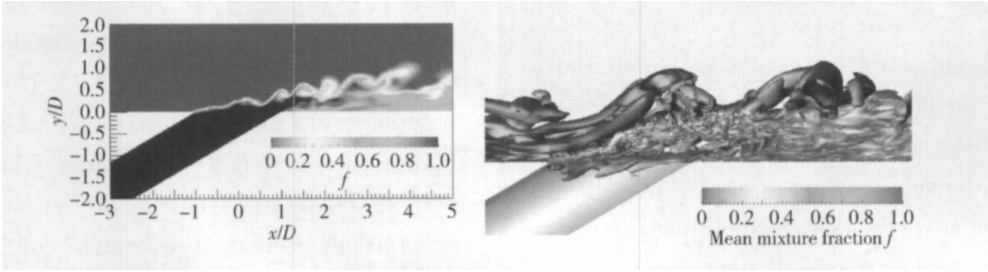


Fig. 9 Left: Instantaneous contours of the mixture fraction f in the symmetry plane at a velocity ratio of $VR=0.28$, $DR=1.53$, right: coherent structures indicated by the λ_2 criterion with mapped on mixture fraction f

5.5 Cooling efficiency

Following the heat-mass transfer analogy the penetration of the CO₂ jet into an air cross-flow mimics the density ratio between the cooling fluid and the hot boundary layer flow in a gas turbine. Hence, the distribution of the mean mixture fraction f along the flat plate downstream of the jet hole resembles the film-cooling efficiency. The contours of the mixture fraction f

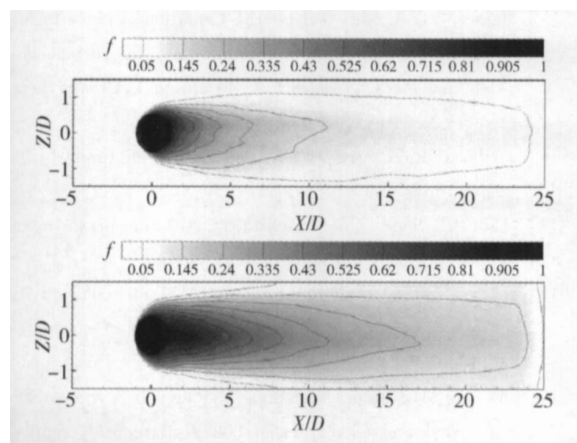


Fig. 10 Left: contours of the mean mixture fraction f along the flat plate at $MR=0.153$ (top) and at $MR=0.43$ (bottom); right: mean mixture fraction along the centerline at $MR=0.43$ compared with the film cooling efficiency η measured at $MR=0.5$ by Sinha et al.^[6]

The graph on the right shows the mean mixture fraction along the plate centerline as a function of the streamwise coordinate for the higher velocity ratio case $VR=0.28$ that corresponds to a mass flux ratio $MR=0.43$. The data is compared with the findings of Sinha et al.^[6]. The symbols represent the measured film cooling efficiency η , which is a temperature based variable. The mass fraction of the dense gas is related to the cooling efficiency for the analogous heat transfer situation. It is $\eta=f$ as stated in [4]. In the experiments the density ratio is $DR=2$ and $VR=0.25$, which leads to a slightly higher mass flux ratio of $MR=0.5$. Nevertheless, it can be concluded that the good agreement between both curves shows the mixture fraction distribution along the plate to correctly predict the film cooling efficiency of such flow configurations.

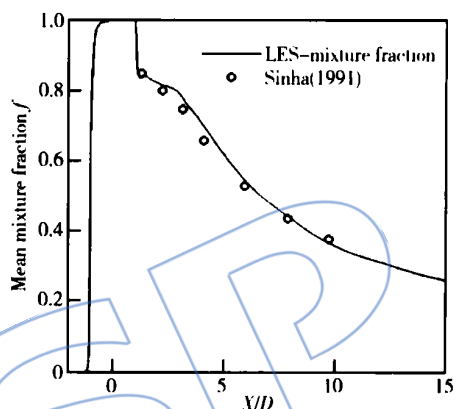
6 Conclusion

Large eddy simulations of the jet-in-a-cross-

flow problem have been performed to investigate in detail the impact of the density ratio parameter on the physics of film cooling flows. The high resolution of the computational domain, the time accuracy, and the possibility to track the passive mixture fraction scalar f of the cooling fluid allows a detailed analysis of the mean and the instantaneous flow field. The numerical simulations are conducted using the same flow parameters as the simultaneously performed PIV measurements. The LES predictions for the velocity distributions are in excellent agreement with the experimental data.

Variations of the velocity ratio parameter significantly impacts the flow field with respect to the dynamic separation and reattachment process, the turbulence statistics, and the cooling efficiency, whereas a variation of the density ratio has only a minor influence, if the velocity ratio is kept constant.

The flow physics is discussed by identifying



the dominant vortical structures. The formation of the CVP is generated by the initial shearing of the boundary layer by the jet. Finally, the cooling efficiency along the flat plate is identified by the mean mixture fraction distribution. Comparisons with findings from the literature convincingly show the validity of the heat-mass transfer analogy to investigate such flows. The evaluation of the cooling efficiency is strongly dependent on density effects and thus, the mass flux ratio has to be considered.

Acknowledgments The support of this research by the Deutsche Forschungsgemeinschaft (DFG) in the frame of SFB 561 is gratefully acknowledged.

References:

- [1] Andreopoulos K, Rodi W. Experimental investigation of jets in a crossflow[J]. *J Fluid Mech*, 1984, 138: 93-127.
- [2] Plesniak M, Cusano D. Scalar mixing in a confined rectangular jet in a crossflow[J]. *J Fluid Mech*, 2005, 524: 1-45.
- [3] Goldstein R J, Eckert E R G. Effects of hole geometry and density on three-dimensional film cooling[J]. *Int J Heat Mass Transfer*, 1974, 17: 595-607.
- [4] Pedersen Ø R, Eckert E R G, Goldstein R J. Film cooling with large density differences between the mainstream and the secondary fluid measures by the heat-mass transfer analogy[J]. *ASME J Turbomech*, 1977, 99: 620-627.
- [5] Pietrzyk J R, Bogard D G, Crawford M E. Effects of density ratio on the hydrodynamics of film cooling[J]. *ASME J Turbomech*, 1990, 112: 437-443.
- [6] Sinha A K, Bogard D G, Crawford M E. Film cooling effectiveness downstream of a single row of holes with variable density ratio[J]. *ASME J Turbomech*, 1991, 113: 442-449.
- [7] Hoda A, Acharya S. Predictions of a film coolant jet in crossflow with different turbulence models[J]. *ASME J Turbomech*, 2000, 122: 558-569.
- [8] Walters D L, Leyke J H. A detailed analysis of film cooling physics: Part I—streamwise injection with cylindrical holes[J]. *ASME J Turbomech*, 2000, 122: 102-112.
- [9] Tyagi M, Acharya S. Large Eddy Simulation of Film Cooling Flow From an Inclined Cylindrical Jet[J]. *ASME J Turbomech*, 2003, 125: 734-742.
- [10] Guo X, Meinke M, Schröder W. Large eddy simulation of film cooling flows[J]. *Comput Fluids*, 2006, 35: 587-606.
- [11] Iourokina I, Lele S K. Large eddy simulation of film cooling above the flat surface with a large plenum and short exit holes[R]. Technical Report 2006-1102, AIAA, 2006.
- [12] Guo X, Schröder W, Meinke M. LES of film cooling. In *International Gas Turbine Congress*[R]. Tokyo, Mar. 2003. Paper IGTC 2003TS 06.
- [13] Guo X, Meinke M, Schröder W. Flow prediction for film cooling by large eddy simulation[R]. In *The 10th of International Symposium on Transport Phenomena and Dynamics of Rotating Machinery*, Honolulu, Mar. 2004. Paper 157.
- [14] Guo X, Meinke M, Schröder W. Large eddy simulation of a jet in a crossflow[A]. In Geurts B, Friedrich R, and Metais O, editors, *Direct and Large Eddy Simulation V*[C]. pages 603-610. Kluwer, 2003.
- [15] Jessen W, Schröder W, Klaas M. Evolution of jets effusing from inclined holes into crossflow[A]. In *Turbulence, Heat and Mass Transfer 5*[C], Dubrovnik, Croatia, September 25-26th, 2006.
- [16] Wilke C R. A viscosity equation for gas mixtures[J]. *J. Chem. Phys.*, 1950, 18: 517.
- [17] Bird R B, Stewart W E, Lightfoot E N. *Transport Phenomena*[M]. John Wiley and Sons Inc., 1960.
- [18] Mathur S, Tondon P K, Saxena S C. Thermal conductivity of binary, ternary and quaternary mixtures of rare gases[J]. *Mol. Phys.*, 1967, 12: 569.
- [19] Meinke M, Schröder W, Krause E, Rister T. A comparison of second and sixth order methods for large eddy simulations[J]. *Comput Fluids*, 2002, 31: 695-718.
- [20] Fureby C, Grinstein F F. Monotonically integrated large eddy simulation of free shear flows[R]. AIAA J, 1999, 37: 544-556.
- [21] Grinstein F F, Fureby C. Recent progress on models for high reynolds number flows[J]. *J Fluids Engineering*, 2002, 124: 848-861.
- [22] Alkishriwi N, Meinke W S M. A large eddy simulation method for low mach number flows using preconditioning and multigrid[J]. *Comput Fluids*, 2006, 35 (10): 1126-1136.
- [23] Turkel E. Preconditioning techniques in computational fluid dynamics[J]. *Annu Rev Fluid Mech*, 1999, 31: 385-416.
- [24] El Askary W, Schröder W, Meinke M. LES of compressible wall bounded flows[R]. Technical Report 2003-3554, AIAA, 2003.
- [25] Ewert R, Schröder W, Meinke M, El Askary W. LES as a basis to determine sound emission[R]. AIAA paper 2002-0568, AIAA, 2002.
- [26] Peterson S D, Plesniak M W. Evolution of jets emanating from short holes into crossflow[J]. *J Fluid Mech*, 2004, 503: 57-91.
- [27] Kelso R M, Lim T T, Perry A E. An experimental study of round jets in a crossflow[J]. *J Fluid Mech*, 1996, 306: 111-144.
- [28] Morton B R, Ibbetson A. Jets deflected in a crossflow[J]. *Exp. Therm. Fluid Sci*, 1995, 12: 112-133.
- [29] Jeong J, Hussain F. On the identification of a vortex[J]. *J Fluid Mech*, 1995, 285: 69-94.