Conference on Turbulence and Interactions TI2006, May 29 - June 2, 2006, Porquerolles, France

LES OF TURBULENT MIXING IN FILM COOLING FLOWS

P. Renze[†,*], **M. Meinke, W. Schröder**

[†]Institute of Aerodynamics, RWTH Aachen University, 52062 Aachen, Germany
[*]Email: p.renze@aia.rwth-aachen.de

ABSTRACT

The 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 different species to simulate the non-reacting gas mixture of an air-like crossflow and a CO_2 *cooling jet. The parameters governing the flow field have been chosen to mimic the conditions in a gas turbine, i.e., the freestream Reynolds number is* $Re_\infty = 400,000$, *the streamwise inclination angle of the cooling jet injection is* $\alpha = 30^o$, *the velocity ratio ranges from* $VR = 0.1 - 0.48$, *and the ratio of boundary layer thickness to jet hole diameter is* $\delta/D = 2$. *An efficient method of solution for low subsonic flows is applied based on an implicit dual time-stepping scheme combined with low Mach number preconditioning. The comparison with experimental data shows an excellent agreement with respect to velocity profiles and film cooling efficiency. Moreover, the investigations evidence volume effects to dominate the flow field in the vicinity of the jet hole.*

INTRODUCTION

Film cooling techniques are applied in modern gas turbines to reduce the thermal loads on turbine components that result from the high inlet temperatures needed for a high thermal efficiency. 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 study the cooling film is generated by injecting a cooling fluid through a row of staggered holes drilled into the wall surface. 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. Plesniak and Cusano [1] provided an excellent summary of the influence of the inclination angle, blowing ratio, and hole geometry on the cooling efficiency.

Most numerical investigations of the JICF problem have been based on Reynolds-averaged Navier-Stokes equations, e.g., Hoda and Acharya [2] or Walters and Leylek [3]. Since the JICF problem is influenced by the effects of wall-bounded as well as free turbulence, most turbulence models cannot be applied without proper scaling of the coefficients. This, however, requires a-priori knowledge of the flow field. For this reason, it is necessary to apply a more general numerical ansatz such as LES to investigate JICF problems. Such an analysis was performed, for instance, by Guo *et al.* [4] who investigated among other issues the effects of inclination angle and blowing ratio on the flow field.

In the present paper the impact of the density ratio between coolant and crossflow is analyzed by LES and compared with experimental data. However, experimental investigations of turbine blade flows at real temperature levels require an extremely complex and expensive wind tunnel equipment. Therefore, the effect of density differences between the coolant and the crossflow are simulated by a CO_2 injection to mimic a density ratio of 1.6.

NUMERICAL METHOD

The three-dimensional Navier-Stokes equations plus separate transport equations for individual species of an ideal gas mixture are solved using an LES method. The discretization of the governing equations is based on a mixed central-upwind AUSM (advective upstream splitting method) scheme with low numerical dissipation, see Meinke *et al.* [5].

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 and preconditioning methods [6]. The matrix of the preconditioned scheme has been modified to account for the additional mass-fraction equations.

The simulation of the flat plate boundary layer requires instantaneous turbulent flow data that can be prescribed at the inflow boundary. Therefore, a combined rescaling and slicing method is applied. That is, the flow variables at the entrance of the domain are obtained by an independent spatially developing boundary layer simulation (Fig. 1).

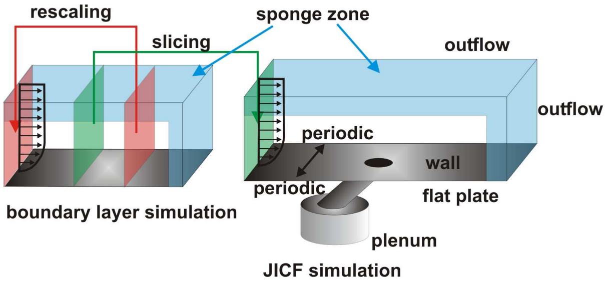

Fig. 1. Schematic of the computational domain and the boundary conditions

This auxiliary simulation generates its own turbulent inflow data using the compressible rescaling method proposed by El-Askary *et al.* [7]. The rescaling method is a means of approximating the properties at the inlet via a similarity approach applied to the downstream solution.

The resulting turbulent boundary layer simulation matches very well the flow physics as demonstrated in Fig. 2. The comparison of the turbulent intensities at a momentum thickness based Reynolds number of $Re_\theta = 1850$ with data from Lund and Wu [8] evidences the good agreement.

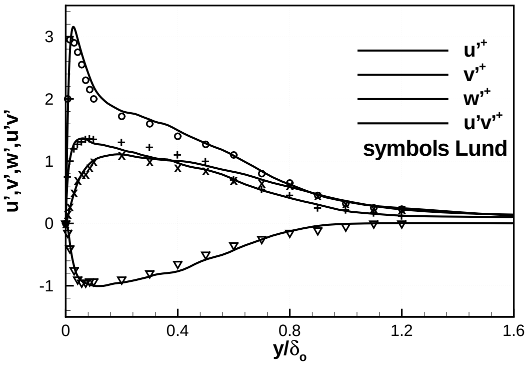

Fig. 2. Turbulent intensities scaled in inner wall units compared with data from Lund and Wu [8]

RESULTS

The discussion of the results consists of a comparison of numerical and experimental data to show the quality of the computational method. Then the cooling efficiency is analyzed, before the impact of the density ratio and the mixing process are investigated.

Comparisons with Measurements

The large-eddy simulation 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.* [9]. To obtain the time-averaged statistics the flow field has been sampled over 8 time periods. Here, one period is the time that the crossflow fluid needs to pass over the length of the plate.
In Fig. 3 profiles of the streamwise velocity are compared for the case of a CO_2 jet injection into the boundary layer of an air stream 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 $X/D = 2$. At both velocity ratios the predicted flow field is in excellent agreement with the PIV measurements.

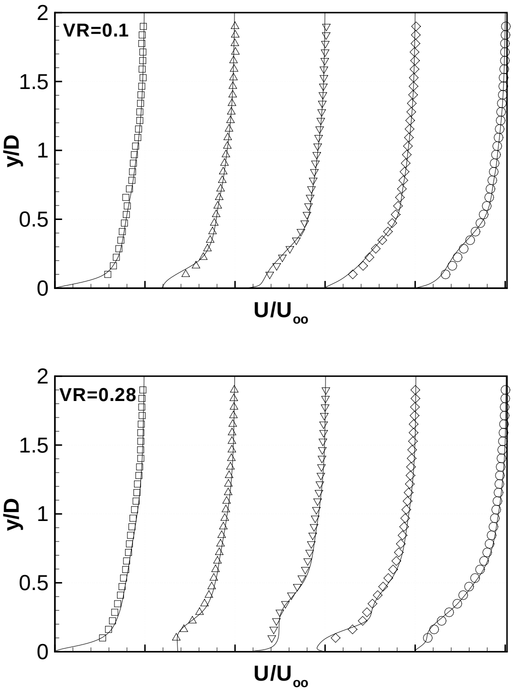

Fig. 3. Velocity profiles in the symmetry plane at different streamwise locations, top: $VR = 0.1$, bottom: $VR = 0.28$, experimental data $\square : x/D = -1$, $\triangle : x/D = 0$, $\nabla : x/D = 1$, $\diamond : x/D = 1.5$, $\circ : x/D = 2$, solid lines: LES data.

Prediction of Cooling Efficiency

The penetration of the CO_2 jet into an air-like crossflow mimics the density ratio between the cooling fluid and the hot gas stream in a gas turbine. Hence, the distribution of the CO_2 mixture fraction f along the flat plate downstream of the jet hole resembles the film-cooling effectiveness. The contours of the mixture fraction f along the plate at a velocity ratio of $VR = 0.28$ and a mass flux ratio of $MR = 0.45$, respectively, are shown in Fig. 4. The graph shows the mean mixture fraction in the spanwise symmetry plane as a function of the streamwise coordinate. The data is compared with the findings of Sinha *et al.* [10]. The symbols represent the measured film cooling effectiveness η, which is a temperature based variable, at a slightly different mass flux ratio of $MR = 0.5$. 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.

Effect of Velocity and Mass Flux Ratio

Large-eddy simulations have been performed at different velocity ratios $VR = 0.1,\ 0.28,$ and 0.48. A variation of the velocity ratio has a large impact on the depth of the jet penetration into the crossflow and the size of the recirculation region. However, a variation of the mass flux at a constant velocity ratio has only little effect on the dynamics of the flow field in the vicinity of the jet hole, as shown in Fig. 5. Mach number contours and streamlines are extracted in the JICF symmetry plane of an air-to-air and a CO_2-to-air injection at an equal velocity ratio of $VR = 0.28$. The size of the recirculation region and the penetration depth are almost identical.

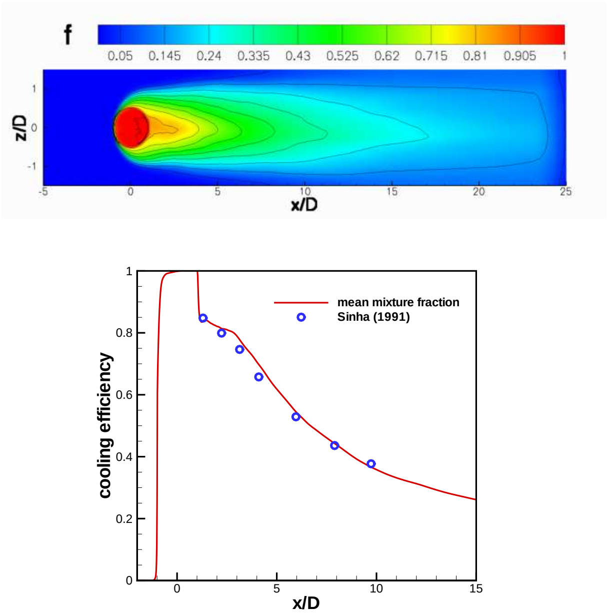

Fig. 4. Top: contours of the mean mixture fraction along the flat plate, bottom: mean mixture fraction in the spanwise symmetry plane compared with film cooling efficiency η measured by Sinha *et al.* [10]

Coherent Structures and Mixing Process

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. The CVP dominates the mixing process of the mixture fraction scalar in the boundary layer. The development of the CVP is visualized in Fig. 6 showing two cross sections within the recirculation region at the same instantaneous time level. The vorticity, which originates from the spanwise edges of the jet hole, causes the counter-rotating eddies governing the initial mixing between cooling fluid and crossflow. Thus, high density jet fluid is lifted off the plate at the trailing edge and boundary layer fluid is entrained between the plate and the jet.

The separation and reattachment of the jet as well as the turbulent diffusion and convection process downstream of the injection can be observed in Fig. 7. The mixture fraction field in the $Z/D = 0$ plane and the corresponding coherent vortical structures are shown at the same instantaneous time level.

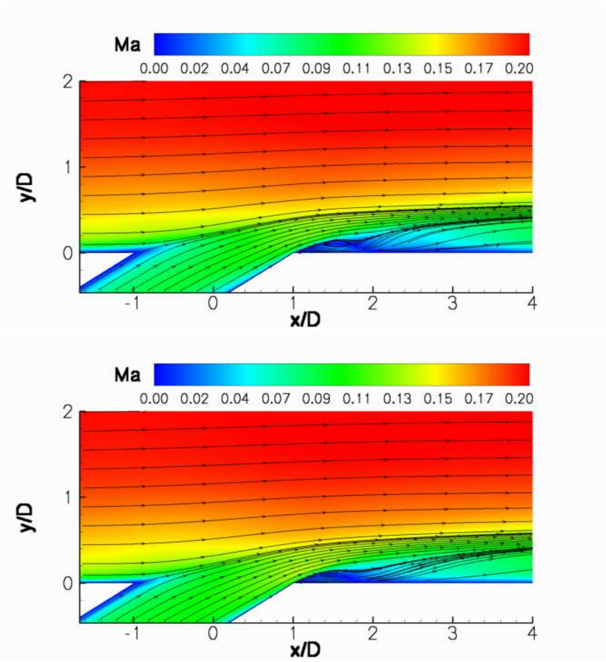

Fig. 5. Mach number contours in the JICF symmetry plane with streamlines at a velocity ratio of $VR = 0.28$, top: air injection at $MR = 0.28$, bottom: CO_2 injection at $MR = 0.45$

CONCLUSION

An LES is applied to simulate the penetration of a cooling jet into a turbulent boundary layer. The time-averaged LES predictions are in excellent agreement with experimental data. The flow physics is discussed by identifying the dominant vortical structures. The simulations of different velocity and mass flux ratios show volume effects to dominate the flow field in the vicinity of the jet hole. A detailed study of the mixture fraction scalar distribution proves the significance of the coherent structures in the dynamic flow field on the mixing process.

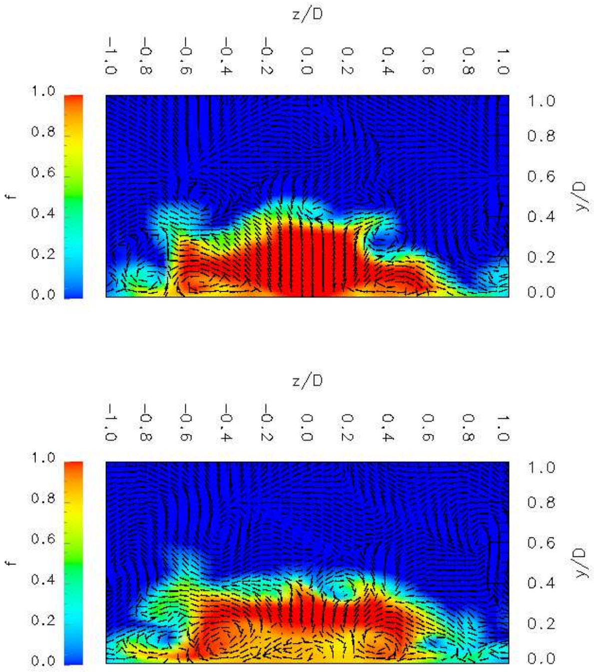

Fig. 6. Instantaneous mixture fraction contours and velocity vectors at $X/D = 1$ (top) and $X/D = 1.5$ (bottom)

BIBLIOGRAPHY

[1] M. W. Plesniak, D.M. Cusano, ”Scalar mixing in a confined rectangular jet in a crossflow”, J. Fluid Mech., vol. 524, pp. 1-45, 2005.

[2] A. Hoda, S. Acharya, ”Predictions of a film coolant jet in crossflow with different turbulence models”, ASME J. Turbomach., vol. 122, pp. 558-569, 2000.

[3] D.K. Walters, J.H. Leylek, ”A detailed analysis of film-cooling physics part 1: streamwise injection with cylindrical holes”, ASME J. Turbomach., vol. 122, pp. 102-112, 2000.

[4] X. Guo, M. Meinke, W. Schröder, ”Large-eddy simulation of film cooling flows”, Comput. Fluids, vol. 35, pp. 587-606, 2006.

[5] M. Meinke, W. Schröder, E. Krause, T. Rister, ”A comparison of second- and sixth-order methods for large-eddy simulations”, Comput. Fluids, vol. 31, pp. 695-718, 2002.

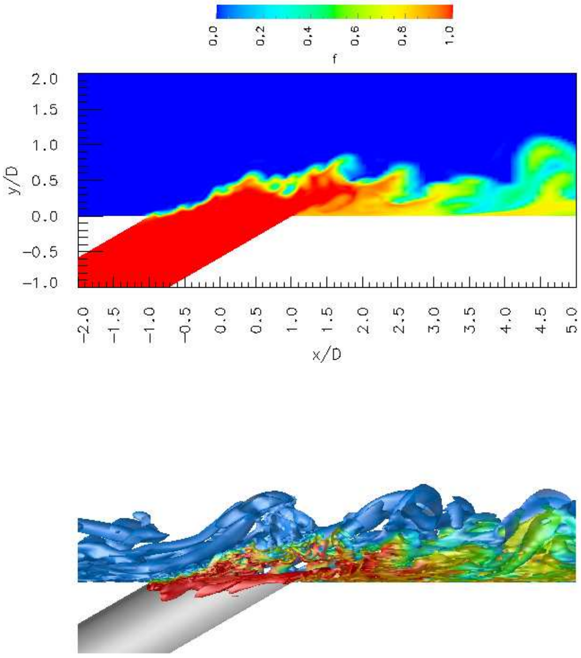

Fig. 7. Top: mixture fraction contours in the symmetry plane at a velocity ratio of $VR = 0.28$, bottom: coherent structures indicated by the λ_2 criterion and mapped mixture fraction f

[6] N. Alkishriwi, M. Meinke, W. Schröder, ”A large-eddy simulation method for low mach number flows using preconditioning and multigrid”, Comput. Fluids, in press, 2006.

[7] W.A. El-Askary, W. Schröder, M. Meinke, ”LES of compressible wall-bounded flows”, AIAA paper 2003-3554

[8] T.S. Lund, X.S. Wu, ”Generation of turbulent inflow data for spatially developing boundary layer simulations”, J. Comput. Phys., vol. 140, pp. 233-258, 1998.

[9] W. Jessen, W. Schröder, M. Klaas, ”Evolution of jets effusing from inclined holes into crossflow”, submitted to Turbulence, Heat and Mass Transfer, 5, Dubrovnik, 2006.

[10] A.K. Sinha, D.G. Bogard, M.E. Crawford, ”Film-cooling effectiveness downstream of a single row of holes with variable density ratio”, ASME J. Turbomach., vol. 113, pp. 442-449, 1991.