TRANSACTIONS OF THE INSTITUTE OF FLUID-FLOW MACHINERY

No. 113, 2003, 201-??

MICHAŁ KARCZ* and JANUSZ BADUR

An alternative turbulent heat flux modelling for gas turbine cooling application

Institute of Fluid-Flow Machinery Polish Academy of Sciences, Thermo-Chemical Power Department, Fiszera 14, 80-952 Gdańsk, Poland

Abstract

The paper deals with the implementation of two-equation turbulent heat flux closures in the form proposed by Abe et al. and Deng et al., combined with the second order turbulence model for momentum field for gas cooling application. Implemented models have been compared with the experimental data by Gartshore et al. and Kohli et al. concerning the flat plate cooling with a compound angle orientation. It has also been compared with the standard solution of turbulent heat flux by means of a constant and variable turbulent Prandtl number.

Keywords: Cooling of gas turbine elements; Cooling system

1 Introduction

The need of cooling of gas turbine elements that are highly mechanically and thermally loaded is obvious. Temperature levels that occur in a modern gas turbine blade path are often higher then temperature of the melting point of element material. There are two common ways to protect gas turbine elements from thermal over-loading, namely internal and external cooling. Internal cooling systems consist mainly of ribbed U-bend ducts located inside of a blade. External cooling is connected with jets of coolants injected into a hot mainstream by a system of holes.

The cooling systems involve many of physical features that are not fully recognized and which are still very difficult to calculate even with the sophisticated models. Development of computational methods have lead to the rapid growth of

*Corresponding author. E-mail address: mkarcz@imp.gda.pl

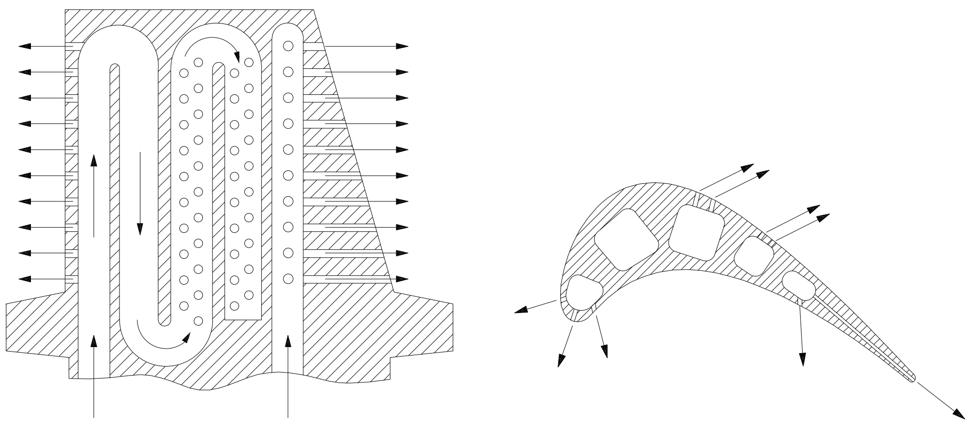

Figure 1. Internal and external cooling systems [2]

a number of publications concerning internal and external cooling of gas turbine. Most of them have employed two-equation turbulent momentum flux closures combined with the constant turbulent Prandtl number assumption for the turbulent heat flux modeling. Such a way is very robust and effective, but recently some papers underline that only second order closures could be succesfully implemented for gas turbine cooling flow prediction [6, 12]. It is connected with the lack of the eddy viscosity two-equation closures to cope with prediction of secondary flows generated by rotation or strong streamline curvature.

Also the turbulent Prandtl number assumption couldn't be sufficient when complicated three-dimensional flows with heat transfer are analysed [8]. It is known that from numerical point of view the order of turbulent heat flux closures should be equal or at least only one order lower than turbulent momentum flux closure. When the Reynold stress turbulence model is employed then the two-equation $\overline{\theta'^2}$–ϵ_θ heat flux model seems to be most suitable [13].

2 Computational model

Present analysis bases on some experimental results presented by Gartshore et al. [5]. They have considered two geometrical cases of compound-oriented holes on a flat plate – one square and the second round shaped, and three flow cases represented by the velocity ratio $VR = v_{jet}/v_\infty$ namely 0.5, 1.0 and 1.5. The compound angle employed in [5] involve a common value of inclination angle α that was equal 30° and a lateral angle $\beta = 45°$. Present numerical analysis has been limited only to round shaped hole.

The computational domain is presented in Fig. 2. It has consisted of two structured grid blocks – one for the cooling channel and second for the mainstream above the plate. Computational domain has been limited in such a way that it

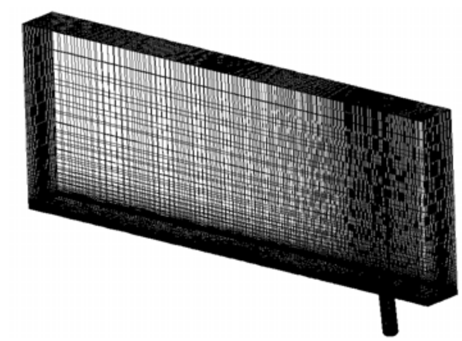

Figure 2. Computational grid for flat plate cooling

involves one hole only, so the periodic condition at the side surfaces could be naturally applied. Then the width of the domain has been equal $3d$, its height $25d$ and length $35d$, where d means hole diameter. Hole has been located at the distance $5d$ from the inlet of the domain. The lenght of channel that feeds the hole with coolant has been equal $4d$.

Numerical procedure Commercial computational fluid dynamic package FLUENT [4] has been employed for the numerical analysis. It is based on the finite volume method. For different cases we have used both standard and personally modified solver. The modifications have concerned changes of turbulent heat flux definition and the material properties.

The working medium – air, has been treated as an ideal fluid, and its molecular viscosity μ, thermal conductivity λ and heat capacity at constant pressure c_p have been modelled with the aid of temperature polynomial functions. The governing transport equations have been discretized with the second order scheme. The pressure-velocity coupling has been achieved by means of the SIMPLEC method.

Velocity components, temperature, turbulent kinetic energy and its dissipation rate profiles for fully developed turbulent flow have been applied at the inlets of mainstream and coolant.

It has been necessary to perform some grid-dependence studies. Solution that seems to be independent from grid size has been obtained after third subsequent adaption. The total size of grid after adapting arise from 100 thousands to over 180 thousands computational cells. The results of calculations for different grids are presented in Fig. 3 by means of changes of the coefficient cooling effectivness η (**??**) at cross-section location $x/d = 5$.

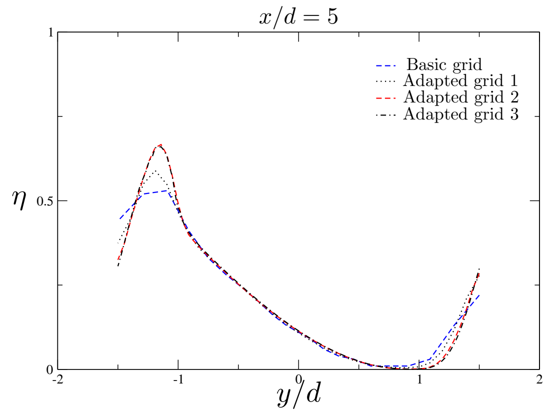

Figure 3. Cooling effectivness η at the cross-section $x/d = 5$ after subsequent grid adaption

Model implementation The standard FLUENT's second order RSM turbulence closure has been employed for the velocity field modelling. This model has been validated and widely tested for different three-dimensional experiments [7].

Turbulent heat flux is usually defined on the ground of the simple gradient hypothesis as:

$$q_i^t = -\overline{v_i'\theta'} = \alpha_t \frac{\partial T}{\partial x_i} \qquad i = x, y, z \quad , \tag{1}$$

where the concept of turbulent heat diffusivity α_t is introduced. Common practice in computational modeling is employing an assumption of some analogy between heat and momentum transfer that is called a Reynolds analogy. It directly leeds to the simple turbulent Prandtl number definition that corelates the turbulent diffusivity of heat α_t with known turbulent viscosity ν_t:

$$\alpha_t = \frac{\nu_t}{\mathrm{Pr}_t} \quad . \tag{2}$$

Usually a constant Pr_t is invoked. Standard values of the turbulent Prandtl number applied in almost all CFD codes are near unity and in FLUENT code it is equal $\mathrm{Pr}_t = 0.85$.

The modification of turbulent heat flux in such configuration could be done only in two ways – by the algebraic modification of turbulent Prandtl number (1) or by the changing of the definition of α_t (2).

The turbulent Prandtl number has been applied in the present analysis as:

Case A standard constant value $\mathrm{Pr}_t = 0.85$ [4],

Case B algebraic formulae $\mathrm{Pr}_t = var$ in the form proposed by Kays and Crawford [8]:

$$\frac{1}{0.5882 + 0.228\,(\nu_t/\nu) - 0.0441\,(\nu_t/\nu)^2\left[1 - \exp\left(\frac{-5.165}{(\nu_t/\nu)}\right)\right]} \quad , \tag{3}$$

These are the simplest propositions that are based on the empiricism. More advanced models use additional transport equations that allow to calculate turbulent thermal time scale $\overline{\theta'^2}/2\epsilon_\theta$ that together with turbulent mechanical time scale k/ϵ could be incorporated into the formulae (2), [1, 3, 7].

As a consequence in the present analysis the turbulent heat diffusivity α_t have been modeled as:

Case C DWX two-equation model

$$\alpha_t = C_\lambda f_\lambda k\,(k/\epsilon)^l\left(\overline{\theta'^2}/\epsilon_\theta\right)^m \quad , \tag{4}$$

where temperature variation $\overline{\theta'^2}$ and its destruction rate ϵ_θ are obtained from an additional tranport equation in the form proposed by Deng et al. [3],

Case D AKN two-equation model

$$\begin{aligned}\alpha_t =& C_\lambda\left[\frac{k^2}{\epsilon}\left(\frac{2R}{C_m + R}\right) + 3k^{\frac{1}{2}}\left(\frac{\nu^3}{\epsilon}\right)^{\frac{1}{4}}\frac{(2R)^{\frac{1}{2}}}{\mathrm{Pr}}f_d\right]\times \\ &\times\left[1 - \exp\left(-\frac{y^*}{14}\right)\right]\left[1 - \exp\left(-\frac{\mathrm{Pr}^{\frac{1}{2}}y^*}{14}\right)\right] \quad ,\end{aligned} \tag{5}$$

where temperature variation $\overline{\theta'^2}$ and its destruction rate jej ϵ_θ, that are involved in turbulent scale ratio defined as $R = (k/\epsilon)\,/\left(\overline{\theta'^2}/2\epsilon_\theta\right)$, are obtained from additional tranport equations in the form presented by Abe et al. [1].

All modifications presented in this paper are introduced to the solver by the user defined subroutines [7].

3 Analysis of the results

There are many ways to analyse the film cooling effects. Two of them are usually employed namely heat transfer coefficient h and the film cooling effectivness η. In the present analysis film cooling effectivness have been adopted. It is defined as [2, 5, 7, 9, 10, 11]:

$$\eta = \frac{T - T_\infty}{T_{jet} - T_\infty} \quad , \tag{6}$$

where T represents a local static temperature, T_{jet} is the temperature of the

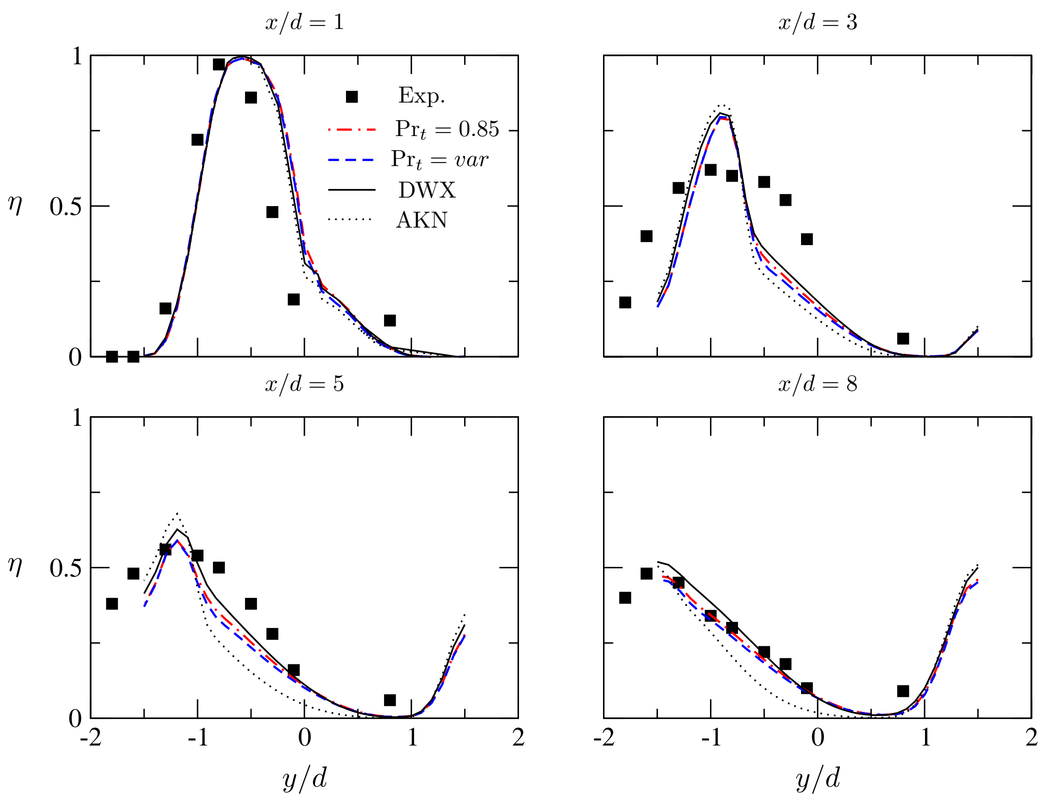

Figure 4. Cooling effectivness for velocity ratio $VR = 0.5$

The results of spanwise distribution of η coefficient for different velocity ratios VR and different implemented models will be presented at the subsequent diagrams, for four location of duct cross-sections from the cooling hole outlet, namely $x/d = 1$, 3, 5 and 8.

The results for velocity ratio $VR = 0.5$ are presented in Fig. **??**. One can see that the strong similarity exists between computational and experimental data. Especially at the distance $x/d = 1$ and 8 numerical results show excellent agreement with the measurements. All models work in the same manner, with only slight differences observed for the AKN model far away from the injection hole outlet. Similar conclusions could be drawn for the velocity ratio $VR = 1.0$

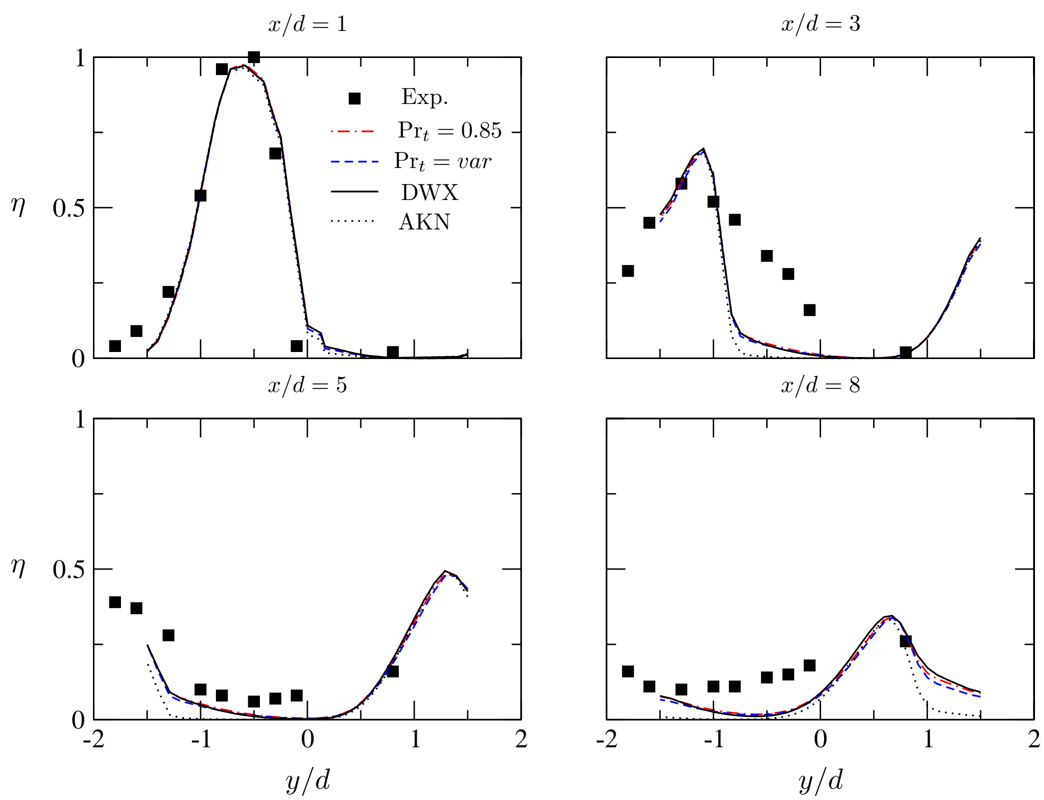

Figure 5. Cooling effectivness for velocity ratio $VR = 1.0$

and $VR = 1.5$ presented in Figs. **??** and **??**, respectively. For all cases the biggest diferences are noticed at the cross-section $x/d = 3$, where the shape of η doesn't fit the experimental data. The probable reason for this could be the effect of underestimation of heat diffusion intensity in the spanwise direction. The underestimation of spanwise turbulent heat diffusion has been also observed by Lakehal et al. [11] and confirmed in report [7] for computation of flat plate film cooling with different geometrical configuration of cooling hole, namely $\alpha = 30°$ and $\beta = 90°$.

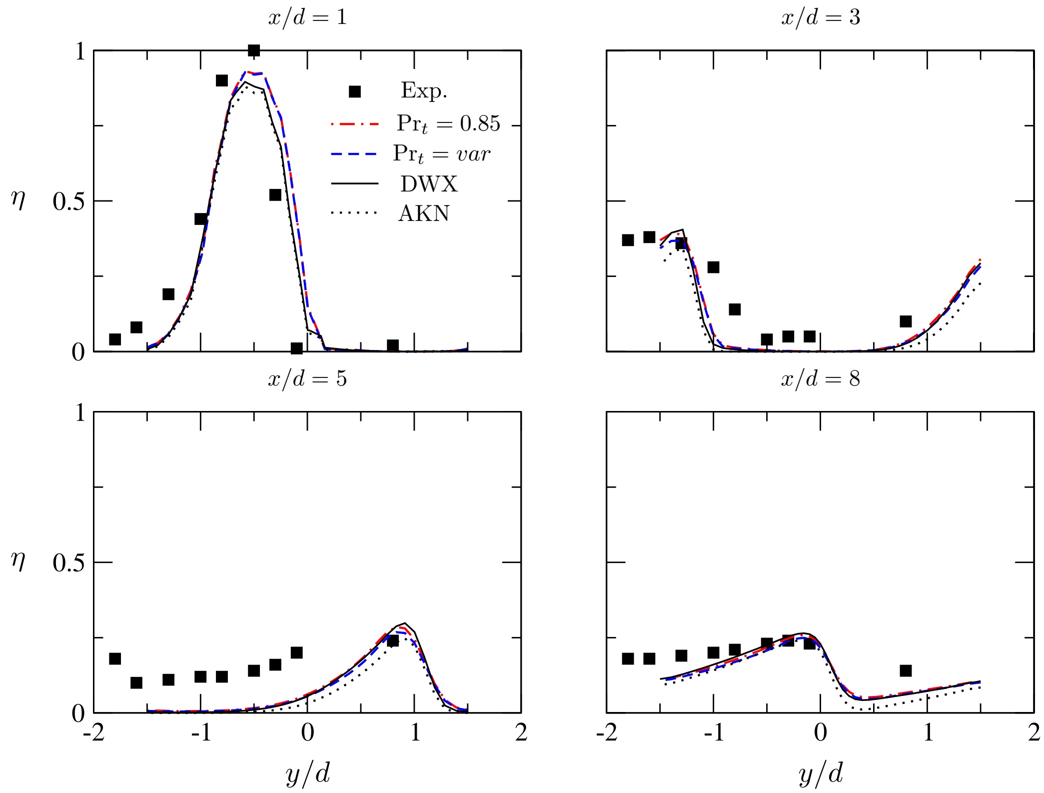

Figure 6. Cooling effectivness for velocity ratio $VR = 1.5$

The contours of cooling effectivness coefficient η on the protected plate for velocity ratios $VR = 0.5$ and $VR = 1.5$ are presented in Fig. **??**. There is a strong flow separation at the vicinity of cooling hole due to the higher velocity of injected air $VR = 1.5$ at Fig. **??** b). As a result a weak cooling effectivness of the plate is observed close to the place of injection. At some distance cooling medium starts spreading out in the spanwise direction. As a consequence larger area of the protected plate is insulated, that reveal in more uniform distribution of the film cooling effectivness. For the lower velocity $VR = 0.5$, at Fig. **??** (a), the cooling jet maintain its profile at the whole lenght of plate. This allows for better thermal protection immediately behind the cooling hole but, on the other hand, there exists a strong nonuniformity of η distribution at the some distance far from injection point. Deviation of flow direction, for both velocity ratios, are connected with the mainstream-jet, that is injected with the angle $\beta = 45°$, interaction. Such interaction cause also emerging of single vortex[1] and rising intensity of temperature fluctations [9, 10].

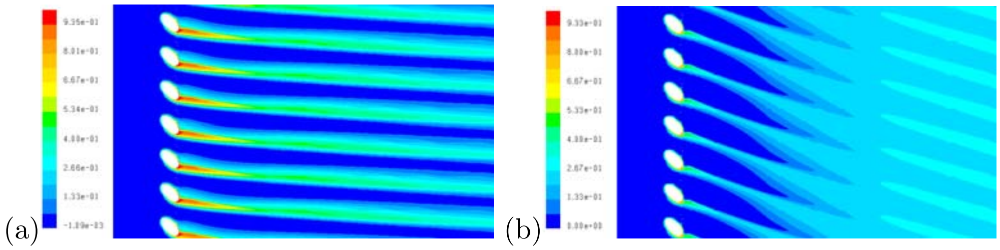

Figure 7. Distribution of cooling effectivness, (a) $VR = 0.5$ and (b) $VR = 1.5$

The contour of the root-mean-sqaure temperature fluctuation θ'_{rms}, normalized by the formulae:

$$\theta'_{rms} = \frac{\sqrt{\overline{\theta'^2}}}{(T_{jet} - T_\infty)} \quad , \tag{7}$$

at the vicinity of cooling air injection hole is presented at Fig. **??**. The highest level of temperature fluctuation could be observed in the region of attack edge of cooling hole and far at the interface between the coolant jet and mainstream, where the velocity and temperature gradients are greatest, and turbulent mixing process is most intensive.

The maximum observed value for the highest velocity ratio $VR = 1.5$ is equal $\theta'_{rms} = 0.28$, that is close to the experimental data presented by Kohli et al. [9, 10], where the measured level of temperature fluctuation has been estimated as high as $\theta'_{rms} \geq 0.25$. Such level has been independent of turbulence level applied at inlet. The localisation of maximum is almost identical, despite of some differences between the geometry of present numerical and former experimental domain which were prepared for the same inclination angle $\alpha = 30°$ but different lateral angle $\beta = 0°$. Similar results were obtained for the calculation with lateral angle $\beta = 90°$ [7]. The strong interaction on the interface between cooling jet and mainstream are explained by instability of shear layer, that generates some large-scale eddy structures, which are responsible for the fast dilution of the coolant jet by the hot mainstream [9, 10], so the rapid drop of cooling effectivness is observed close to the injection hole.

[1] Counter rotating vortex pair are characteristic for lower value of lateral angle [5]

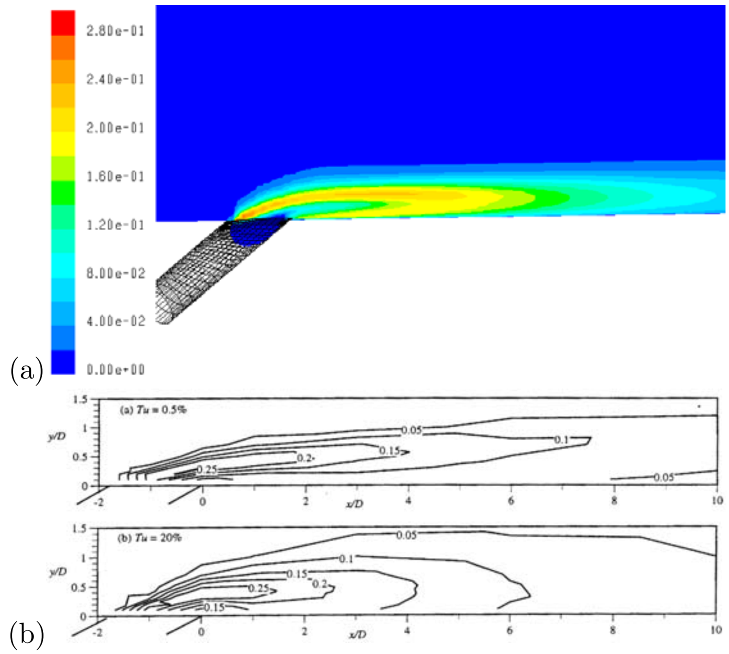

Figure 8. Contour of θ'_{rms} for velocity ratio $VR = 1.5$ i $Tu = 5\%$, (a) results of computation with DWX model, (b) experimental data by Kohli et al. [9, 10]

4 Summary

The second order RSM closure, employing for velocity field prediction, gives a much better agreement with the experimental data, than the results obtained with the two-equation k–ϵ turbulence closure presented by Gartshore et al. [5]. Particular enhancement are noticed at the cross-sections located far away from the coolant inlet.

More advanced turbulent heat flux closures are still needed. Two equation models, that are based on the temperature variation $\overline{\theta'^2}$ and its destruction rate ϵ_θ transport equations, have become reliable alternative for the standard constant turbulent Prandtl number assumption, without losing film cooling effectivness estimation quality. Some disadvantages due to increasing computation time by employing additional equations are recompensed by obtaining additional information about the flow structure.

The interesting results concern the distribution of normalized temperature fluctuation θ'_{rms}, calculated on the base of temperature variance equation $\overline{\theta'^2}$. Good agreement has been achieved with the experimental results of Kohli et al. [9 ,10], both in the level of θ'_{rms} and its localization.

Acknowledgement This research has been supported by the State Committee of Scientific Research (KBN) under the contract PB 1733/T10/2000/19.

Received 30 May 2003

References

[1] Abe K., Kondoh T., and Nagano Y.: *A new turbulence model for predicting fluid Flow and heat transfer in separating and reattaching flows. Part 2. Thermal field calculations*, Int. J. of Heat and Mass Transfer, Vo. 38(1995), 1467-1481.

[2] Chmielniak T., Rusin A., and K. Czwiertnia: *Gas Turbines*, Zakład Narodowy im. Ossolińskich, Wrocław 2001 (in Polish).

[3] Deng B., Wu W., and Xi S.: *A near-wall two-equation heat transfer model for wall turbulent flows*, Int. J. of Heat and Mass Transfer, Vol. 24(2000), 691-698.

[4] *FLUENT User's Guide*, Fluent Inc., Lebanon, USA, 1997-2002

[5] Gartshore I., Salcudean M., and Hassan I.: *Film cooling injection hole geometry: hole shape comparison for cooling orientation*, AIAA Journal, Vol. 39(2001), No. 8, 1493-1499.

[6] Iacovides H., and Launder B.E.: *Computational fluid dynamics applied to internal gas-turbine blade cooling: a review*, Int. J. of Heat and Fluid Flow, Vol. 16(1995), 454-470.

[7] Karcz M.: *Performance Analysis of the Thermal Diffusers in the Sense of Gas Turbine Cooling*, PhD Thesis, Institute of Fluid-Flow Machinery PAS, Thermo-Chemical Power Department, Gdańsk 2003 (in Polish).

[8] Kays W.M.: *Turbulent Prandtl number – where are we?*, ASME J. of Heat Transfer, Vol. 116(1994), 284-295.

[9] Kohli A., and Bogard D.G.: *Effects of very high free-stream turbulence on the jet-mainstream interaction in a film cooling flow*, ASME J. of Turbomachinery, Vol.120(1998), 785-790.

[10] Kohli A., and Bogard D.G.: *Fluctuating thermal field in the near-hole region for film cooling flows*, ASME J. of Turbomachinery, Vol. 120(1998), 86-91.

[11] Lakehal D., Theodoridis G.S., and Rodi W.: *Computation of film cooling of a flat plate by lateral injection from a row of holes*, Int. J. of Heat and Fluid Flow, Vol. 19(1998), 418-430.

[12] Simoneau R.J., and Simon F.F.: *Progress towards understanding and predicting heat transfer in the turbine gas path*, Int. J. of Heat and Fluid Flow, Vol. 14(1993), No. 2, 104-128.

[13] Sommer T.P., So R.M.C., and Zhang H.S.: *Heat transfer modeling and the assumption of zero wall temperature fluctuations*, ASME J. of Heat Transfer, Vol. 116(1994), 855-863.