THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS
345 E. 47th St., New York, N.Y. 10017

98-GT-204

CALCULATION OF CONVECTIVE HEAT TRANSFER IN SQUARE-SECTIONED GAS TURBINE BLADE COOLING CHANNELS

Arash Saidi

Bengt Sundén

Division of Heat Transfer
Lund Institute of Technology
Box 118, 221 00 Lund,
Sweden
Tel: +46 46 222 86 10
Fax: +46 46 222 86 12
bengts@emvox2.vok.lth.se

ABSTRACT

Internal cooling channels are commonly used to reduce the thermal loads on the gas turbine blades to improve overall efficiency. In this study a numerical investigation has been carried out to provide a validated and consistent method to deal with the prediction of the fluid flow and the heat transfer of such channels with square cross sections. The rotation modified Navier-Stokes and energy equations together with a low-Re number version of the k-ε turbulence model are solved with appropriate boundary conditions. The solution procedure is based on a numerical method using a collocated grid, and the pressure-velocity coupling is handled by the SIMPLEC algorithm. The computations are performed with the assumption of fully developed periodic conditions.

The calculations are carried out for smooth ducts with and without rotation and effects of rotation on the heat transfer are described. Similar numerical calculations have carried out for channels with rib-roughened walls. The obtained results are compared with available experimental data and empirical correlations for the heat transfer rate and the friction factor. Some details of the flow and heat transfer fields are also presented.

NOMENCLATURE

C_t	Constant in GGDH formulation
$C_{\varepsilon 1}, C_{\varepsilon 2}, C_\mu$	Turbulence model constants
D	Hydraulic diameter
Nu	Nusselt number
P	Pressure
P^*	Periodic pressure
Pr_k	Turbulence Prandtl number for k
Pr_ε	Turbulence Prandtl number for ε
R_b	Base radius
Re_t	Turbulence Reynolds number, Eq.(14)
Ro	Rossby number, DΩ/U
T	Mean temperature
U,V,W	Components of mean velocity
e	Rib height
f	Friction factor
f_ε, f_μ	Damping functions
n	Wall normal distance
p	Rib pitch, distance between ribs
t	Fluctuating temperature
u_ε	Kolmogorov velocity scale
u,v,w	Fluctuating velocity components
y^*	Wall Reynolds number, Eq. (15)

Greek symbols

Ω	Rotational speed
Ψ	Periodic source term, Eq. (7)
α	Thermal diffusivity
λ	Temperature gradient, Eq. (8)
μ	Kinematic viscosity
μ_t	Turbulence viscosity
θ	Dimensionless temperature
ρ	Density

Subscripts

b	Bulk
w	Wall

INTRODUCTION

In recent years economical and environmental concerns forced power generation technology to achieve higher levels of efficiency. In order to increase the performance of gas turbine cycles it is desirable to raise the temperature of the gas entering the turbine. However, this increase is limited by the mechanical and thermal characteristics of the materials used in gas turbine blades. It has been shown that the rate of heat transfer from blades can considerably affect the life time.

Presented at the International Gas Turbine & Aeroengine Congress & Exhibition Stockholm, Sweden — June 2–June 5, 1998

To solve this problem, starting from 60s, cooling methods for blades were introduced. One of the more recent methods applied is providing internal cooling channels in the blades, see Fig. 1. There are various types of such channels, but in this investigation attention is focused on square ducts with and without embedded ribs.

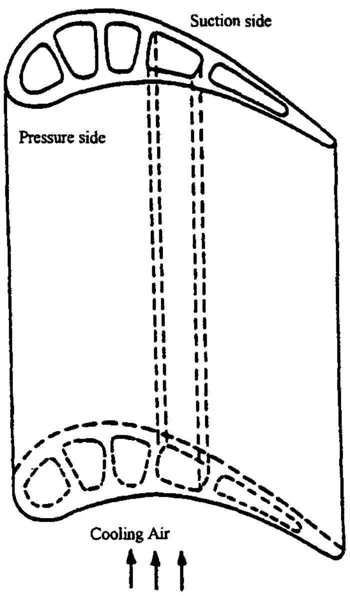

Figure 1. Turbine blade internal cooling passages.

Regarding the rotating feature of the problem, required experimental set ups could be very sophisticated. At the same time, performed experimental works have given the integral characteristics of the heat transfer, but it is shown that using Computational Fluid Dynamics detailed information concerning the fluid flow structure and the temperature field may be achieved.

The fluid flow structure inside such ducts is quite complex and subject to body forces due to the rotation, namely the coriolis and centrifugal-buoyancy forces. These forces create the secondary flows of Prandtl's first type in addition to the secondary flows of second type which are due to the turbulence and appear in non-circular duct flows. The secondary flows which are caused by the coriolis and centrifugal-buoyancy forces have impacts on the flow field structure (and consequently on the heat transfer), and they also affect turbulence through modification of its production.

Literature survey shows that there are some experimental and theoretical investigations of this problem. Hwang and Kuo (1993) have done experiments in rotating smooth square-sectioned ducts and obtained mean heat transfer rates for different speeds of rotation. Further investigations have been carried out on ribbed square channels with different model orientations by Johnson *et al.* (1994) and Parsons *et al.* (1995), and they obtained the heat transfer rates for the leading and the trailing faces, with and without rotation. There are also experimental works on these kinds of channels without consideration of rotation effects. For example, Han (1988) performed an investigation on channels with a range of different roughness, pitch and aspect ratios. Liou and Hwang (1993) examined the effect of rib shapes on the heat transfer rate, and both of the works suggest experimental correlations for the friction factor and the heat transfer rates.

Computational results are reported for the fully developed flow through a square-sectioned duct rotating in orthogonal mode by Iacovides and Launder (1987) in which they used the standard k-ε and also an algebraic stress model. Prakash and Zerkle (1991) calculated the developing flow inside a square-sectioned channel and took into account both rotation induced forces, namely centrifugal-buoyancy and coriolis, and they used the standard k-ε model together with wall functions. Younis (1993) used non-linear strain-stress relations instead of simple eddy diffusivity concept to capture the turbulence induced secondary flows in addition to secondary flows generated by rotation which were already obtained by previous methods. Dutta *et al.* (1994) made calculations for a developing flow inside a smooth rotating channel with the aid of some type of coriolis modified turbulence models. Bo *et al.* (1995) have carried out an extensive investigation on the effect of density variation and its effect on centrifugal-buoyancy force in developing flow cases. Prakash and Zerkle (1995) investigated the fluid flow and the heat transfer characteristics of ribbed ducts with standard k-ε model and wall functions.

Consideration of the works already published shows that different research groups are focusing on different aspects of this challenging problem. One area of the future research might be the application of the state-of-the-art turbulence modeling practices in enhanced or rib roughened ducts. But it should be taken into account that computer limitations require a delicate choice of simplifying assumptions. As a first step, examination of such simplifications via extensive numerical calculations and validation of the results is suggested. Therefore the main aim of the current effort is to use a satisfactory turbulence modeling approach in ribbed coolant channels and discuss the disadvantages and suggest future improvements for prediction of thermal-hydraulic performance of such ducts.

PROBLEM STATEMENT

The problem to be solved in this investigation is the fluid flow and heat transfer inside a rotating channel of square cross section with outward flow (Fig. 2). The axis of rotation is perpendicular to the main flow direction. The channel is of finite length and is attached to a base with radius equal to R_b (base radius).

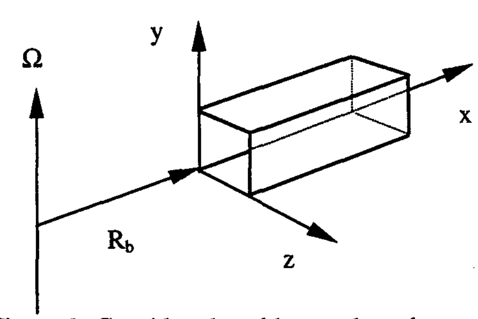

Figure 2. Considered problem and used coordinate system and axis of rotation.

MATHEMATICAL MODEL

The governing equations are the continuity, the time averaged Navier-Stokes and the energy equations for turbulent flow, i. e.

$$\frac{\partial}{\partial x_j}(\rho U_j)=0 \tag{1}$$

$$\frac{\partial}{\partial x_j}(\rho U_j U_i)=-\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial U_i}{\partial U_j}+\frac{\partial U_j}{\partial U_i}\right)\right]$$
$$+\frac{\partial}{\partial x_j}(-\rho\overline{u_i u_j})-2\rho\varepsilon_{ilj}\Omega_l U_j \tag{2}$$
$$-\rho\left[(\Omega_j x_j)\Omega_i-(\Omega_j\Omega_j)x_i\right]$$

$$\frac{\partial}{\partial x_j}(\rho U_j T)=\frac{\partial}{\partial x_j}\left[\alpha\frac{\partial T}{\partial x_j}-\rho\overline{u_j t}\right] \tag{3}$$

The additional source terms appearing in the momentum equations are centrifugal-buoyancy and coriolis forces. In this case incompressible flow is considered and the density is thus constant.

In practical gas turbine applications, only the first few blade rows are exposed to the very hot gas stream entering the turbine. In this work only these first blade rows are considered. The length of these blades is relatively short compared to the rotation radius and this fact justifies an assumption of a constant centrifugal force based on the mean rotation radius. The centrifugal force then can be added to the main flow direction pressure gradient and the flow and temperature fields then will behave periodically in this direction. The procedure for handling fully developed periodic flow and heat transfer was introduced already by Patankar *et al.* (1977) and has then been applied extensively. This procedure is followed in this study as well. One introduces,

$$P=-\beta x+P^* \tag{4}$$

where β resembles non-periodic pressure gradient and P^* is the periodic part of pressure in the main flow direction.

The dimensionless temperature θ is defined as

$$\theta=\frac{T-T_w}{T_b-T_w} \tag{5}$$

and by insertion this definition into the energy equation one obtains,

$$\frac{\partial}{\partial x_j}(\rho U_j\theta)=\frac{\partial}{\partial x_j}\left[\alpha\frac{\partial\theta}{\partial x_j}-\rho\overline{u_j\theta}\right]+\Psi \tag{6}$$

where Ψ is,

$$\Psi=\lambda\left[\alpha\frac{\partial\theta}{\partial x}+\frac{\partial}{\partial x}(\alpha\theta)-\rho u\theta\right]+\alpha\theta\left(\lambda^2+\frac{\partial\lambda}{\partial x}\right) \tag{7}$$

and λ is

$$\lambda=\frac{\partial(T_b-T_w)}{\partial x}/(T_b-T_w) \tag{8}$$

and the bulk temperature, T_b, is evaluated using a velocity weighted average.

Turbulence model

Several models have been suggested for calculation of turbulent flow and heat transfer, see e.g. Wilcox (1993). For engineering calculations, two equation models accompanied with the eddy viscosity concept have become popular since they provide reasonable overall results in many cases although not providing correct local distributions of e.g. turbulence generated secondary flows in ducts. In applications like the present one secondary flows due to the rotation occur. Recently, Abe *et al.* (1993) proposed a low-Re number version of the k-ε model. Their model has the following advantages: firstly it uses a velocity scale near the wall other than the friction velocity and this velocity scale solves the singular point problem close to the reattachment in the separated flow cases, and secondly, numerical experimentation showed that this model does not have the initial value dependency problem of some other models in periodic boundary conditions which were reported in the literature (Rokni and Sundén, 1997). This model, which is used in this investigation, can be summarized as,

$$\frac{\partial}{\partial x_j}(\rho U_j k)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{Pr_k}\right)\frac{\partial k}{\partial x_j}\right]-\rho\overline{u_i u_j}\frac{\partial U_i}{\partial x_j}-\rho\varepsilon \tag{9}$$

$$\frac{\partial}{\partial x_j}(\rho U_j\varepsilon)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{Pr_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]-C_{\varepsilon1}\frac{\varepsilon}{k}\rho\overline{u_i u_j}$$
$$-C_{\varepsilon2}f_\varepsilon\rho\frac{\varepsilon^2}{k} \tag{10}$$

$$\mu_t=C_\mu f_\mu\frac{k^2}{\varepsilon} \tag{11}$$

where the model damping functions and constants are

$$f_\mu=\left[1-\exp\left(-\frac{y^*}{14}\right)\right]^2\left[1+\frac{5}{Re_t^{3/4}}\exp\left\{-\left(\frac{Re_t}{200}\right)^2\right\}\right] \tag{12}$$

$$f_\varepsilon=\left[1-\exp\left(-\frac{y^*}{3.1}\right)\right]^2\left[1-0.3\exp\left\{-\left(\frac{Re_t}{6.5}\right)^2\right\}\right] \tag{13}$$

$$Re_t=\frac{\rho k^2}{\mu\varepsilon} \tag{14}$$

$$y^*=\frac{\rho u_\varepsilon n}{\mu} \tag{15}$$

$$C_\mu=0.09, Pr_k=1.4, Pr_\varepsilon=1.4, C_{\varepsilon1}=1.5, C_{\varepsilon2}=1.9$$

and $u_\varepsilon = (\mu\varepsilon/\rho)^{1/4}$ is the Kolmogorov velocity scale.

The standard k-ε turbulence model does not include rotational effects, because of the fact that rotational terms vanish in the k-equation. In order to improve the predictability of this model, it is an accepted practice to add rotation related source terms to the ε-equation. Howard *et al.* (1980) have evaluated three different rotational improvements and Younis (1993) claimed advantages of one of these methods, and used it successfully in his calculations. This method is a modification in the $C_{\varepsilon 2}$ constant of the ε-equation and is adopted here. The results also confirmed that it was a proper choice of rotational modification of the turbulence model in this stage of method development. The modification reads,

$$C_{\varepsilon 2} = 1.9\left(1 + 0.4\left(\frac{k}{\varepsilon}\right)^2 \Omega \frac{\partial U}{\partial z}\right) \tag{16}$$

The Reynolds stresses are determined using the simple eddy viscosity concept, and the low-Re form of the Generalized Gradient Diffusion Hypothesis (Rokni and Sundén, 1997) is utilized to calculate the turbulent heat fluxes, i.e.

$$-\rho\overline{u_i u_j} = \mu_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{2}{3}\rho k \delta_{ij} \tag{17}$$

$$\overline{u_j t} = -C_t f_\mu \frac{k}{\varepsilon}(T_b - T_w) \times \left[\overline{u_j u}\left(\lambda\theta + \frac{\partial\theta}{\partial x}\right) + \overline{u_j v}\frac{\partial\theta}{\partial y} + \overline{u_j w}\frac{\partial\theta}{\partial z}\right] \tag{18}$$

where $C_t = 0.35$.

BOUNDARY CONDITIONS

Periodic boundary conditions are applied at the inlet and outlet of every periodic module (which is equal to one pitch in the ribbed wall channel case). This condition is defined as

$$\Phi(x, y, z) = \Phi(x + L, y, z)$$

$$\Phi = U, V, W, P^*, k, \varepsilon$$

The boundary conditions for walls are imposed as:

$$U = V = W = k = 0$$

and

$$\varepsilon_w = 2\frac{\mu}{\rho}\left(\frac{\partial\sqrt{k}}{\partial n}\right)^2$$

where n represents the normal wall distance.

NUMERICAL SOLUTION PROCEDURE

A computer code based on the finite volume technique is modified and applied to solve the governing equations. The code uses collocated grid arrangement which stores the values of the dependent variables in the center of each control volume and employs the Rhie and Chow (1981) method to interpolate values of velocity on the control volume faces. The SIMPLEC algorithm is employed to handle the coupling between pressure and velocity. Convective fluxes are determined by the hybrid scheme in all equations.

A non-uniform grid has been chosen, and grid refinement has been applied in the near wall regions. Grid point numbers ranging between 3×42×42 and 3×82×82 were used for the straight duct case (three points in the main fully-developed direction), while the grid point number for the ribbed channel was 34×39×39. Further increase in the number of grid points did not change the results considerably. As a low Reynolds number formulation is applied, it is important that the y^+-value of the grid points close to the walls is of the order of unity. In this work the y^+-values were in the range of 0.3-0.8 for all considered cases.

The calculations were terminated when the absolute residual of all the variables became less than 10^{-6}. The under-relaxation factors for all calculations were set to values ranging between 0.5-0.6. In the ribbed channel case, in order to obtain converged solutions, under-relaxation of the source terms were applied in the early stages of the calculation procedure.

The non-periodic pressure gradient β is specified to create a flow field. Through the rate of mass flow, the Reynolds number is coupled to β. Thus various values of β correspond to various Reynolds numbers.

RESULTS

1. Straight duct with and without rotation

The straight duct with a square cross section has been chosen as the first test case to validate the method without rotational effects. The computations were made for Reynolds numbers ranging from 3000 to 30000. Numerical results for friction factor and the average Nusselt numbers were calculated, and these results were compared with corresponding experimental correlations, which are the McAdams (1954) correlation for the friction factor and the Dittus-Boelter correlation for the average Nusselt number,

$$f = 0.046 Re^{-0.2} \tag{19}$$

$$Nu_{DB} = 0.023 Re^{0.8} Pr^{0.4} \tag{20}$$

The results for the stationary case (straight duct without rotation) were in excellent agreement with these experimental correlations, about 3-7 percent error for the friction factor and 5-8 percent deviation for the Nusselt number. Based on this decent agreement, it was justified that the present method would give acceptable results for the main thermal-hydraulic characteristics of stationary straight ducts with square cross sections.

In order to investigate the rotational effects on the heat transfer rate and also validate the present method, calculations have been carried out for a rotating square-sectioned duct at some of the Re and Ro numbers as in the experiments by Hwang and Kuo (1993). Both the highest and lowest Ro numbers are included. The results for these cases are presented in Table 1. In this table, PS and SS represent pressure (trailing) and suction (leading) sides of the duct, respectively.

As is found from the experimental and calculated results there is an increase in the heat transfer rate on the pressure side. This fact can be explained by the existence of secondary flows (see Fig. 3), which are directed towards the pressure side (or trailing side) of the channel. Indeed, this flow brings the cold air from the center of the channel in contact with that wall, but on the other hand, the air is heated up in the vicinity of the same wall and relatively hot air returns towards the suction side. This increase has been successfully predicted by the present method with reasonably small errors as is shown in table 1.

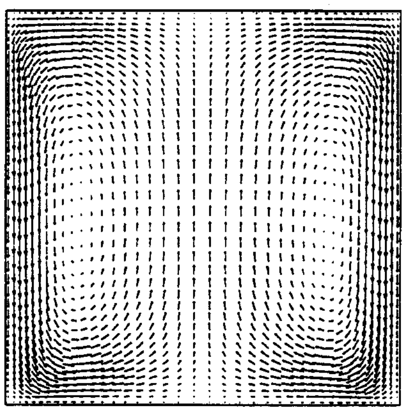

Figure 3. Secondary velocities in the plane normal to the main flow direction due to the rotation induced body forces, Re=4000.

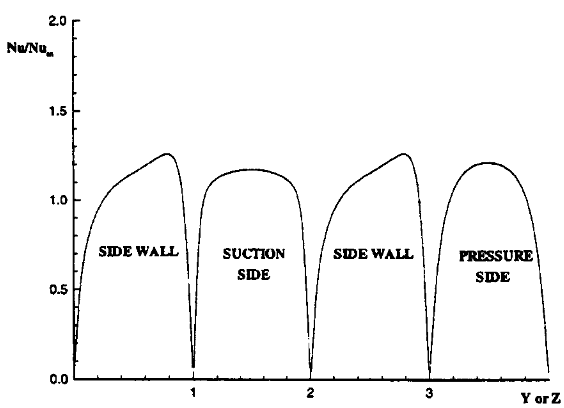

Figure 4. Normalized Nusselt number along the walls of the rotating duct, Re=4000, Ro=0.0141.

Figure 4 shows distributions of the local Nu number on the four walls in Fig. 3. It is obvious that the distributions on the side-walls are skewed due to the varying boundary layer thickness along these walls. On the suction side the distribution is rather flat while on the pressure side the highest values appear at the impingement zone. The results in Fig. 4 are consistent with the flow structure in Fig. 3.

The calculations for the rotating duct case are limited to the Ro range of the experimental results in order to validate the calculations. The highest Ro number in the experimental setup was at Re equal to 4000 as reported in table 1.

Table 1. Experimental and calculated Nusselt numbers for a rotating straight smooth duct

Re	Ro	Nu/Nu_{DB} (exp)	Nu/Nu_{DB} (cal)	Error (%)
4000	0.0141	PS: 1.10	1.17	+6
		SS: 0.88	0.84	-4
4000	0.0787	PS: 1.73	1.83	+5
		SS: 0.85	0.80	-4
8200	0.0069	PS: 1.10	1.07	-3
		SS: 0.92	0.97	+6
8200	0.0198	PS: 1.20	1.11	-7.5
		SS: 0.88	0.93	+6

The overall effect of the rotation on the thermal-hydraulic features of the smooth ducts can described as a proportional increase in f and Nu with an increase in the rotation (Rossby) number on the pressure side and a small increase in the mean values. This fact is mainly due to the relative magnitude of the secondary flows, and this relative magnitude is depending on the Ro number.

2. Square-sectioned duct with embedded ribs

The case which is studied here is a duct with a square cross sectional area. There are two ribs of a square shape and these are placed on two opposite sides of the channel. The distance between two subsequent ribs, or the pitch, is chosen equal to 10, and the rib height to diameter is set equal to 0.04 (Fig. 5). There are experimental correlations for the friction factor and the Nusselt number for these channels in non-rotating mode. These are given by Han (1988). Due to the complexity, these correlations are not given here. The computational results for the friction factor and the average Nusselt numbers are compared with results of Han's (1988) correlations (Table 2). The satisfactory agreement between the theoretical results and the experimental ones shows that the present method can handle such problems with reasonable errors.

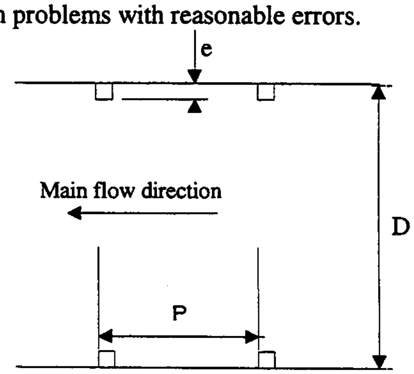

Figure 5. Cross section of the enhanced duct with the ribs and their spacing.

Table 2. The friction factors and the Nusselt numbers for a ribbed duct with P/D=10 and e/D=0.04, non-rotating case. (ff=friction factor, n=Nu/Nu_{DB})

Re	Han's corr. (1988)	Present Cal.	Difference (%)
10765	ff: 0.04065	0.03830	-5.8
	n: 2.098	2.169	+3.4
14410	ff: 0.04065	0.04398	+8.0
	n: 2.090	2.271	+8.0
17819	ff: 0.04065	0.04329	+6.5
	n: 2.083	2.270	+9.0

The results obtained for the ribbed duct showed that there is almost a 5-6 fold increase in the friction factors but only a two times increase in the Nu numbers compared to the values for a smooth duct with the same cross section. So, there is an enhancement of the heat transfer rate but it is accompanied by a large pressure loss penalty.

Figure 6 shows a velocity vector plot of the flow field in the mid plane. The recirculating zone downstream the rib and the reattachment region are clearly seen. Also there is a recirculating zone upstream a rib. The velocity vectors in the vicinity of the ribs are deflected towards the wall downstream a rib and deflected away from the wall upstream a rib.

In Fig. 7 isotherms are depicted. On the plane wall the temperature gradients are high in particular between the recirculating zones. The heat transfer coefficient becomes high locally. Far away from the ribs their influence on the temperature field is weak.

CONCLUSIONS

The calculations have shown that the presented method can successfully predict the friction factor and the heat transfer rate in smooth and rib enhanced ducts in the range of the Re numbers of interest in this study. Therefore, the existing combination of a low-Re form of the k-ε turbulence model and GGDH method for the turbulence heat flux determination, has the ability to predict the overall thermal-hydraulic characteristics of the considered cases. Studies of the local flow and heat transfer improved the understanding of the fundamental mechanisms of momentum and heat transfer.

There are effects of the secondary flows of the second type which are not captured by the present method. These effects become important with higher ribs. The most straight forward way to abandon this problem is to use some non-linear approach to determine the Reynolds stresses, instead of the simple eddy diffusivity concept, which is used here.

The other disadvantage of the present approach is the ignorance of the secondary effects of the rotation on the turbulence field. Once again, the authors believe that the best way to include these effects in the model is to use a non-linear approach in determination of the Reynolds stresses.

It was shown that in the studied range of the rotation numbers, there is a pronounced increase in the Nu number on the pressure side, but a slight increase in the average heat transfer rate. The numerical experiments with the ribbed duct showed considerable enhancement of the average heat transfer rate, but this enhancement is accompanied with a very large increase in the friction factor.

REFERENCES

1. Abe, K., Kondoh, T. and Nagano, Y., 1994, "A New Turbulence Model for Predicting Fluid Flow and Heat Transfer in Separating and Reattaching Flows-I. Flow Field Calculations", *Int. J. Heat Mass Transfer*, Vol. 37, pp. 139-151.
2. Bo, T., Iacovides, H. and Launder, B. E., 1995, "Developing Buoyancy-Modified Turbulent Flow in Ducts Rotating in Orthogonal Mode", ASME *J. Turbomachinery*, Vol. 117, pp. 474-484.
3. Dutta, S rews M. J. and Han, J. C., 1994, "Heat Transfer and Flow Predictions in a Rotating Square Duct with Coriolis Modified Turbulence Models", *Proc. Tenth Int. Heat Transfer Conf.*, Vol. 4, pp. 219-224.
4. Han, J.C., 1988, "Heat Transfer and Friction Characteristics in Rectangular Channels with Rib Turbulators", ASME *J. Heat Transfer*, Vol. 110, pp. 321-328.
5. Howard, J.H.G., Patankar, S.V. and Bordynuik, R.M., 1980, "Flow Prediction in Rotating Ducts Using Coriolis-Modified Turbulence Models", ASME *J. Fluids Eng.*, Vol. 102, pp. 456-461.
6. Hwang, G. J. and Kuo, C. R., 1993, "Experimental Study of Convective Heat Transfer in a Rotating Square Duct with Radial Outward Flow", ASME HTD-Vol. 242, pp. 53-60.
7. Iacovides, H. and Launder, B. E., 1987, "Turbulent Momentum and Heat Transport in Square-Sectioned Ducts Rotating in Orthogonal Mode", *Numer. Heat Transfer*, Vol. 12, pp. 475-491.
8. Johnson, B. V., Wagner, J. H., Steuber, G. D. and Yeh, F. C., 1994, "Heat Transfer in Rotating Serpentine Passages with Selected Model Orientations for Smooth or Skewed Trip Walls", ASME *J. Turbomachinery*, Vol. 116, pp. 738-744.
9. Liou, T. and Hwang, J., 1993, "Effect of Ridge Shapes on Turbulent Heat Transfer and Friction in a Rectangular Channel", *Int. J. Heat Mass Transfer*, Vol. 36, pp. 931-940.
10. McAdams, W.H., 1954, *Heat Transmission*, 3rd ed., McGrawHill.
11. Parsons, J. A., Han, J. C. and Zhang, Y., 1995, "Effect of Model Orientation and Wall Heating Condition on Local Heat Transfer in a Rotating Two-Pass Square Channel with Rib Turbulators", *Int. J. Heat Mass Transfer*, Vol. 38, pp. 1151-1159.
12. Patankar, S. V., Liu, C. H. And Sparrow, E. M., 1977, "Fully Developed Flow and Heat Transfer in ducts Having Streamwise-periodic Variations of Cross-sectional Area", ASME *J. Heat Transfer*, Vol. 99, pp. 180-186.
13. Prakash, C. and Zerkle, R., 1991, "Prediction of Turbulent Flow and Heat Transfer in a Radially Rotating Square Duct", ASME HTD-Vol. 188, pp. 1-13.
14. Prakash, C. and Zerkle, R., 1995, "Prediction of Turbulent Flow and Heat Transfer in a Ribbed Rectangular Duct With and Without Rotation", ASME *J. Turbomachinery*, Vol. 117, pp. 255-264.

15. Rhie, C. M. and Chow, W. L., 1983, "Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation", *AIAA J.*, Vol. 21, pp. 1525-1532.
16. Rokni, M. and Sundén, B., 1997, "Improved Modeling of Turbulent Forced Convective Heat Transfer in Straight Ducts", ASME HTD-Vol. 346, pp. 141-149.
17. Wilcox, D. C., 1993, "Turbulence Modeling for CFD", DCW Industeries Inc., USA.
18. Younis, B. A., 1993, "Prediction of Turbulent Flows in Rotating Rectangular Ducts", ASME *J. Fluids Engineering*, Vol. 115, pp. 646-652.

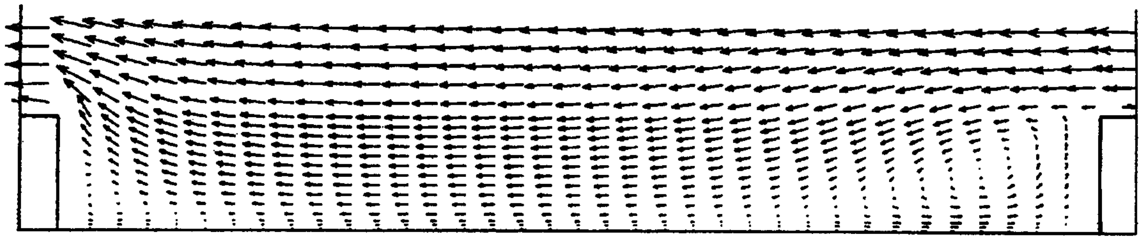

Figure 6. Velocity vector plot of the flow in the enhanced duct, Re=14410.

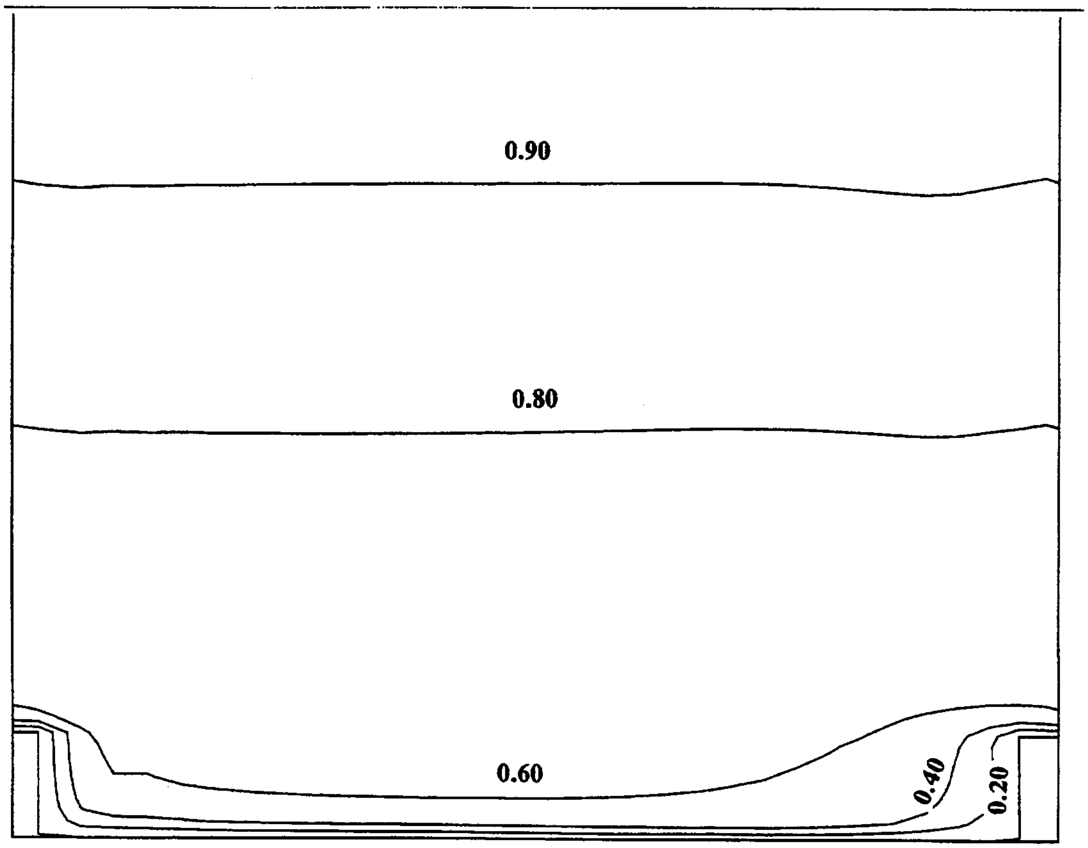

Figure 7. Isotherm lines for non-dimensionalized temperature for enhanced duct case, Re=14410.