

NUMERICAL ANALYSES OF TURBULENT FLOWS BY MEANS OF ALGEBRAIC REYNOLDS STRESS AND TURBULENT HEAT FLUX MODELS

Hitoshi Sugiyama

Department of Advanced Interdisciplinary Sciences, Utsunomiya University
(7-1-2 Yoto, Utsunomiya, 321-8585, Japan)

Fax: +81-28-689-6031 Email: sugiyama@cc.utsunomiya-u.ac.jp

ABSTRACT

In this paper, simulated results for three types of turbulent flows are introduced, which are predicted by using anisotropic turbulence models, in order to examine the capabilities for predicting turbulent flow with and without heat transfer. The first example is to predict a turbulent flow with heat transfer in a square duct with one roughened wall. Bottom wall of square duct is roughened by small ribs with square cross section which are arranged periodically over bottom wall. The second example is to simulate a turbulent flow in a rotating U-bend of strong curvature with roughened wall. This flow is characterized by interaction between Coriolis's and centrifugal forces. The third example is to calculate a turbulent flow in a rectangular duct with a sharp 180-degree turn. This type of flow is characterized by a large-scale separated flow, and it is difficult to predict correctly the reattachment point of a separated flow.

NOMENCLATURE

a : thermal diffusivity
 D : side length of square cross-section
 D_h : hydraulic diameter
 h : rib height
 k : turbulent energy
 P : mean pressure
 P_k : production term of turbulent energy
 p : fluctuating pressure
 R_c : central radius of the U-bend
 Re : Reynolds number DU_b/ν
 Ro : rotation number $D\Omega/U_b$
 T_c : axial centerline temperature
 T_w : wall temperature
 T' : temperature fluctuation
 U_θ, U_r : mean velocity in θ and r directions
 u_θ, u_r : fluctuating velocity in θ and r directions
 $\overline{u_i u_j}$: Reynolds stresses

$\overline{u_i T'}$: i 'th turbulent heat flux correlation; $i=1,2,3$
 U_b : bulk velocity
 U_s : maximum velocity in square duct
 U_τ : friction velocity
 U_i : i 'th mean velocity component; $i=1,2,3$
 X_i : i 'th Cartesian coordinate; $i=1,2,3$
 X_1^* : normalized axis X_1/D_h
 X_2^{**} : middle location of 180-deg. curved duct
 X_2^* : normalized axis X_2/D
 X_3^* : normalized axis $X_3/(H/2)$
 r : radius
 t : time
 $(\)$: ensemble averaged value
Greek letters
 δ_{ij} : Kronecker delta
 ε : turbulent dissipation
 ε_{ijk} : Levi-Civita's alternating tensor
 ν : kinematics viscosity
 θ : bend angle
 Ω_i : angular rotation speed of duct of X_i axis

INTRODUCTION

The purpose of this paper is to introduce current capabilities for predicting turbulent flow with and without heat transfer by using algebraic Reynolds stress and turbulent heat flux models. In calculation of turbulent flow, it is important to predict anisotropic turbulence correctly in order to reproduce the characteristic features of turbulent flow such as the secondary flow of the second kind which is caused by anisotropic turbulence. Algebraic Reynolds stress and turbulent heat flux models have potentiality to be able to reproduce anisotropic turbulent flow saving computing time. In this paper, simulated results for three types of turbulent flows are introduced.

In some square duct heat exchanger applications, one or more walls of the duct are roughened in order to enhance heat transfer. Research on turbulent flow

in square ducts with one or more rib-roughened walls can be classified into two types of basic studies: those concerned with local flow and heat transfer in the vicinity of a rib-roughened wall and those concerned with global flow and heat transfer behavior across the entire duct cross section. The present study lies within the latter category, and focuses on the nature of temperature and turbulent heat flux distributions in the duct cross plane, as influenced by one roughened wall for fully-developed flow conditions. In reference to previous research in this area, Humphrey and Whitelaw [1] and Fujita et al. [2] have measured mean flow and turbulence properties in a square duct having one or more rib-roughened walls. The results of Fujita et al. [2] show that the overall flow pattern in a square duct with one rib-roughened wall does not depend strongly on whether measurements are made in the duct cross section directly over a rib or midway between adjacent ribs, provided the rib height compared to the duct height is small. This conclusion provides justification for treating a rib-roughened wall as one of uniform roughness when the flow is fully developed, as is done in the present study.

Heat transfer measurements and predictions in square ducts with one or more roughened walls are fairly scarce. However, Hirota et al. [3] present cross planar distributions of the three turbulent heat flux components in a square duct with one rib-roughened wall. Measurements such as these provide valuable data from the standpoint of assessing proposed models of the turbulent heat flux transport equation. Therefore, turbulent flow with heat transfer in a square duct with one roughened wall is simulated and compared with the experimental data measured by Hirota et al. [3].

Advanced gas turbine blades have been designed to operate with high inlet temperature that is far about the allowable metal temperature. Therefore, it is indispensable to cool the turbine blades efficiently to maintain the performance. As for the internal cooling of the blades, cool air extracted from compressor stages of the engine is circulated through internal cooling passages inside the turbine rotor blades in order to maintain the operating temperature of the blades at safe levels. From this point of view, a lot of research works have been reported [4],[5]. Iacovides et al. [6] have measured turbulent flows in a square sectioned 180 degree sharp turn of coolant passage with and without rotation by laser-Doppler anemometry in detail including Reynolds stress distributions. At the same time, Iacovides [7] carried out the calculation of turbulent flow in stationary and

rotating U-bend of square cross section with rib-roughened walls along the straight sections.

However, the calculated results of turbulent flow in rotating sharp turn channel with roughened walls are few and far between. Besides, there is almost nothing to compare in detail with the distributions of Reynolds stresses in rotating sharp turn channel with roughened walls. Considering these situation, this paper shows predicted results of turbulent flow in rotating sharp turn channel with roughened walls by using algebraic Reynolds stress model.

A separated turbulent flow with large scale is one of the most complicated and difficult flows to predict correctly. A great number of numerical and experimental measurement results [8],[9] have been reported for a turbulent flow in a duct with separation but few numerical or experimental results have been reported related to turbulent separation including detailed measurement of Reynolds stresses. For example, Iacovides and Raisee [10] predicted the turbulent flow and convective heat transfer in passages affected by strong curvature and rib-roughness using effective-viscosity and Reynolds stress models. However, they did not compare the Reynolds stresses distributions with the experimental data because they examined the prediction of convective heat transfer rather than that of the velocity field. They reported that a low-Reynolds differential stress closer yields thermal predictions that are superior to those of the low-Reynolds effective-viscosity model. Nakayama et al. [11] reported in detail the measurement results for a turbulent flow in a rectangular duct with a sharp 180-degree turn, including the distributions of Reynolds stresses. They measured the velocity profiles of the mean velocity in the separated region and the distributions of Reynolds stress using laser-Doppler anemometry. As a third example, this paper represents the simulated results of separated turbulent flow in a rectangular channel with a sharp turn comparing with the experimental data reported by Nakayama et al. [11] in order to estimate the validity of an algebraic Reynolds stress model.

MATHEMATICAL MODEL

Flow field model: The anisotropy nature of the turbulence is expressed by the Reynolds stress equations. In order to deal with anisotropic turbulence precisely, we have adopted the transport equation of Reynolds stress in numerical analysis. The exact formula of the transport equation of Reynolds stress is shown as follows.

Table 1

Modeling of the Pressure-strain Correlation Term

$\pi_{ij,1} + \pi_{ji,1}$	$-C_1 \frac{\varepsilon}{k} \left(\overline{u_i u_j} - \frac{2}{3} k \delta_{ij} \right)$
$\pi_{ij,2} + \pi_{ji,2}$	$-\frac{C_2 + 8}{11} \left(P_{ij} - \frac{2}{3} P_k \delta_{ij} \right) + \varsigma_k \left(\frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial X_i} \right)$ $-\frac{8C_2 - 2}{11} \left(D_{ij} - \frac{2}{3} P_k \delta_{ij} \right)$
$[\pi_{ij} + \pi_{ji}]_w$	$C_1 = C_1^* + C_1' f \left(\frac{L}{X_w} \right) \quad C_2 = C_2^* + C_2' f \left(\frac{L}{X_w} \right)$ $\varsigma_2 = \varsigma_2^* + \varsigma_2' f \left(\frac{L}{X_w} \right)$
$P_{ij} = -\overline{u_i u_j} \frac{\partial U_j}{\partial X_k} - \overline{u_j u_k} \frac{\partial U_i}{\partial X_k} \quad D_{ij} = -\overline{u_i u_k} \frac{\partial U_k}{\partial X_j} - \overline{u_j u_k} \frac{\partial U_i}{\partial X_k}$ $P_k = -\overline{u_k u_l} \frac{\partial U_l}{\partial X_i} \quad f \left(\frac{L}{X_w} \right) = \frac{C_\mu^{3/4} k^{3/2}}{\kappa \varepsilon X_w}$	

Table 2

Constants of the Pressure-strain Correlation Term

C_1^*	C_2^*	ς^*	C_1'	C_2'	ς'	C_μ	κ
1.4	0.44	-0.16	-0.35	0.12	-0.1	0.09	0.42

$$\frac{Du_i u_j}{Dt} = - \left(\overline{u_i u_j} \frac{\partial U_j}{\partial X_k} + \overline{u_j u_k} \frac{\partial U_i}{\partial X_k} \right) + \frac{p}{\rho} \left(\frac{\partial u_i}{\partial X_j} + \frac{\partial u_j}{\partial X_i} \right) - \frac{\partial}{\partial X_k} \left\{ \overline{u_i u_j u_k} - \nu \frac{\partial \overline{u_i u_j}}{\partial X_k} + \frac{p}{\rho} \left(\delta_{jk} u_i + \delta_{ik} u_j \right) \right\} - 2\nu \frac{\partial u_i}{\partial X_k} \frac{\partial u_j}{\partial X_k} \quad (1)$$

The convection and diffusion terms in the above equation were modeled using Rodi's approximation, [12] i.e.,

$$\frac{Du_i u_j}{Dt} - \text{Diff}_{ij} = \frac{\overline{u_i u_j}}{2k} (P_k - \varepsilon) \quad (2)$$

where Diff_{ij} corresponds to the third term on the right hand side of Equation (1).

Inasmuch as wall functions were used in the present study for the computations, the dissipation rate everywhere in the computed flow was assumed to be locally isotropic; i.e.

$$\varepsilon_{ij} = 2\nu \frac{\partial u_i}{\partial X_k} \frac{\partial u_j}{\partial X_k} = \frac{2}{3} \delta_{ij} \varepsilon \quad (3)$$

A particularly problematic task here is the modeling of the pressure-strain correlation term, which is also defined as the redistribution term and is shown as the second term on the right-hand side of Equation (1). The pressure-strain term is composed of three parts, which are the interaction of the fluctuating velocities ($\pi_{ij,1} + \pi_{ji,1}$) and the interactions of the mean strain with the fluctuating velocities ($\pi_{ij,2} + \pi_{ji,2}$) and the wall proximity effects ($\pi_{ij,w} + \pi_{ji,w}$). In the present calculation, we have adopted Rotta's linear return to isotropy mode for the ($\pi_{ij,1} + \pi_{ji,1}$) term, as shown in Table 1. The interactions of the mean strain with the fluctuating velocities ($\pi_{ij,2} + \pi_{ji,2}$) is modeled basically based on the forth order tensor proposed by Launder et al. [13], However, the model modified by

Table 3

Modeling of the Pressure-temperature Gradient Term

$\pi_{iT,1}$	$-C_{1T} \frac{\varepsilon}{k} \overline{u_i T'} - C_{1T}' \frac{\varepsilon}{k} \left(\frac{\overline{u_i u_j}}{k} - \frac{2}{3} \delta_{ij} \right) \overline{u_j T'}$
$\pi_{iT,2}$	$C_{2T} \overline{u_m T'} \frac{\partial U_i}{\partial X_m} - C_{2T}' \overline{u_m T'} \frac{\partial U_m}{\partial X_i}$
$\pi_{iT,w}$	$C_{1T} = C_{1T}^* \left\{ 1 + C_{1T,w} \cdot f \left(\frac{L}{X_w} \right) \right\}$ $C_{1T}' = C_{1T}'^* \left\{ 1 + C_{1T,w} \cdot f \left(\frac{L}{X_w} \right) \right\}$ $C_{2T} = C_{2T}^* \left\{ 1 + C_{2T,w} \cdot f \left(\frac{L}{X_w} \right) \right\}$ $C_{2T}' = C_{2T}'^* \left\{ 1 + C_{2T,w} \cdot f \left(\frac{L}{X_w} \right) \right\}$

Table 4

Constants of the Pressure-temperature Gradient Term

C_{1T}^*	$C_{1T}'^*$	C_{2T}^*	$C_{2T}'^*$	$C_{1T,w}$	$C_{2T,w}$
3.9	-2.5	0.8	0.2	0.25	-0.46

Sugiyama et al. [14] is adopted in this calculations. In terms of the modeling of ($\pi_{ij,2} + \pi_{ji,2}$), the modeling process is described in detail in the report by Sugiyama et al. [14]. The wall effect term ($\pi_{ij,w} + \pi_{ji,w}$) on turbulent stresses is modeled as shown in Table 1 by changing the model constants. In Table 1, $f(L/X_w)$ is a function related to the dimensionless distance from the wall, and C_μ and κ represent the empirical constant and the von Karman constant, respectively. The function $f(L/X_w)$ is of unit value near the wall and approaches zero with increasing distance from the wall. The symbol X_w is the normal distance from the wall, and L defines the length scale of turbulence. When $f(L/X_w)$ is zero, the model yields the correct Reynolds stress components for the nearly homogeneous shear flow of Champagne et al. [15]

The coefficient values of the pressure-strain correlation term are specified in Table 2.

Temperature field model: The mean temperature and turbulent heat flux fields were calculated from the boundary layer form of the thermal energy equation in conjunction with algebraic models for the turbulent heat flux correlation which appears in thermal energy equation. The development of the model starts with the exact form of the turbulent heat flux transport equation, namely:

$$\frac{Du_i T'}{Dt} = - \left(\overline{u_i u_j} \frac{\partial T'}{\partial X_j} + \overline{u_j T'} \frac{\partial U_i}{\partial X_j} \right) + \frac{p}{\rho} \left(\frac{\partial T'}{\partial X_i} \right) - \frac{\partial}{\partial X_j} \left(\overline{u_i T' u_j} + \frac{p}{\rho} T' \delta_{ij} \right) - (\nu + a) \frac{\partial T'}{\partial X_j} \frac{\partial u_i}{\partial X_j} \quad (4)$$

where the terms on the right hand side represent, respectively, the production of turbulent heat flux

from the mean flow, the pressure-temperature gradient effect which leads to inhomogeneity among the individual heat flux components, the diffusion of turbulent heat flux, and its dissipation. The convection and diffusion terms were modeled in a manner similar to Rodi's approximation for similar terms that appear in the Reynolds stress transport equation, namely:

$$\frac{Du_i T'}{Dt} - \text{Diff}_{iT} = \frac{u_i T'}{2k} (P_k - \varepsilon) \quad (5)$$

where Diff_{iT} corresponds to the third term on the right hand side of Equation (4). The fourth (dissipation) term on the right hand side of Equation (4) was neglected on the basis of assumed high Reynolds number flow.

It is necessary to express the pressure-temperature gradient term as the mathematical model as well as pressure strain term. The pressure-temperature gradient term is modeled as shown in Table 3. As for modeling slow return effects, Lumley [16] modified the model presented by Monin [17] by introducing an anisotropic tensor into the model coefficients which yielded the expression for $\pi_{iT,1}$. In order to model rapid return effects, the model proposed by Lumley [16] and independently by Launder [18] was adopted in this study. Inasmuch as wall effects on the model coefficients are important, coefficient values are evaluated from the expressions which take into account wall proximity effects as shown in Table 3. The model constants are listed in Table 4.

Boundary-fitted coordinate system: In the calculation, the boundary condition must be set precisely next to the complicated shape. In this calculation, a boundary-fitted coordinate system, which is a type of coordinate transformation method, is introduced. The coordinate in the physical plane can be transformed to that in the calculation plane using the boundary-fitted coordinate system. Numerical calculation is performed in the calculation plane because setting the boundary conditions next to the complicated shape is easy. In addition, note that the governing equations are transformed into complicated equations from simple equations that are expressed in the physical plane, although it is easy to set boundary conditions next to the complicated shape.

The transformation from the physical plane to the calculation plane is performed by using the following mathematical theorem:

$$\frac{\partial}{\partial X_i} = \frac{\partial \xi}{\partial X_i} \frac{\partial}{\partial \xi} + \frac{\partial \eta}{\partial X_i} \frac{\partial}{\partial \eta} + \frac{\partial \zeta}{\partial X_i} \frac{\partial}{\partial \zeta} \quad (6)$$

where ξ , ζ , and η represent the coordinates of the calculation plane and correspond to the main flow

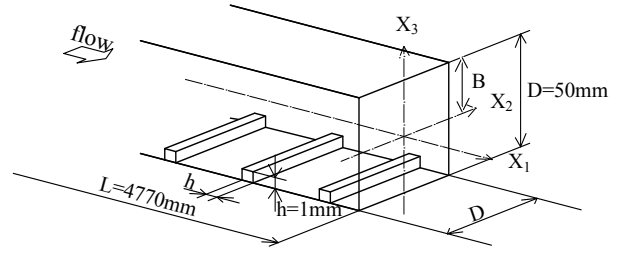


Figure 1

Square duct with one roughened wall

and the cross sectional directions along the computational grid.

RESULTS AND DISCUSSION

Turbulent heat transfer in a square duct with one roughened wall: The duct configuration and coordinate axes are shown in Figure 1. Experimental configuration consisted of a square duct, 50mm×50mm in cross section, with an overall length of 4770 mm. The unheated portion of the duct consisted of a 3020 mm long section, which yielded fully developed turbulent flow at the entrance to the heated section at $X_1/D=60.4$. This section, which consisted of 10mm thick aluminum walls and extended from $X_1/D=60.4$ to $X_1/D=94.5$, was surrounded by a constant temperature steam bath at 373 K. The rib-roughened wall shown in Figure 1 was generated by machining square ribs, 1mm×1mm in cross section, directly on one wall with a periodic spacing of 10mm between adjacent ribs over the length of the duct. Measurements were made in the duct cross section near the end of the duct midway between adjacent ribs at $X_1/D=93.4$ where both the mean temperature and mean velocity fields were fully developed. The operating Reynolds number was 6.5×10^4 based on properties evaluated at the entrance of the heated section.

Fully developed flow and heat transfer in a smooth walled duct with one roughened wall was assumed. The operating Reynolds number was specified as 6.5×10^4 and all duct walls were assumed to be at the same constant temperature. These conditions correspond to the experimental operating conditions. Computations were performed relative to a 22×44 uniform grid in the duct half cross section, with the first grid line near each wall located in the log-law layer, as confirmed by comparison with the data of Hirota et al.[3]. In the present study, the velocity field was calculated first, and then the temperature field, using velocity field results as input for the temperature field calculations.

The conventional wall functions for k and ε were specified along the first grid line near each wall. On the line adjacent to the roughened wall, the log-law velocity distribution measured by Fujita et al. [2] was applied, namely:

$$\frac{U_1}{U_\tau} = \frac{1}{0.42} \ln \left(\frac{U_\tau y}{\nu} \right) - 8.4 \quad (7)$$

which also was applied at grid points on the corner bisector near each smooth/rough wall intersection. Along the first grid line adjacent to each smooth wall, the conventional form of the log-law was specified; i.e.,

$$\frac{U_1}{U_\tau} = \frac{1}{0.42} \ln \left(\frac{U_\tau y}{\nu} \right) - 5.5 \quad (8)$$

Primary and secondary flow velocity distributions are shown in Figure 2, in order to demonstrate the effect of one roughened wall on the cross-sectioned mean flow pattern in a square duct for fully developed flow conditions. Contour values of the primary and secondary velocity components are normalized by the maximum primary velocity U_s . The dashed line above the lower wall in both figures represents the height of the ribs in the experiments. Predicted primary velocity contours are in relatively good agreement with their experimental counterparts over much of the duct cross section, except in the central region where predicted contour levels slightly exceed experimental values. The secondary flow of the second kind caused by anisotropic turbulence shows that the experimentally observed, large counter-clockwise cell is simulated well by the predictions. The computations also show, however, that a much smaller counter-rotating cell is predicted near the corner where adjacent smooth walls intersect. This cell is not readily evident in the experimental pattern. It should be noted, however, that Naimi and Gessner [19] have shown that this small cell is also predicted when alternate turbulence models are employed.

Figure 3 shows the contour maps of normal stress for streamwise flow direction and shear stress between axial and vertical fluctuating velocities. The calculated results of streamwise normal stress is good agreement with the experiment. The maximum value of normal stress is produced actively near roughened wall. It is pointed out as characteristic feature from the experiment that different sign regions appear in the distribution of shear stress. The compared results with the experiment of shear stress suggest that the prediction can also reproduce such characteristic features of the experimental results.

Mean temperature and turbulent heat flux $\overline{u_1 T'}$ are compared with the experimental contours in Figure 4.

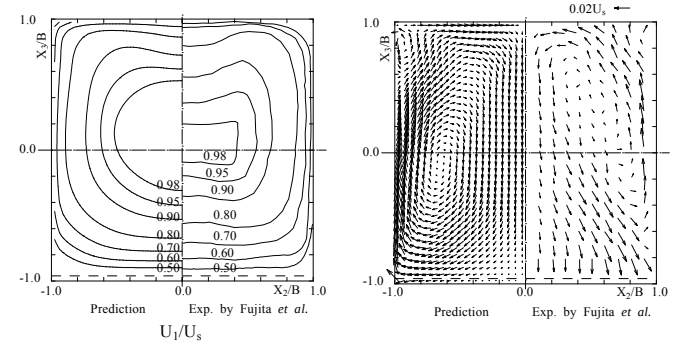


Figure 2
Comparison of primary and secondary flow

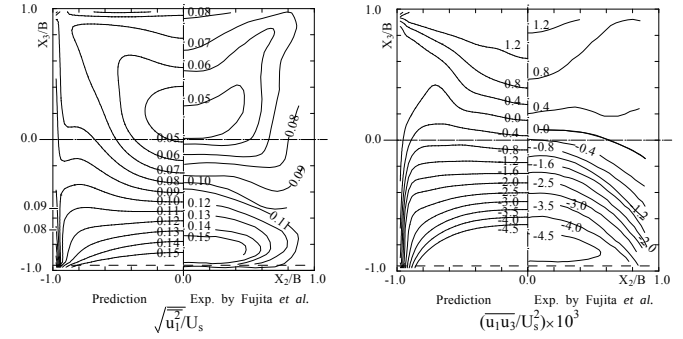


Figure 3
Comparison of normal and shear stresses

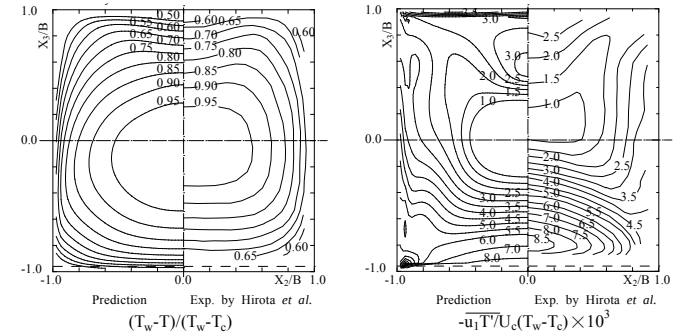


Figure 4
Comparison of mean temperature and turbulent heat flux

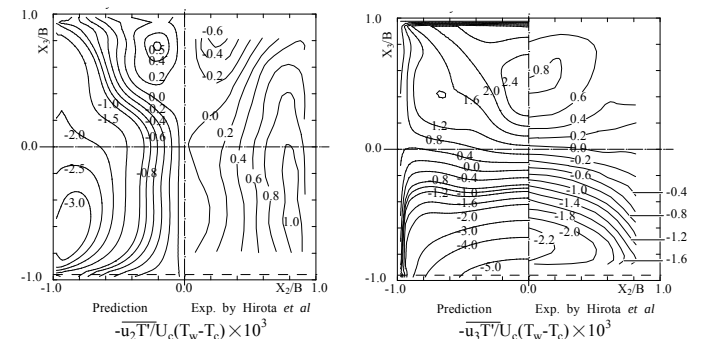


Figure 5
Comparison of turbulent heat flux $\overline{u_2 T'}$ and $\overline{u_3 T'}$

As for turbulent heat flux, the presented model tends to overestimate contour levels in the upper half of the duct and underestimate contour levels in the lower half of the duct. In the immediate vicinity of lower roughened wall, predicted contour levels are in relatively good agreement with experimental levels.

Predicted contours of the turbulent heat flux components $\overline{u_2 T'}$ and $\overline{u_3 T'}$ are compared with the experimental results in Figure 5. As shown in Figure 5, distribution of $\overline{u_2 T'}$ is characterized by forming the positive and negative sign regions. The calculated result shows that there is fair agreement between predicted and experimental contour levels. However, the presented model is unable to simulate the experimentally observed peaking characteristic of contours measured near the side wall. The experimental results near the smooth wall opposite the roughened wall appear to indicate the presence of a closed contour pattern.

The experimental contours show that $\overline{u_3 T'}$ changes sign in the vertical direction, with the zero line nominally aligned with horizontal bisector of the duct cross section. This behavior is predicted relatively well. Adding to this, the distribution of turbulent heat flux $\overline{u_3 T'}$ is similar to that of shear stress $\overline{u_1 u_3}$ as shown in Figure 3. This result implies that similarity between heat transfer and momentum transfer comes into existence in this turbulent flow. It has been pointed out as a discrepancy between calculated result and experimental result that the presented model tends to overpredict $\overline{u_3 T'}$ values in the upper half of the duct, and predicted negative contour levels near the roughened wall are larger in magnitude than experimentally observed values. It is true that predicted results depend on the turbulent heat flux model. With reference to this model dependency, Sugiyama et al.[20] presented the difference between anisotropic turbulence models by calculating the square duct with roughened wall.

Turbulent flow in rotating U-bend with roughened wall: In the case of the calculation for turbulent flow with rotation, it is necessary to take Coriolis's and centrifugal forces induced by rotation into account. The transport equation of momentum is expressed in the following form through ensemble averaging in the case of rotating coordinate:

$$\frac{D(\overline{u_i u_j})}{Dt} = -\frac{1}{\rho} \frac{\partial P}{\partial X_i} + \frac{\partial}{\partial X_j} \left[\nu \left(\frac{\partial \overline{u_i}}{\partial X_j} + \frac{\partial \overline{u_j}}{\partial X_i} \right) - \overline{u_i u_j} \right] - 2\epsilon_{ijk} \Omega_p U_j - (\Omega_j X_j \Omega_i - \Omega_j X_i \Omega_j) \quad (9)$$

The third and forth terms of right hand side of Equation (9) represents Coriolis's and centrifugal forces induced by rotation, respectively. The

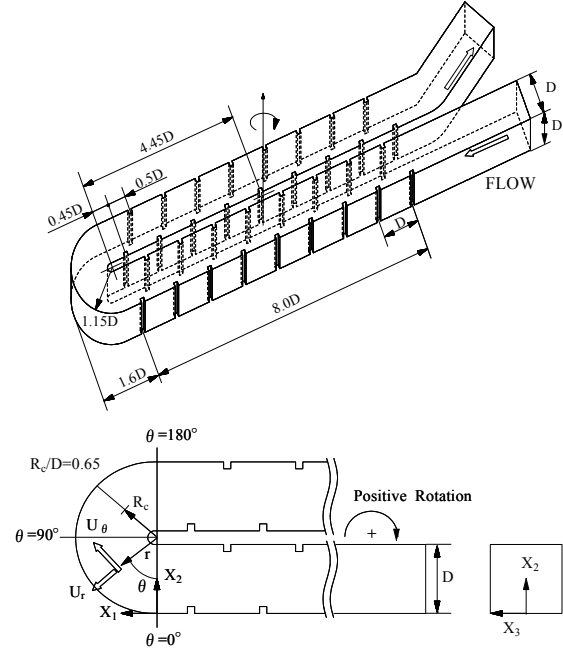


Figure 6
Rotating U-bend with roughened wall and definition of coordinate system

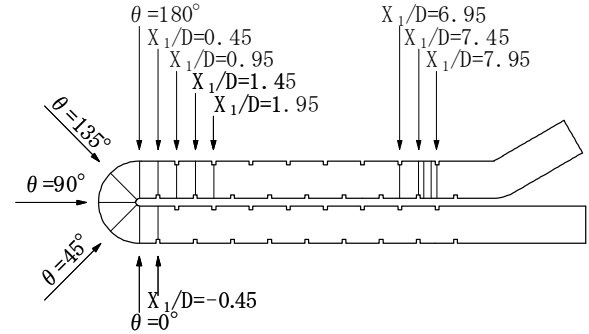


Figure 7
Measuring location of rotating U-bend with roughened wall

Reynolds stresses that appear in the transport equation must be solved to obtain the velocity fields completely. In this calculation, we have adopted the transport equation of Reynolds stresses to accurately predict anisotropic turbulence. The transport equation of Reynolds stresses is displayed exactly in the following form:

$$\begin{aligned} \frac{D\overline{u_i u_j}}{Dt} = & - \left(\overline{u_i u_k} \frac{\partial \overline{u_j}}{\partial X_k} + \overline{u_j u_k} \frac{\partial \overline{u_i}}{\partial X_k} \right) + \frac{p}{\rho} \left(\frac{\partial \overline{u_i}}{\partial X_j} + \frac{\partial \overline{u_j}}{\partial X_i} \right) \\ & - \frac{\partial}{\partial X_k} \left\{ \overline{u_i u_j u_k} - \nu \frac{\partial \overline{u_i u_j}}{\partial X_k} + \frac{p}{\rho} (\delta_{jk} \overline{u_i} + \delta_{ik} \overline{u_j}) \right\} \\ & - 2\Omega_k (\overline{u_j u_m} \epsilon_{ikm} + \overline{u_i u_m} \epsilon_{jkm}) \\ & - 2\nu \frac{\partial \overline{u_i}}{\partial X_k} \frac{\partial \overline{u_j}}{\partial X_k} \end{aligned} \quad (10)$$

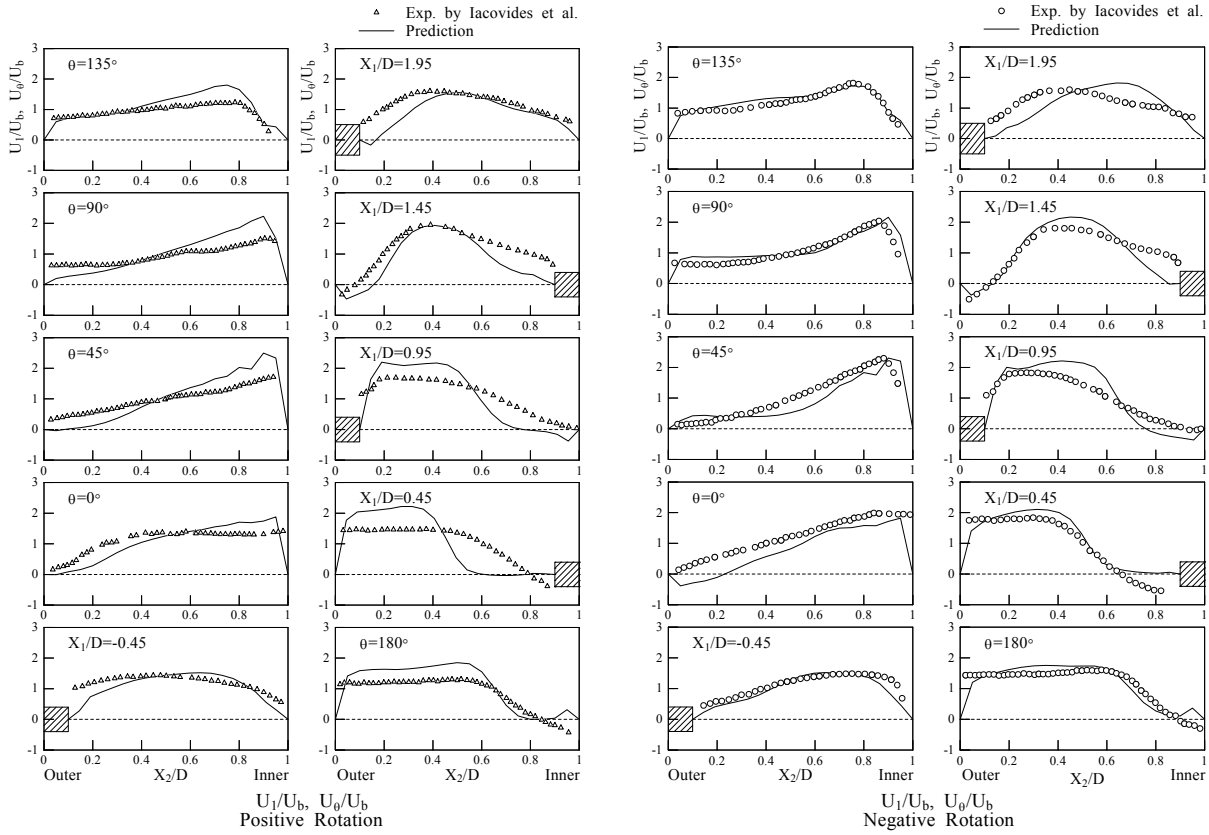


Figure 8
Comparison of streamwise velocity for positive and negative rotation

The forth term of right hand side is production term generated by rotation. It is impossible to numerically solve the above equation directly, so it is necessary to rewrite several of the terms of the Reynolds stress equation by introducing the concept of the turbulent model which is described in previous chapter.

Figure 6 shows schematic diagrams of rotating U-bend with roughened walls which is manufactured by Iacovides et al. [4] and coordinate systems including curved duct. A rotating U-bend of 0.05 m square cross-section and with a bend curvature ratio $Rc/D=0.65$ is mounted on a turntable so that the curvature axis of the duct is parallel to the axis of rotation. The rib-height to duct diameter ratio, h/D , is 0.1 and the pitch to height ratio, P_i/h , is 10. They have measured the turbulent flows of three cases, all for a flow Reynolds number of 100,000; stationary U-bend case, a case for a U-bend rotating positively at $Ro=0.2$ and a case with negative rotation at $Ro=-0.2$. Positive rotation represents clockwise direction as shown in Figure 6. Figure 7 shows the locations of measuring cross sections where the calculated results are compared with the experimental data of mean velocity and Reynolds stresses.

The inlet conditions of turbulent energy and dissipation were assumed to be $k = U_b^2 \times 10^{-5}$ and $\varepsilon =$

$k^{3/2}/D$, respectively, because the inlet condition were uncertain even in the experiment. Since the present model can be classified as a high-Reynolds-number turbulent model, the wall function method is adopted as the boundary condition for turbulent energy and dissipation at the wall. Although it has been pointed out that the wall function method is not suitable for separated turbulent flow, it is also true that the wall function method is effective way to save calculation time and to set easily boundary condition. Neumann condition is set as the outlet boundary condition. The governing equations were discretized by the differencing scheme, and QUICK scheme (quadratic interpolation for convective kinematics) was used for the convection term. The Reynolds number is 10,000, based on the hydraulic radius and the bulk velocity. The number of computational grid points in all over cross section is 21×21 , and 446 grid points are set along the streamwise direction. In the curved duct, the grids are arranged in 5-degree intervals.

The calculated results for the streamwise flow velocities are compared with the experimental data at symmetric plane $X_3/D=0$ as shown in Figure 8. In Figure 8, two rows viewed on the left and the right sides are compared results in the case of positive and negative rotation, respectively. Iacovides et al. [4]

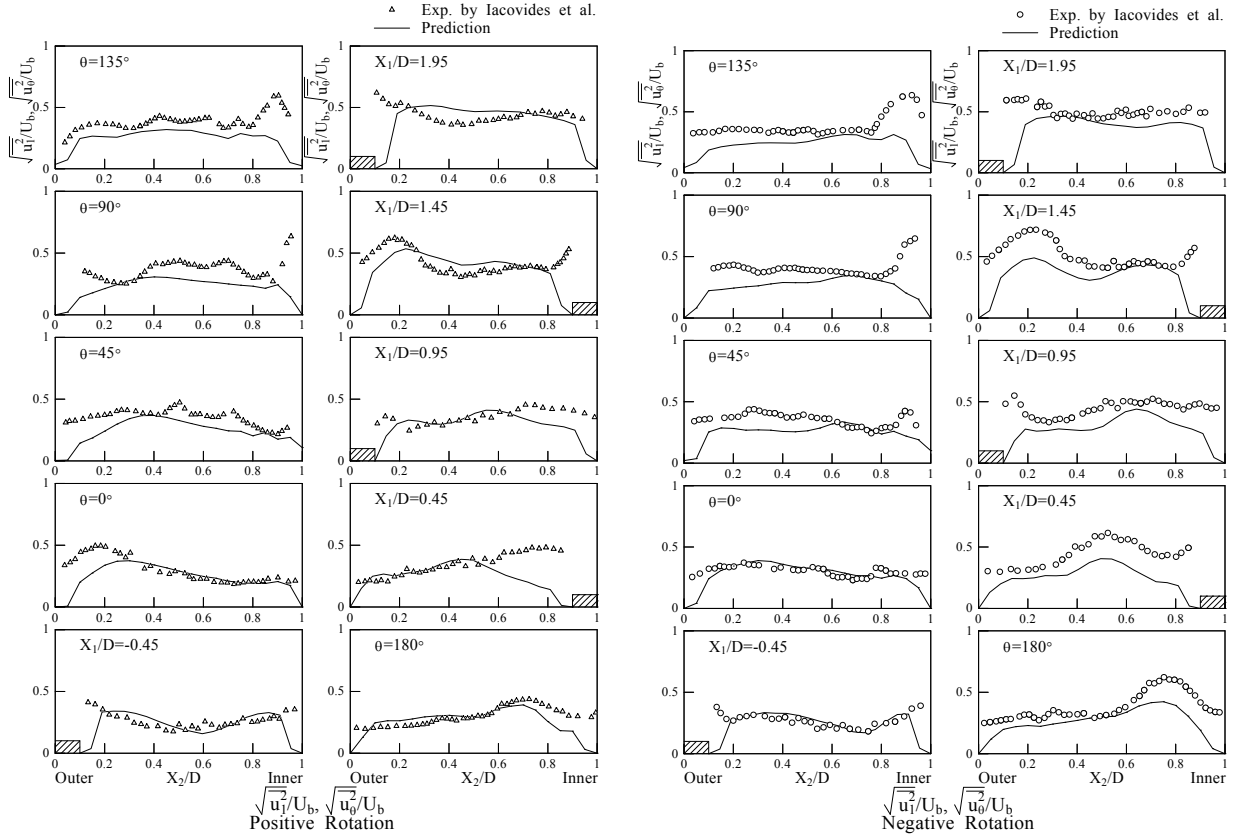


Figure 9
Comparison of streamwise normal stress for positive and negative rotation

have pointed out as characteristic features from the experimental data that streamwise velocity shows uniform profile in the case of positive rotation while, in the case of negative rotation, the streamwise velocity with high value is produced near the inner wall side. These characteristic features are caused by Coriolis's effects, i.e., Coriolis's force performs from the inner wall to the outer wall in positive rotation and from the outer wall to the inner wall in the case of negative rotation. When the calculated velocity profile in positive rotation is compared with that in negative rotation at $X_1/D = -0.45$, the maximum velocity is generated closer to the inner wall side in negative rotation than in positive rotation. Besides, these characteristic features are also recognized especially for the velocity profile of inlet cross section $\theta = 0^\circ$ of U-bend.

Figure 9 represents comparisons of the streamwise fluctuating velocity for both positive and negative rotations. The comparison results are displayed in two rows for each rotation in the same manner as results for the streamwise velocity. The calculated and the experimental results are normalized by the bulk velocity. Both results of the experiment and

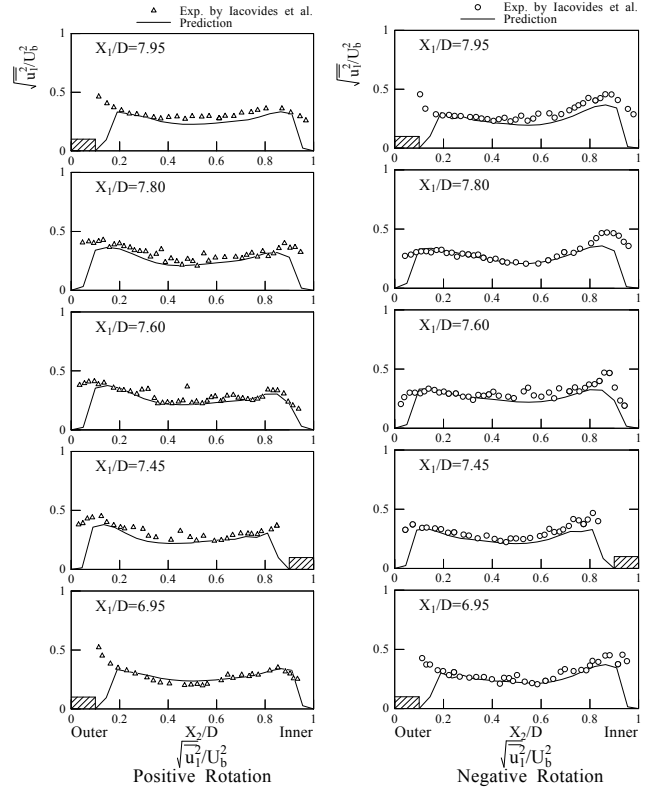


Figure 10
Comparison of streamwise normal stress in straight duct for positive and negative rotation

calculation at $X_1/D = -0.45$ section explain that the distributions of the streamwise fluctuating in positive rotation is different from that in negative rotation. The production term of the streamwise fluctuating velocity induced by rotation, which is the forth term of right side of Reynolds transport Equation (10), is consisted of the product term between angular rotation speed of duct and shear stress $-\overline{u_1 u_2}$. This means that the stress $-\overline{u_1 u_2}$ has influence on the production of the streamwise fluctuating.

Adding to this, it is found from the experimental result of $\theta = 0^\circ$ cross section in positive rotation that the maximum value of streamwise fluctuating velocity is generated near the outer wall. This characteristic feature can be reproduced by the presented calculation. Contrary to this result, it is also pointed out as different feature that the experimental data shows peak value in the vicinity of the inner wall side at $\theta = 90^\circ$ and 135° cross sections, but the calculation is not able to predict such peak value. These differences between calculated and experimental results imply that it is necessary to predict correctly the separated and reattachment points of separated flow.

Figures 10 represents the comparison of the streamwise fluctuating velocities in straight duct located downstream of curved duct. All cross sections in figures are situated after $X_1/D = 6.95$ where corresponds to the fourteenth rib counted from outlet of curved duct. Since the influence of separated region on flow behavior decays gradually as the flow is far from curved duct, the calculated results are good agreement with the experiment.

The compared results of the other Reynolds stress components are presented minutely in the research work reported by Sugiyama et al.[21].

Turbulent flow separation in a rectangular duct with a 180-degree sharp turn: Figure 11 shows schematic diagrams of rectangular channels with 180-degree sharp turn. The experimental flume with a rectangular cross section has a width of 50 mm, a depth of 25 mm, and a length of 334 mm. The Reynolds number is 3.5×10^4 based on hydraulic diameter and bulk velocity. In this calculation, the channel width in the sharp turn section is 70 mm.

The inlet length of the rectangular duct is the same as that of the experimental apparatus, and an outlet length of thirty times the hydraulic diameter is set downstream from end wall in order to make use of the free stream condition at the outlet section in the numerical analyses. Reynolds number 3.5×10^4 in the calculation is the same of the experiment. The

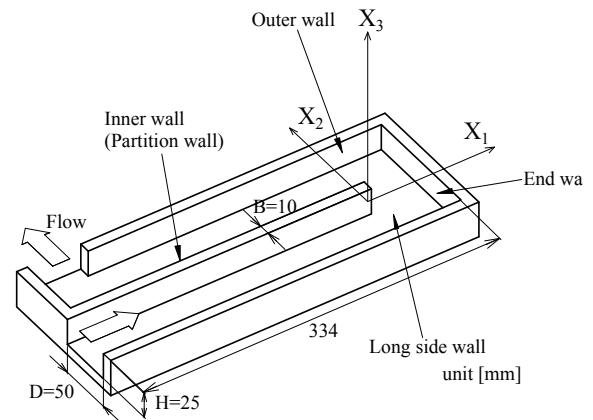


Figure 11

Rectangular duct with a 180-degree sharp turn

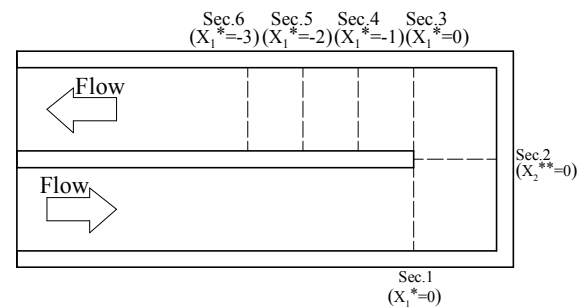


Figure 12

Measuring location of rectangular duct with a 180-degree sharp turn

each rectangular cross section has a total of 45×23 computational grid points, and 218 grid points are set along the main flow direction. Therefore, the total number of computational grids is 225,630.

Figure 12 shows the measurement sections which are denoted as Sec.1 through 6. The calculated results for the main flow velocities are compared with the experimental data at the six sections, as shown in Figure 13. When the location of the zero-valued contour line is considered, the calculation correctly reproduces the experimental contour lines, except for those of Sec.5. For Sec.5, the experimental zero-valued contour line is located a little bit far from the inner wall, whereas the calculated contour line is located near the inner wall.

The vector plots of Figure 14 present a comparison of the secondary flow for cross sections of the streamwise flow. The calculated rotating direction shows the opposite direction of the experiment because symmetrical cross sections with respect to X_2 axis are compared. In Sec.1, the calculation predicts the secondary flow to move uniformly from the outer wall to the inner wall, as is the case for the experimental results. This secondary flow behavior for a curved duct with a circular cross

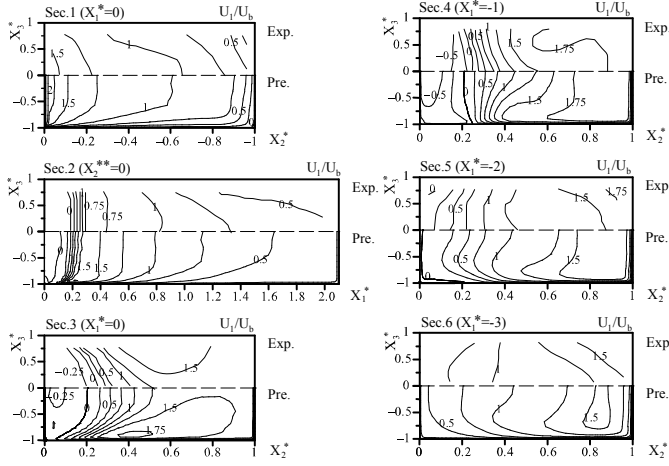


Figure 13
Comparison of streamwise velocity

section has been reported to be caused by the pressure imbalance between the low pressure of the inner wall side and the high pressure of the outer wall side. The experimental measurement of Sec.2 shows the generation of the circulation flow along the upper wall, which is formed by the secondary flow. Although the proposed methods also reproduce this circulation flow, the region covered by the circulated flow is larger and clear than in the experiment.

The cross sections of Sec.3 through 5 are located only in the separated flow generated in the straight duct following the curved duct with a sharp turn. The experimental results for Sec.3 and 4 indicate that the secondary flow located in the separated region has a low velocity on the inner wall side, and, in contrast, except for the separated region, the secondary flow

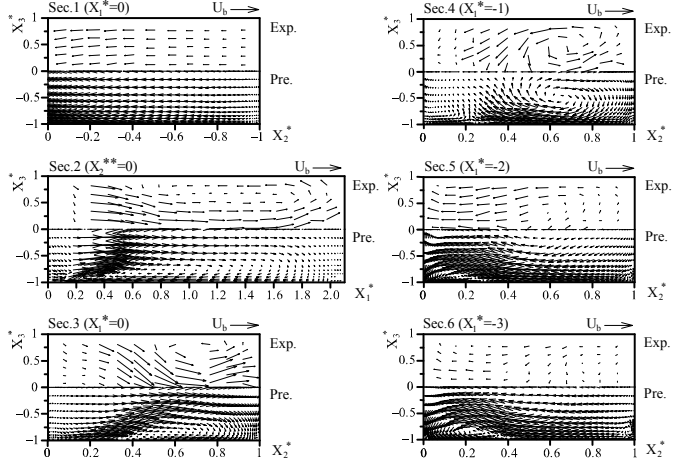


Figure 14
Comparison of secondary flow vectors

has a high value for the circulation flow. The calculated results also reproduce these characteristic features.

Figure 15 compares the results for the streamwise velocity vectors in the X_1 - X_2 plane. In both the experimental and calculated results, large-scale and small-scale separations are observed in the downstream straight duct and in the corner regions of the curved duct with a sharp turn. Adding to this, it is found from the both results that accelerated velocity is generated near the upper wall of straight duct because of turbulent flow separation. Although separated flows are recognized in both the experimental and calculated results, the calculated velocity profiles in the separated region differ slightly from the experimental values.

For the turbulent model, precise prediction of the reattachment point of the separated flow is important. Therefore, the distribution of the stream function is compared with the experimental results for the velocity field. Generally, the stream function

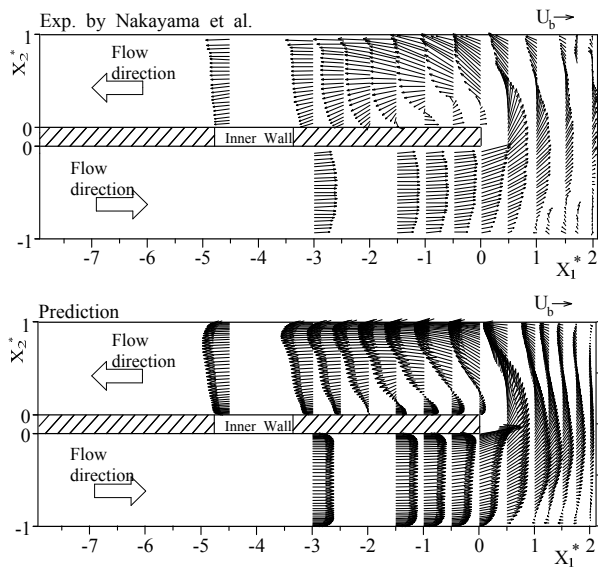


Figure 15
Comparison of streamwise velocity vectors

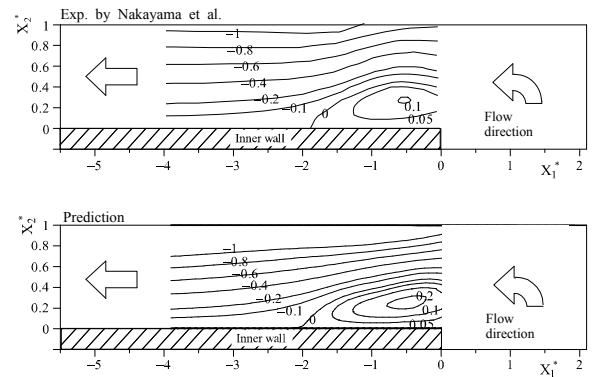


Figure 16
Comparison of distribution of stream function

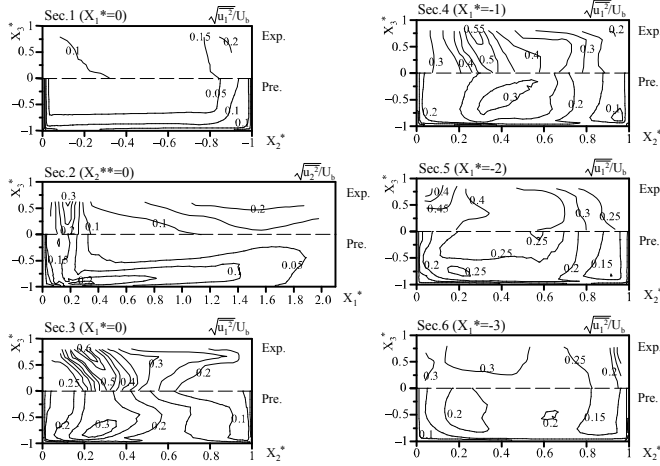


Figure 17

Comparison of streamwise normal stress

is used for two-dimensional flows, but the distribution of the stream function is introduced in the experiment in order to clarify the reattachment point of the separated flow. Figure 16 compares the results for the stream function in the X_1 - X_2 plane. The presented methods predict well the reattachment point without great discrepancy.

Figures 17 and 18 show the compared results of streamwise and vertical normal stresses, respectively. The calculated contour lines are normalized by the average mean velocity, as are the experimental results. For Sec.1 in Figure 17, the calculated and experimental maximum values are generated near the outer wall. Further downstream, in Sec.2, the experimental maximum value appears on the inner wall side. This coincides with the location of the shear layer generated by the separated flow. The experimental results for Sec.3 through 5, which are located in the separated flow, show that the maximum value is generated near $X_2^* = 0.2$ of the upper wall in all cross sections. On the other hand, the calculated results for Sec.3 show the same tendency. However, except for Sec.3, the maximum values occur in the central region of the cross sections, the feature of which differs from the experimental feature. In Sec.6, which is located after the reattachment point, the maximum value is produced near the inner wall for both results.

Generally, the vertically normal stress is high along the vertical walls in the straight rectangular duct. The calculated results for Sec.1 clearly have this characteristic feature, which is not always maintained through the entire flow and changes gradually as the flow develops. In the separated region, the high value near the inner and outer walls in Sec.1 changes its value because of the separated flow, as shown in Sec.

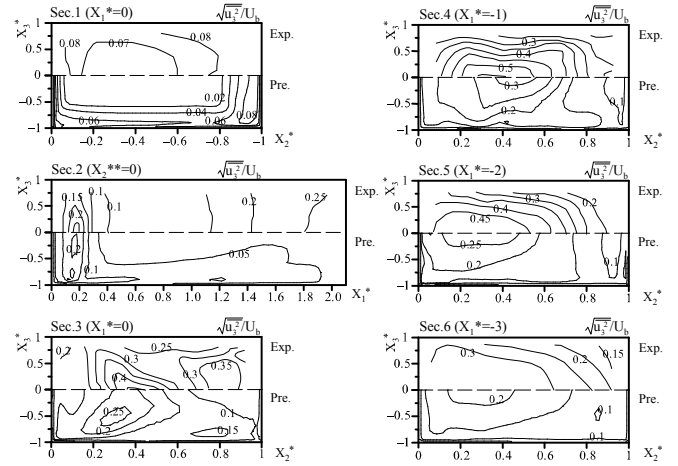


Figure 18

Comparison of vertical normal stress

2. Based on the comparison results for Sec.3 through 5, a feature common to both the calculation and experimental results is that the maximum value is produced in the central region of the cross section and the location of the maximum value moves toward the inner wall as the flow develops. Further downstream, in Sec.6, the maximum value of the vertical normal stress for the calculation appears on the inner wall side, rather than on the outer wall side, which is the same for the streamwise normal stress.

In this calculation for turbulent flow separation, wall function method is adopted as boundary condition of turbulent energy and dissipation. It is important to examine the validity of wall function method. As for this controversy problem, Sugiyama et al.[22] presented modified wall function method including the calculated results of rectangular duct with 180-degree sharp turn.

CONCLUSIONS

In this paper, simulated results for three types of complicated turbulent flows are presented, which are predicted by using algebraic Reynolds stress and turbulent heat flux models, in order to examine the capabilities for predicting turbulent flow with and without heat transfer. Calculated results are compared with the experimental data in detail to make the difference between calculated results and the experimental results clear. Although the agreement with experiment is certainly not perfect in detail, the main features, such as the secondary flow of the second kind generated by anisotropic turbulence, are able to reproduce reasonably by the presented models and numerical method. As a result of these calculations, it is found that algebraic

Reynolds stress and turbulent heat flux models are practicable approximation for a various range of complex engineering applications such as heat transfer enhancement, turbine blade problem, etc.

REFERENCES

1. Humphrey, J. A. C. and Whitelaw, J. H., 1980, Turbulent flow in a duct with roughness, *Turbulent Shear Flows* 2. Springer-Verlag, Berlin Heidelberg, Germany, pp.174-188.
2. Fujita, H., Yokosawa, H., Hirota, M. and Nagata, C., 1988, Fully developed flow and heat transfer in a square duct with two roughened facing walls, *Chem. Eng. Comm.* **74**, pp.95-110.
3. Hirota, M., Fujita, H., Itakura, S. and Yuasa, H., 1997, Turbulent heat transfer in square ducts with rough walls, *Proc. 34th National Heat Transfer Conference*, pp.347-348(in Japanese).
4. Han J.C., Chanda P.R. and Lau, S.C., 1988, Local heat/mass transfer distributions around sharp 180 degree turns in two-pass smooth and rib-roughened channels, *J. Heat Transfer, ASME*, **110**, pp.91-98.
5. Ekkad S.V., Pamula G. and Acharaya S., 2000, Influence of crossflow-induced swirl and impingement on heat transfer in an internal coolant passage of a turbine airfoil, *J. of heat transfer, ASME*, **122**, pp. 587-597.
6. Iacovides H., Jackson D.C., Kelemenis G., Launder B.E. and Yuan Y-M., 1996, LDA study of the flow developing through an orthogonally rotating U-bend of strong curvature and rib roughened walls. Rodi W., Bergeles G. (Eds.), *Engineering Turbulence Modelling and Experiments* 3, pp.561-570.
7. Iacovides H., 1999, The computation of turbulent flow through stationary and rotating U-bends with rib-roughened surface. *Int. J. Numer. Meth. Fluids* **29**, pp.865-876.
8. Liou, T. M., Chen, C. C. and Chen, M. Y., 2003, Rotating effect on fluid flow in two smooth ducts connection by a 180-degree bend, *Transactions of the ASME, J. Fluid Eng.*, **125**, pp.138- 148.
9. Hsieh, S. S., Chen, P. J. and Chin, H. J., 1999, Turbulent flow in a rotating two pass smooth channel, *Transactions of the ASME, J. Fluids Eng.*, **121**, pp.725- 734.
10. Iacovides, H. and Raisee, M., 1999, Recent progress in the computation of flow and heat transfer in internal cooling passages of turbine blades, *Int. Journal of Heat and Fluid Flow* **20**, pp.320-328.
11. Nakayama, H., Hirota, M., Fujita, H., Yamada, T. and Koide, Y., 2003, Flow characteristics in rectangular ducts with a sharp 180-degree turn, *Transactions of the JSME* **69-681**, pp.1171-1179 (in Japanese).
12. Rodi, W., 1976, A new algebraic relation for calculating the Reynolds stresses. *ZAMM* (56), T219-T221.
13. Launder, B.E., Reece, G.J. and Rodi, W., 1975, Progress in the development of a Reynolds stress turbulent closure., *J. Fluid Mech.* **22**, pp.537-566.
14. Sugiyama, H. and Hitomi, D., 2005, Numerical analysis of developing turbulent flow in a 180° bend tube by an algebraic Reynolds stress model. *Int. Journal for Numerical Methods in Fluid.* **47**, pp.1431-1449.
15. Champagne, F. H., Harris, V. G. and Corrsin, S., 1970, Experimental on nearly homogeneous turbulent shear flow. *J. Fluid Mech.* **41**, pp. 81-139.
16. Lumley, J. L., 1975, Introduction. In *Prediction methods for turbulent flows. Lecture Series 76*, von Karman Inst., Belgium.
17. Monin, A. S., 1965, On the symmetry properties of turbulence in the surface layer of air. *Atmos. and Oceanic Phys.* **1** (1), pp.25-30.
18. Launder, B. E., 1976, Heat and mass transport. In *Turbulence, Topics in Applied Physics* 12. Springer-Verlag, Berlin Heidelberg, Germany, pp.231-287.
19. Naimi, M., Gessner, F.B., 1997, Calculation of fully-developed turbulent flow in rectangular ducts with non-uniform wall roughness. *J. Fluids Eng., Trans. ASME* **119** (3), pp.550-558.
20. Sugiyama, H., Akiyama, M., Nemoto, Y. and Gessner, F.B., 2002, Calculation of turbulent heat flux distribution in a square duct with one roughened wall by means of algebraic heat flux model, *Int. Journal of Heat and Fluid Flow* **23**, pp.13-21.
21. Sugiyama, H. and Watanabe, C., 2003, Numerical analysis of developing turbulent flow in a rotating U-bend of strong curvature with roughened walls, *Trans. Atomic Energy Society, Vol.2., No. 2*, pp. 88-98(in Japanese).
22. Sugiyama, H., Tanaka, T. and Mukai, H., 2008, Numerical analysis of turbulent flow separation in a rectangular duct with a sharp 180-degree turn by algebraic Reynolds stress model, *Int. Journal of Heat and Fluid Flow* **56**, pp.2207-2228.