PVP-Vol. 491-1, Computational Technologies for Fluid/Thermal/ Structural/Chemical Systems With Industrial Applications — 2004
Volume 1
July 25–29, 2004, San Diego, California USA
PVP2004-3094

Computational Analysis of Heat Transfer Enhancement Due to Rectangular Ribs in a Turbulent Duct Flow

Alok Chaube **P.K. Sahoo**
S.C. Solanki

Department of Mechanical and Industrial Engineering
Indian Institute of Technology Roorkee
Email-alokcdme@iitr.ernet.in

ABSTRACT

A computational analysis of heat transfer enhancement due to artificial roughness in the form of rectangular ribs has been carried out. A turbulence model is selected by comparing the predictions of different turbulence models with experimental results available in the literature. A detailed analysis of heat transfer variation within inter rib region is done by using the selected turbulence model. The analysis shows that peak in local heat transfer coefficient occurs at the point of reattachment of the separated flow as observed experimentally. The results predict a significant enhancement of heat transfer in comparison to that for a smooth surface. There is a good matching between the predictions by SST k-ω and experimental results.

INTRODUCTION

Artificial roughness up to laminar sub-layer to enhance heat transfer coefficient is used in various applications like gas turbine blade cooling channels, heat exchangers, nuclear reactors and solar air heaters. A number of experimental studies [1,2] in this area have been carried out but very few attempts of numerical investigation have been made so far due to complexity of flow pattern and computational limitations. In the present work an attempt is made to predict numerically, the details of both the velocity and temperature fields responsible for heat transfer enhancement. The presence of rib may enhance heat transfer because of interruption of the viscous sub layer, which yields flow turbulence, separation and reattachment leading to a higher heat transfer coefficient. The enhancement of heat transfer by flow separation and reattachment caused by ribs is significantly higher compared to that by the increased heat transfer area due to ribs(fin-effect) [3]. The heat transfer measurements results for two different steps, p/e = 14 & p/e = 8, indicate the importance of roughness geometry [4]. Liou et.al.[6] have performed a numerical and experimental analysis to investigate the heat transfer and fluid flow behavior in a rectangular channel flow with stream wise periodic ribs mounted on one of the principal walls. They have concluded that the flow acceleration and the turbulence intensity are two major factors influencing the heat transfer coefficient. The combined effect is found to be optimum for the pitch to rib height ratio equal to 10, which results in the maximum value of average heat transfer coefficient. Rau et.al. [5] experimentally found optimum pitch to rib height ratio to be equal to 9. Hence these investigations reveal that not only the rib geometry but also its geometrical arrangement play vital role in enhancing the heat transfer coefficient. Karwa [7] has reported an experimental investigation for the same configuration for the Reynolds number range of 4000-16000. The main aim of the present analysis is to investigate the flow and heat transfer characteristics of a 2 dimensional rib roughened rectangular duct with only one principal (broad) wall subjected to uniform heat flux by making use of computer simulation. The ribs are provided only on the heated wall. The other three walls are smooth (without ribs) and insulated. Such a case is encountered in solar air heaters with artificially roughened absorber plate.

The following sections present solution domain, turbulence models, the best model selection for the present case and then results and discussion of the detailed analysis with the selected turbulence model.

Nomenclature

G_k	turbulent kinetic energy due to mean Velocity gradient
G_b	turbulent kinetic energy due to Buoyancy
h	heat transfer coefficient
k	turbulent kinetic energy
Nu	Nusselt number
p/e	relative roughness pitch
St	Stanton number Nu/Re Pr
S_k, S_ε	source terms
u'	fluctuating axial velocity component
Γ_k, Γ_ω	Effective diffusivity of k & ϵ
ρ	density of air
ϵ	dissipation rate
ω	specific dissipation rate
δ	boundary layer thickness
μ	dynamic viscosity
μ_t	turbulent viscosity
y^+	dimensionless distance from the wall (= $\rho u_\tau y/\mu$)
Subscript "$_s$"	smooth duct

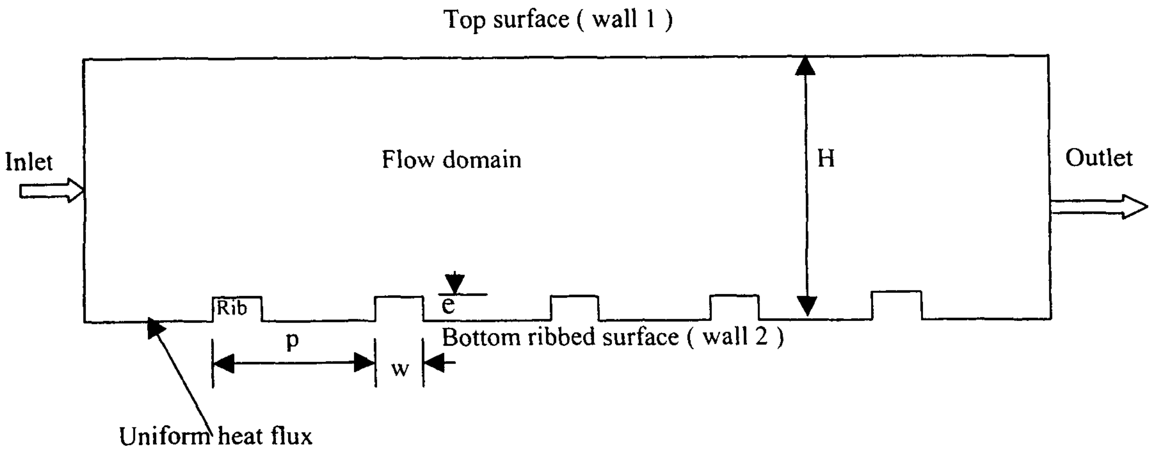

Fig.1 Ribbed duct geometry and the solution domain

Solution Domain

The solution domain shown in Fig.-1 has been selected as per the experimental details given by Karwa [7]. A rectangular duct with the duct height(H) of 40 mm, rib height(e) of 3.4 mm, rib width of 5.8 mm and pitch(p) of 34 mm has been taken for analysis. In the experimental details[7], the thickness of the heated plate is only 1 mm, which is very small in comparison to the surface area normal to the heat flow. Hence the approximate Biot number is also very small (less than 10^{-3}), which allows us to neglect the internal resistance in comparison to convective resistance. Therefore the uniform heat flux of 4 kW/ m^2 is given on ribbed surface, neglecting the conduction resistance within the plate. A 2-D CFD analysis of heat transfer and fluid flow through a high aspect ratio (7.5) rectangular duct with transverse ribs provided on a broad, heated wall and other wall smooth and insulated, is carried out using Fluent 6.1 software package.

A non-uniform rectangular mesh with grid adoption for y+=1 at adjacent wall region is applied as shown in Fig. 2. To minimize the grid size effect, the no. of cells are varied from 27650 to 66524 in various steps. It is found that after 60910 no. of cells, the further increase in cells has negligible effect on the results. Similar analysis is carried out for a smooth duct of the same dimensions for similar range of Reynolds number 3000 to 15000 to find out the ratio of Stanton number for ribbed duct and smooth duct as experimentally determined by Karwa [7]. The inlet velocity ranging from 0.68 m/s to 3.4 m/s, outlet pressure equals to atmospheric pressure and no slip at wall boundary conditions are used for the analysis.

Selection of Turbulence Model

The low – Reynolds number models are used for near wall regions because the high – Reynolds number models do not perform well in these regions. For example standard k-ε model and RSM model [8] do not work well near wall region where k & ε go to zero. Large numerical problems appear in the ε- equation as k becomes zero. The destruction term in ε - equation includes ε^2/k , and this causes problem as $k \rightarrow 0$ even if ε also goes to zero; they must go to zero at an appropriate rate to avoid such a problem and this is often not the case. Similarly LES model does not catch the small eddies near wall. Taking above difficulties into the consideration the low Reynolds number models have been developed. The k-ω model replaces dissipation rate (ε) term by specific dissipation rate(ω) term which transfers the k from denominator to numerator in the specific dissipation rate equation to avoid numerical difficulties. The RNG k-ε model developed using renormalization theory to modify the k-ε model for near wall region by including additional term in ε equation [9]. The realizable k-ε model contains a new formulation for turbulent viscosity and new transport equations for ε have been derived from an exact equation for the transport of the mean – square vorticity fluctuations.

For the flow situations where core and wall bounded regions both are to be modeled with the same accuracies, the blending of both types of models can give satisfactory performance. The Shear Stress Transport(SST) k-ω model is developed using blending function between k-ε and k-ω models. It is developed to effectively blend the robust and accurate formulation of k-ω model in the near wall region with the k-ε model for the core flow region. Both the standard k- ω model and k-ε model are multiplied by a blending

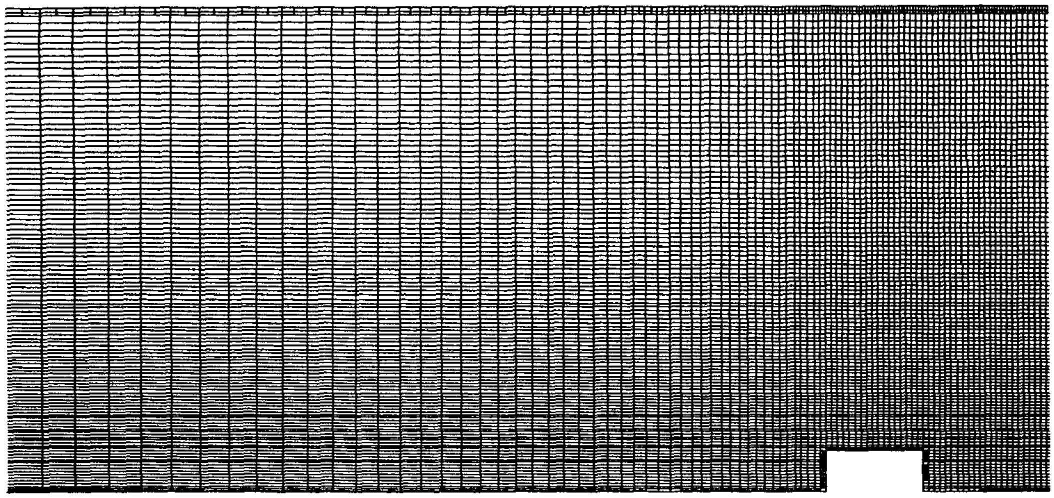

Fig.-2 Rectangular mesh with grid adoption for y+=1 at top and bottom wall

function is designed to be one in the near wall region, which activates the standard k-ω model and zero away from the surface, which activates the k-ε model. This blended model is called Shear Stress Transport k-ω model or **SST k-ω model** [8].

The experimental results of heat transfer enhancement in the form of graph between Stanton number ratio of ribbed and smooth duct versus Reynolds number are compared with the computational results of different models. After comparing the performance of different selected models (Standard k-ω, Renormalization-group k-ϵ, Realizable k-ϵ and Shear stress transport k-ω) with the experimental results, the SST k-ω model is found to yield results closer to the experimental results as compare to other models. Hence further analysis is carried out using SST k-ω model. The comparison between different models and experimental results is shown in Fig.3. The present 2D numerical analysis with the SST k-ω model predicts with the average deviation of about 4.75% and maximum deviation of about 7.4%. The experimental results have uncertainty of about 6-7 % [7]. Hence the prediction through this model can be taken to be very good.

Results and Discussion

The results, both detailed (velocity field) and overall (Stanton number & heat transfer coefficient) are obtained and presented here for flow and convective heat transfer in a high aspect ratio (7.5) rectangular duct (as encountered in solar air heaters) with rectangular transverse ribs provided on one of the broad walls and other three walls are smooth (i.e. commercial surfaces without artificial ribs) . The surface with ribs is heated and subjected to constant heat flux while other smooth surfaces are adiabatic. The results for Reynolds number equal to 5000 are presented here. Similar pattern exists for the Reynolds number range of 3000 to 15000. The solar air heaters are normally operated in this range of Reynolds number [1].

Further analysis is carried out at Reynolds number of 5000 to understand heat transfer and flow characteristics at inter-rib regions. Contours of stream function in the Fig.-4 depicts flow pattern in the duct. Fig. 5 shows the x-velocity (u) distribution around the ribs. Fig. 6 is x-velocity gradient plot, which gives exact position of reattachment point and length of reattachment. At point of separation du/dy becomes zero, moving from positive to negative value. Again it becomes zero at point of reattachment, moving from negative to positive value. The estimated reattachment length is given in Table-1. The flow-pattern, which exists, is clearly depicted on Fig. 4 & 5. Fig. 4 clearly shows the dead regions (eddy formation) adjacent to a rib, both on upstream and down-stream sides. The down streamside has larger eddies. These are regimes of low heat transfer rate (as Fig. 7 reveals) leading

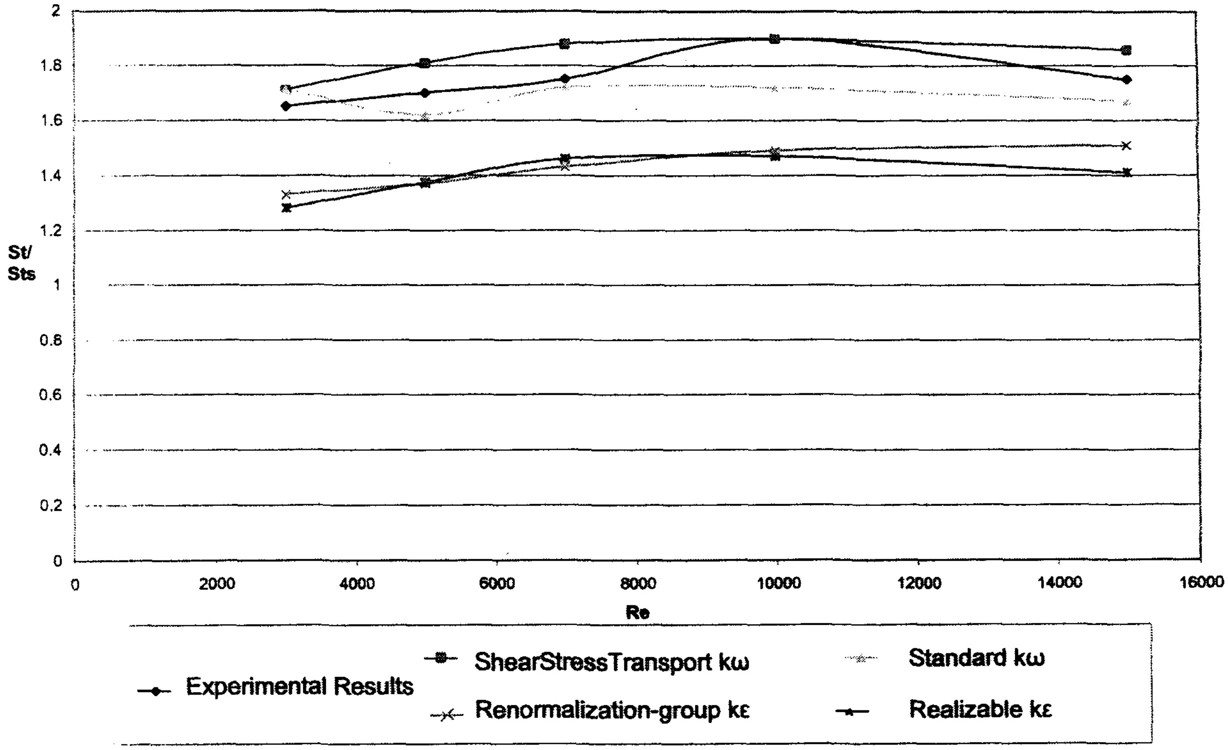

Fig. 3 Comparison of different models with experimental results

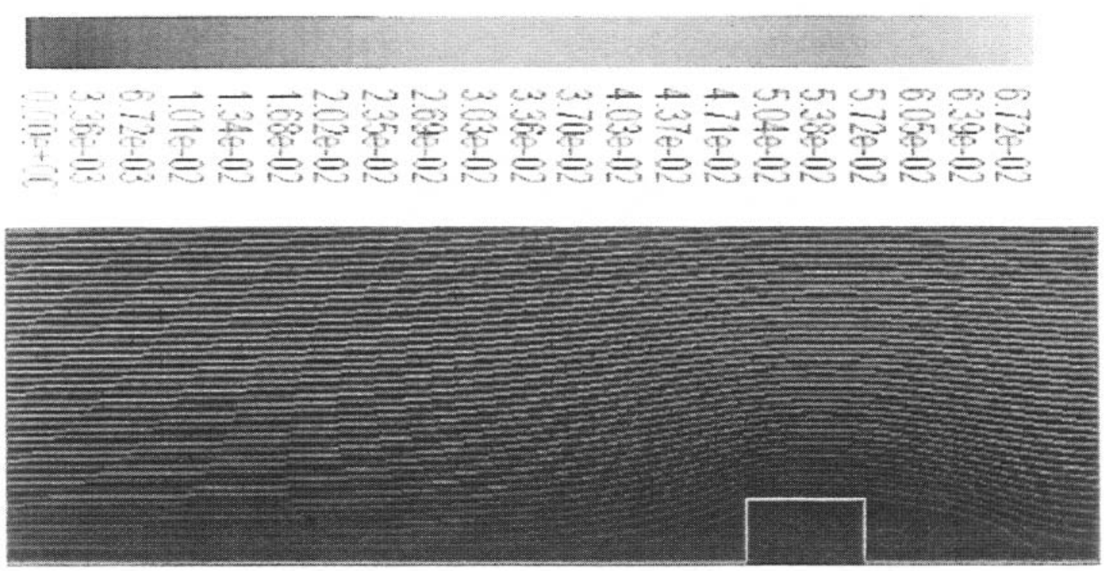

Fig. 4 Contours of stream function (kg/s)

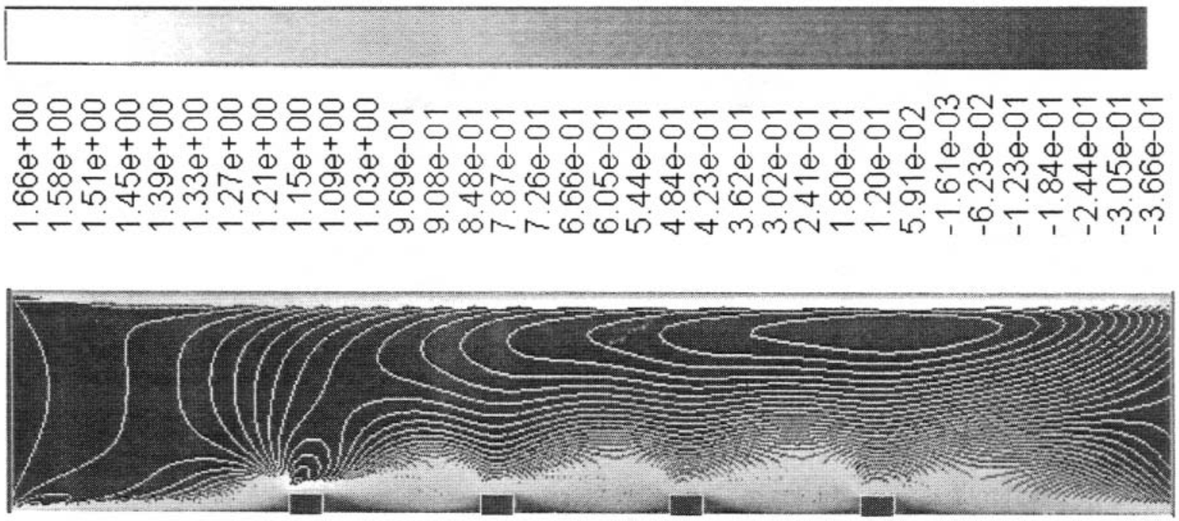

Fig. 5 Contours of x-velocity, (u), (m/s)

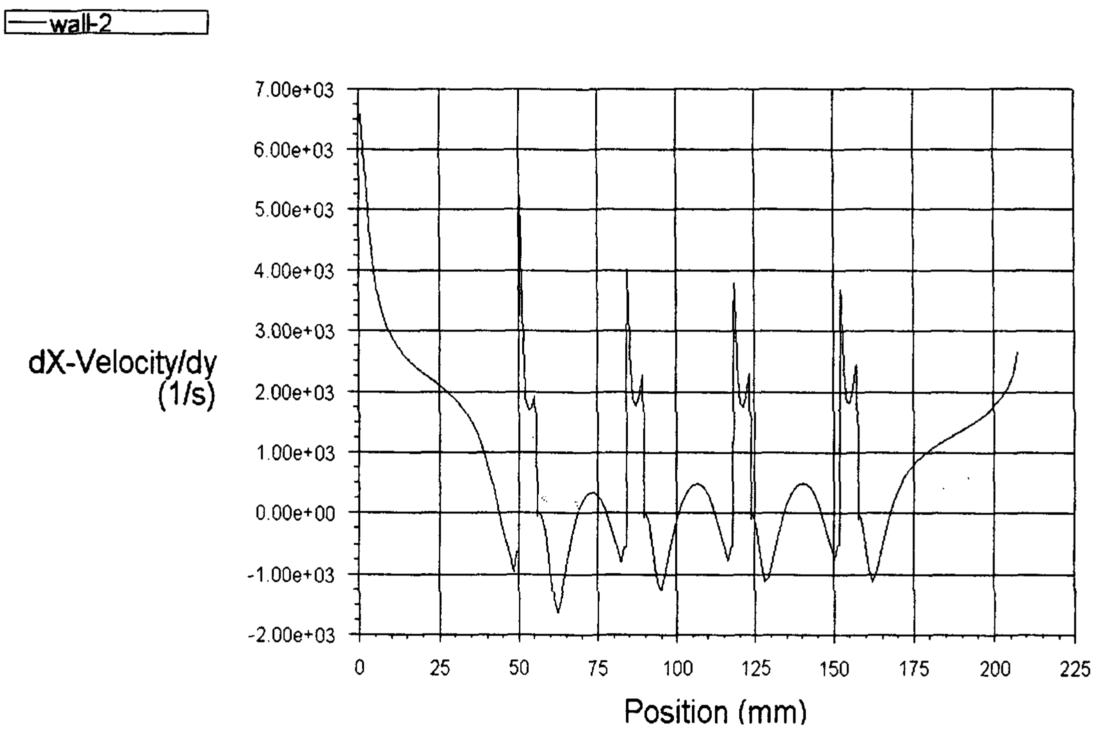

Fig. 6 X-velocity gradient plot (wall 2)

S - Point of separation

R - Point of Reattachment

to hot spots. In Fig. 5 the dark area with negative x-velocity around ribs indicates the existence of separation and reattachment of flow over heated surface. Fig. 7 indicates that the peaks in local heat transfer coefficient occur at the reattachment point as observed by many investigators [3-6]. It is also observed that the peak in local heat transfer coefficient and reattachment length is maximum between first two ribs, reduces gradually as the flow passes down stream over the successive pairs of ribs. This is the indication that the development of flow is occurring. In the developed region both the reattachment point and the heat transfer coefficient do not vary from one region to another. To find out enhancement in heat transfer coefficient due to artificial roughness, a similar analysis with the same parameters is carried out for a smooth duct. We take the commercially available metallic duct, which follows the Blassius correlation for friction factor on the smooth duct. The ratio of average heat transfer coefficient of ribbed duct and the smooth duct is found to be 1.81, which is significant.

TableNo1Reattachment length between successive ribs

S.No.	Location of reattachment point	Reattachment length
1	Between First & Second Rib	12.2 mm
2	Between Second & Third Rib	11.1 mm
3	Between Third & Fourth Rib	10.2 mm
4	Between Fourth & Out-let	9.4 mm

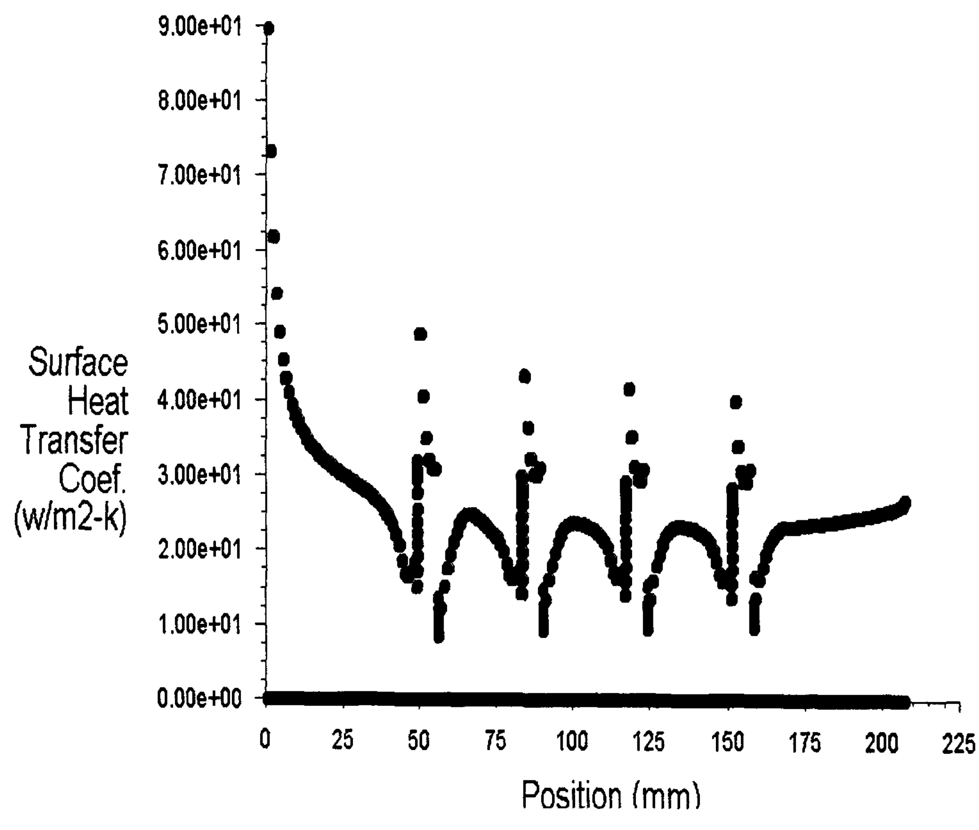

Fig. 7 Variation of local heat transfer coefficient in ribbed duct

Conclusions

From the present CFD analysis for heat transfer and fluid flow characteristics in a rectangular duct (high aspect ratio) having rectangular, transverse ribs on one of its broad walls, which is subjected to uniform heat flux and other three walls smooth & insulated, the following conclusions can be drawn:

1. A very good comparison with the experimental results available in the literature demonstrates the good capabilities of the selected Shear Stress Transport k-ω turbulence model for predicting the local heat transfer coefficients for the ribbed surfaces.
2. The peaks of local heat transfer coefficient are found at the reattachment point.
3. The peaks of local heat transfer and reattachment lengths decreases in stream wise successive inter rib regions.
4. The enhancement of average heat transfer coefficient due to artificial roughness is found about 81% for a typical case ,which is significant

References

1. Muluwork, K.B., Solanki, S.C., and Saini, J.S., 2000, " Study of heat transfer and friction in solar air heaters roughened with staggered discrete ribs," Proc. 4th ISHMT- ASME Heat and Mass Transfer Conf.,Pune, India, pp.391-398.

2. Karwa, R., Solanki, S.C., and Saini, J.S., 2001, " Thermo-hydraulic performance of solar air-heaters having integral chamfered rib roughness on absorber plates," Energy **26**, 161-176 [2001].

3. Lee, C.K., and Abdel-Moneim, S.A., 2001, " Computational Analysis of Heat Transfer in Turbulent Flow Past a Horizontal Surface with 2-D Ribs," Int. Comm. Heat Mass Transfer, Vol. **28** No.2,pp. 161-170.

4. Slanciauskas, A., 2001, " Two friendly rules for the turbulent heat transfer enhancement" Int. Jr. of Heat and Mass transfer **44**, 2155-2161.

5. Rau, G., Cakan, M., Moeller, D., and Arts, T., 1998, "The Effect of Periodic Ribs on the Local Aerodynamic and Heat Transfer Performance of a Straight Cooling Channel,"ASME Vol. **120**, pp. 368-375.

6. Tong-Miin Liou, Jenn-Jiang Hwang and Shih-Hui Chen, "Simulation and measurement of enhanced turbulent heat transfer in a channel with periodic rib on one principal wall" Int. Jr. Heat Mass Transfer, Vol. **36**, pp. 507-517 [1993].

7. Karwa, R., 2003, " Experimental Studies of Augmented Heat Transfer and Friction in Asymmetrically heated Rectangular Ducts with Ribs on the Heated Wall in Transverse, Inclined, V-Continuous and V-Discrete Pattern,"Int. Comm. Heat Mass Transfer, Vol. **30**, No. 2, pp. 241-250.

8. Fluent 6.0 User's Guide Vol. **2**.

9. Davidson, Lars , 1997 "An Introduction to Turbulence models," , Chalmers publication.

10. Versteeg, H.K., and Malalasekera, W., 1995, "An Introduction to Computational Fluid Dynamics", Longman publication.

11. Kakac, Sadik., Shah, Ramesh K., and Aung, Win., 1987, "Handbook of Single Phase Convective Heat Transfer" by John Wiley & sons publication.