Numerical Investigation of Heat Transfer in Square Duct with 45° Rib-Grooved Turbulators

Ananda Theertha K. R.[1], Narahari H. K.[2], Rajan N. K. S.[3]
1-M. Sc. [Engg.] Student, 2-Head of Department,
Automotive and Aeronautical Engineering,
M. S. Ramaiah School of Advanced Studies, Bangalore 560 058.
3-Principal Scientist, CGPL, Dept of Aerospace, IISc, Bangalore, 560 054.

Abstract

Gas turbine plays a vital role in the present day industrial society as they are being extensively used for land based power generation and in aircraft propulsion. There is constant challenge to increase the efficiency of the gas turbine engines which can be increased by raising the turbine inlet temperature. Present day advanced gas turbine blade inlet temperatures can be as high as 1700°C, here as blade materials are capable of withstanding only 1200°C to 1300°C. Cooling air, extracted from the compressor is around 650°C, is passed through the airfoil sections of the blade which lower the temperature to about 1000° C which is safe and permissible for reliable operation of the engine. It is practically very difficult and costly to obtain experimental data on heat transfer and the pressure losses in thin airfoil turbine blade sections at such temperatures and rotational speeds. Hence data generated from numerical investigations will play vital role in design, development and improving the efficiency of gas turbine engines. Flow tripping geometries like pin fin, and dimples are generally used in the trailing edge regions, where as ribs or tabulators are located at middle of the airfoil to enhance the heat transfer.

In this paper, analysis is carried out on three different combinations which are simple U duct, ribs aligned at 45° to flow direction and combination of ribs and grooves at 45° to flow direction. All turbulators are located on trailing face of duct. The simulations are carried on square duct having hydraulic diameter (Dh) of 0.0127m, Reynolds number of 25,000, rotational number (Ro) of 0.24, inlet density ratio (Δρ/ρ) of 0.13. Details for relative rib height (e), pitch distance between ribs (P), distance between ribs and groove centre (g), and the groove angle are similar to experimental reference. The numerical analysis has been carried out on Ansys CFX solving 3D compressible Navier Stokes equation along with k-ω turbulence model.

It was observed from the investigation, that the numerical results are in good agreement with the experimental results in validation stages. For the simple U duct and duct with ribs the simulations are carried out for one complete revolution and for rib-grooved case, only partial results are published. The present results for the turbulators show that there is significant enhancement in heat transfer from the heated wall to coolant when compared with simple U tube.

Key Words: Gas Turbine, Nusselt Number, Ribs, Grooves.

Nomenclature

D	Diameter, m
D_h	Hydraulic diameter, m
e	Relative roughness height
g	Groove position to pitch ratio
h	Heat transfer coefficient, W/m^{2}K
k	Turbulence kinetic energy
L	Length of duct, m
M	Mach number
Nu	Nusselt number
N_{us}	Nusselt number for smooth duct
P	Relative roughness pitch
Po	Pressure ,bar
R_e	Reynolds number
R_i, R_o	Inner and outer radius of 180° bend, m
R_r, R_t	Radius from axis of rotation , m
Ro	Rotation number, rpm
T	Temperature, K
V	Velocity, m/s
X'	Distance along duct measured with duct inlet (m)
γ	Ratio of specific heat
ρ	Density, kg/m^3
Ω	Angular speed of duct, rad/sec
ω	Dissipation rate of k per unit k
φ	Chamfer angle
θ	Groove angle

Abbreviations

AR	Aspect ratio
LF, TF	Leading face, Trailing face

1. INTRODUCTION

Gas turbine blades have complex cooling passages and it is difficult for designers to accurately predict the metal temperature. The aerothermodynamics in these airfoil sections are complex and large thermal gradient exists throughout the blade profile and heat transfer coefficients are high. Heat transfer on the pressure surface increases as the flow accelerates around the blade. For such complex flows, designers need extensive experimental and numerical data to aid them in the development of efficient cooling techniques. Detailed hot gas path, heat transfer distributions will aid in development of efficient airfoil section profiles. The surface heat transfer on a stator vane is affected by the combustor generated high turbulence, the laminar-to-turbulent transition, acceleration, film cooling flow, platform secondary flow and surface roughness. Additional to the above factors the Coriolis force generated due to rotation, rotating buoyancy forces, centrifugal forces and leakage must be considered for the rotating blade which makes design even more complicated. In order to counter such complicated problem cooling techniques like jet impingement coupled with film cooling is employed at the leading edge. Flow tripping geometries like pin fin, and dimples are generally used in the trailing edge regions, where as ribs or tabulators are located at middle of the airfoil to enhance the heat transfer. The physics behind improved heat transfer using ribs is due to secondary flows induced by the ribs and also breaking of viscous sub layer which results in reduced thermal boundary layer thickness and thermal resistance near the wall and this is further enhanced in angled ribs.

Since 1980's many experimental and numerical works have been carried out on prediction and estimation of heat transfer in cooling passage ducts. J. H. Wagner et al [1] conducted a detailed experimental study on non-rotating and rotating four pass duct to study the heat transfer with smooth wall and radial outward flow. The heat transfer data were obtained for different flow rates, rotation model radius and wall to coolant temperatures differences. The experiments were conducted by varying one parameter while holding the rest fixed. Data were analyzed separately to study the Coriolis effect, buoyancy effect in a rotating duct. Some of the major conclusions drawn was that in fully developed flows the heat transfer is strongly affected by rotation and heat transfer increase up to 3.5 times the non rotating duct value and buoyancy force in gas turbine blades are substantial because of the high rotational speeds and large blade-wall-to-coolant temperature differences. Mark A. Stephens et al [2] carried out detailed numerical study of simple U tube with square cross section with side wall dimension 0.127m (0.5inch) The work has been carried out for non- rotating and rotating duct with Re = 25,000 and with two rotational numbers 0.24 and 0.48. Secondary flows at various locations over the length of the tube were located. Secondary flow are developed in the up leg part of the leading face and the D- type secondary flows at the 180° bend which are mainly caused by rotational effects. Guoguang Su et al [3] conducted a numerical investigation on the heat transfer in a two pass rotating rectangular channel with smooth walls for different aspect ratio like 1:2, 1:2and 1:4. Some of the conclusions drawn are Nusselt number ratio decreases with increase in Reynolds number and the heat transfer is higher on the trailing face for the first pass and the trend is reversed in the second pass due to rotation. It was also concluded that the Nusselt number ratio becomes more pronounced for low aspect ratio rectangular channels. Mohammad Al-Qahtani et al [4] also conducted a similar numerical study for a two pass rectangular smooth channel with aspect ratio of 2 implementing Reynolds stress turbulence model. Here the simulations were conducted for two channel inclination 90° and 135° with respect to the direction of rotation. Conclusion were drawn like the movement of the cold fluid from leading face to the trailing face in the first pass is mainly because of the secondary flows induced by Coriolis force. Acceleration of the cold fluid near the trailing face and deceleration of the hot fluid on the leading face is mainly triggered by the buoyancy force. It was also observed that the Nusselt number ratio in the trailing face in first pass and the leading face in the second pass increases for high rotation numbers and density ratios. T.M. Liou et al [5] studied heat transfer for a 45° rib roughened in a rectangular duct for high rotation numbers. The study was done for two channel orientation 0° and 45°. The rotational numbers were varied from 0.1 to 2 for a Reynolds number range of 5,000 to 15,000. The heat transfer enhancement was in the range of 1.6 to 4.3 times for the 45° ribbed channels when compared with rotating smooth channel. The local Nusselt number along the leading centerline of the 45° channels was higher compared to the 0°channel. The individual and interactive correlation were developed between Re, Ro and Bu to calculate the Nusselt number at periodically developed rib and mid rib locations with channel orientation for 0° and 45°. Guoguang Su et al [6] performed computational work to study the effect on heat transfer on rotating rectangular channel with aspect ratio of 4:1 with pin fins. Some of the important outcomes of the simulations were like the heat transfer at the pin fin surface is 10% to 20% more that at the end surfaces. The enhanced heat transfer is mainly because of the turbulent mixing caused by the flow separation around the pin fins. There is predominant formation of 'Horse Shoe' vortices in the junction between the pin fin and the channel surface. One of the important observations was that the Nusselt number ratio reaches maximum value around the third row and decreases slightly toward the channel exit. It was also observed that the pin fins reduce the rotational effects. H Lacovides and B.E. Launder [7] have reported computational and experimental work carried out on internal cooling passages. The authors stress on the need to perform numerical analysis on rotating blades for pin-fin and impingement cooling techniques as no data is available in open source for such cooling techniques. Je-Chin Han and Lesley M.Wright [8] have provided detailed comprehensive data on different types of cooling techniques like impingement cooling, film cooling, pin-fin and tabulators cooling. The authors provide some of the experimental and numerical data for some of the cooling techniques. Authors point out the lack of data for rotating cooling passages and emphasize the need for more work for rotating cases through numerical techniques as experimental data cannot be generated for practical rotational numbers. Y. M. Zhang et al [9] carried out numerical simulations on rectangular channel with ribs and ribs with grooves on walls to study heat transfer and friction effect. important observations from this simulation are The rib-groove roughened wall enhanced the heat transfer by 3.4 times but with considerable pressure. Similarly for the duct with only rib roughened wall the heat transfer was enhanced by a factor of 2.4 but with similar pressure drop penalty. Recently Smith Eiamsa et al [10] conducted experimental studies to study the thermal characteristics of turbulent rib-grooved channel flow. They reported testing three types of rib-groove geometries. It was observed from their experiment that for three different types of rib-groove geometries the heat transfer was increased by 80%, 60% and 40%. They suggested empirical formulation which produces results within 10% of the measured numbers. Apurba Layek et al [11] also conducted experiments to study the heat transfer and friction characteristics for artificially roughened ducts with compound tabulators. The work was carried out on a duct having transferred rib-groove roughness at Reynolds of 3,000 to 21,000. The tests were carried out on ribs with chamfered edges with different chamfer angles. When compared to smooth duct the rib-grooved surface will yield a maximum of about 3.24 times increase in Nusselt number and 3.78 times increases in friction factor.

Maximum heat transfer (maximum Nusselt number) occurs for pitch ratio of 6, relative groove location of 0.4 and a chamfer angle of 18°.

Based on the review of the earlier work it is clear that there is need to carry out detailed study on the gas turbine blade internal cooling with different cooling techniques especially for rotating conditions. Extensive studies on the blade shaped cooling passages with high performance tabulators for rotating condition are in great demand. There is scope to improve the cooling techniques, with the existing experimental and numerical data. The present work aims at carrying out a detailed investigation on gas turbine blade internal cooling using tabulators consisting of ribs and grooves in rotating conditions.

2. GEOMETRICAL MODELING

The geometric modeling of the ducts has been generated on CATIA V5 modeling tool. Figures 1, 2 and 3 gives the geometrical details for all three configurations of duct.

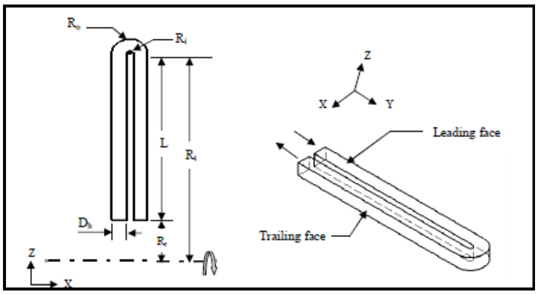

Fig. 1 Geometric model detail of simple U duct

The U shaped square cross section duct dimensions are as follows Duct hydraulic diameter (D_h) = 0.0127 m,

radial position of the duct relative to the axis of rotation (R_r/D_h) = 41.85, Length of the duct (L/D_h) = 14.3, bend inner radius (R_i/D_h) = 0.22, bend outer radius

(R_o/D_h) = 1.44 and R_t/D_h = 56.15

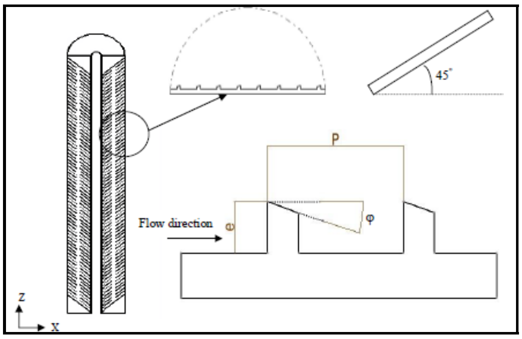

Fig. 2 Geometric and detail view of U duct with 45° ribs

The ribs dimensions are as follows Dh = 0.0127 m, Relative roughness height (e/Dh) = 0.03,Relative roughness pitch (P/e) = 6,Chamfer angle (φ) = 18°

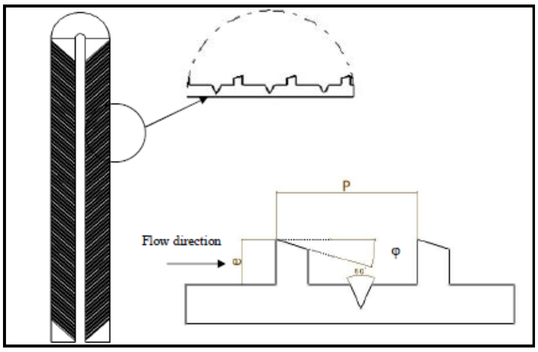

Fig. 3 Geometric and detail view of U duct with 45° ribs and grooves

Dh = 0.0127 m, Relative roughness height (e/Dh) = 0.03, Relative roughness pitch (P/e) = 6, Groove position to pitch ratio (g/P) = 0.4, Chamfer angle (φ) = 18°, Groove angle (θ) = 60°

3. PROBLEM DESCRIPTION

For this work two rotation number Ro = 0 and 0.24 were investigated. Ro = 0 was used to investigate in a non-rotating condition and 0.24 was used to carry out other investigations. The four walls of the duct were maintained at a constant temperature of Tw = 344.85K and the coolant temperature was maintained at 300K. A fully developed flow profile was given at the inlet boundary which was extracted from a 150Dh length straight duct. The inlet density ratio was maintained at 0.13 and the rotational speed was kept at 3132rpm.

4. COMPUTATIONAL MESH

For all geometries structured mesh was generated using Ansys ICEM. For smooth duct three different mesh 0.4million, 0.7milliom and 1.5million mesh were generated for the purpose of grid independent study. For duct with ribs alone all simulations were carried out on 1.5million mesh. For duct with ribs and grooves 3 million mesh was used for all CFD studies. y+ = 1 was maintained for mesh.

5. RESULTS

Initial studies were carried out on smooth duct for non-rotating case. The velocity stream lines and vector plots were compared against the published results in ref [2] for the purpose of validation. Once there was qualitative similarity between the two results, the smooth duct was further subjected to rotation and the Nusselt number comparison was carried out for three different meshes shown in the figure 4. It came out that 0.7million mesh size was adequate to carry out heat transfer analysis for smooth duct. Later the duct was subjected to one complete revolution and the nusselt number normalized against Nusselt number for smooth duct which is given by $N_{us} = 0.021Re^{0.8}\ Pr^{0.5}$ was plotted against the normailsed duct length. Shown in the figure 4.

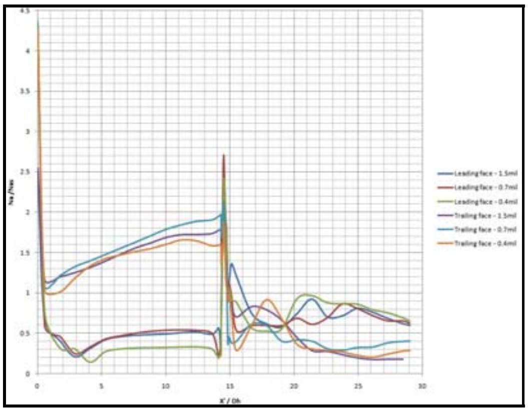

Fig. 4 Nusselt number comparison for different mesh size

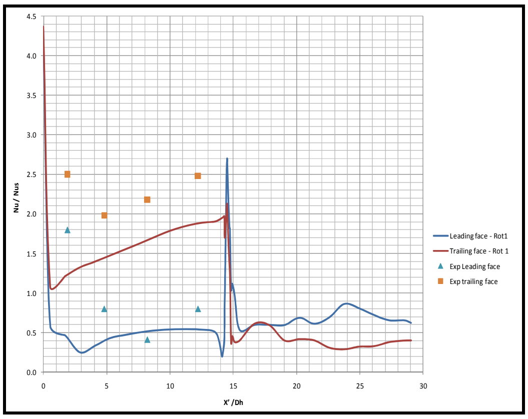

Fig. 5 Nusselt number comparison for one revolution

After extracting the Nusselt number for smooth duct and comparing with the experimental results the rib turbulators were added to smooth duct and the simulations with similar boundary conditions were carried out. Once again the duct was subjected to one complete revolution and the results were compared with the smooth duct Nusselt number.

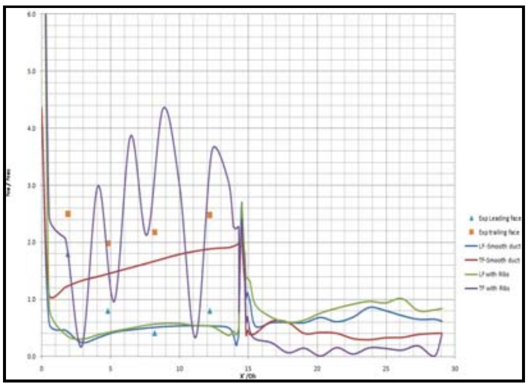

Fig. 6 Nusselt number comparison for smooth and ribbed duct

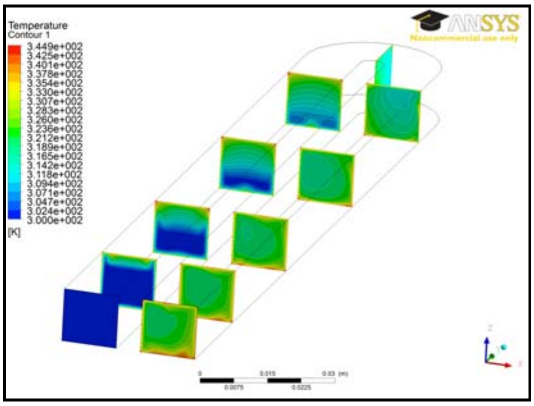

Fig. 7 Temperature contour for smooth duct

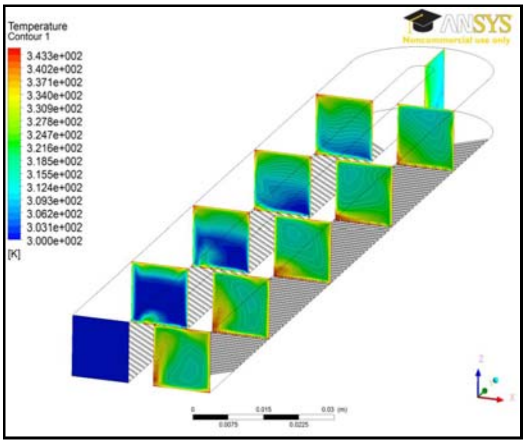

Fig. 8 Temperature contour for ribbed duct

Having done simulations on ribbed duct, similar simulations were conducted on duct with ribs and grooves. But the simulation for this geometry was carried out for 180° revolution only. The results at that angle were compared with other two configurations.

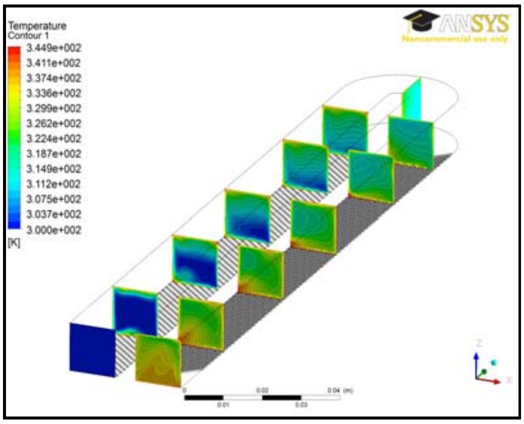

Fig. 9 Temperature contour for rib-grooved duct

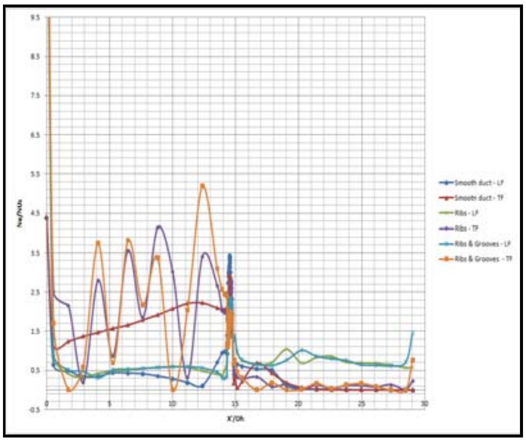

Fig. 10 Nusselt number comparison for three configurations

When the simple U duct is subjected to rotation , the results from one revolution when compared with the experimental results, Nusselt number trends shows that the of wall heat transfer is going toward the experimental results which is clear indication that on further continuing the simulations, the CFD results would match the experimental results within the acceptable error band.The velocity contours show that the rotational effects are well captured. It can be seen that as we move away from the inlet, the fluid is compressed more towards the trailing face due to the rotational force. In section 4 the validation results of duct with ribs and grooves is very close the experimental results .This gives a clear indication that numerical tool is capable of capturing complex The disturbance tends to move toward the outer wall in the inlet side due to the rib orientation which creates low velocity secondary flow.

Similarly Figure 9 shows the temperature contour at different planes. It can be seen from the contours that there is drift in the heat transfer location along the rib orientation. This results in non uniform and enhanced heat transfer from trailing face to coolant.

Figure 7 shows large variations in Nusselt number at the trailing face. This is a clear indication of the enhancement of heat transfer. Figure 7 shows large variations in Nusselt number at the trailing face. This is a clear indication of the enhancement of heat transfer. On figure 8 shows notable difference in velocity contours which is due to addition of ribs. Flows and also the experimental conditions are very well replicated. to the smooth duct. The velocity contour shows a disturbed flow at the trailing face. Further continuing the rotations the effect of ribs on leading face can also be seen and assisted by the rotating forces in the other side of the duct. The Nusselt number comparison in figure 10 shows the addition of grooves had contributed to increase in wall heat transfer. The increase in peak values of Nusselt number shows that there is enhancement in heat transfer caused due to addition secondary flows and recirculation zones

7. CONCLUSIONS

The results generated for smooth duct with rotation is in good agreement with the experimental trends and signifies that CFD results are becoming more and more reliable and can be used real time design of gas turbine blades. Addition of turbulators will definitely increase the wall heat transfer to a large extent and this has been confirmed from the plots. Buoyancy cannot be neglected as it plays vital role in such convective heat transfer problems. Since the results published are only for one revolution and half revolution in ribbed and rib – grooved geometry respectively, further investigation in terms of simulations is required for complete understanding of wall heat transfer. Nevertheless, the present results for the turbulators show that there is significant enhancement in heat transfer from the heated wall to coolant when compared with simple U tube.

8. REFERENCES

[1] J H Wagner, B V Johnson and T J Hajek – '*Heat transfer in Rotating Passgae With Smooth Walls and Radial Outward Flow'*- Journals of Turbomachinery, Vol 113, No,1,1991, pp 42-51.

[2] MarkA.Stephens and TomI-P.Shihf -*'Flow and Heat Transfer in a Smooth U-Duct with and without Rotation*' Carnegie Mellon University, Pittsburgh, Pennsylvania 15213-3890, Journal Of Propulsion And Power Vol. 15, No. 2, March-April 1999.

[3] Je-Chin Han and Lesley M.Wright-'*Enhanced Internal Cooling of Turbine Blades and Vanes', Turbine Heat Transfer Laboratory*, Department of Mechanical Engineering, Texas A&M University College Station, Texas USA.

[4] H lacovides and B.E. Launder - '*Internal Blade cooling The Cinderella of computational and experimental fluid dynamics research in gas turbine*', Proceedings of the Institution of Mechanical Engineers, Part A Journal of Power and Energy 2007 221:265.

[5] Guoguang Su, Hamn – Ching Chen, Je- Chin Han, James D. Heidmann – '*Computation of flow and Heat Transfer in rotating two-pass rectangular channel(AR = 1:1, 1:2,and 1:4)with smooth wall by a Reynolds stress turbulence model'* , Turbine Heat Transfer Laboratory, Department of Mechanical Engineering, Texas A&M University college Station, Texas, International Journal of Heat Transfer 47 (2004) 5665-5683.

[6] Guoguang Su, Hamn – Ching Chen, Je- Chin Han – '*Computation of Flow and Heat Transfer in Rotating Rectangular Channels* (AR = 4:1) With Pin-Fins by a Reynolds Stress Turbulence Model', Journals of Heat Transfer, June 2007, Vol 129, pp 685-696.

[7] Y M Zhang, W Z Gu, J C Han –' *Heat Transfer and Friction in Rectangular Channel with Ribbed or Ribbed-Grooved Walls*', Journals of Heat Transfer, February 1994, Vol 116, pp 58-65.

[8] Akira Murata, Sadanari Maochizuki - '*Effect of centrifugal buoyancy on turbulent heat transfer in an orthogonally rotating square duct with transverse*

or angled rib turbulators', International Journals of Heat and Mass Transfer 44(2001), 2739 – 2750.

[9] T M Liou, S W Chang, J H Hung, S F Chio- *High rotational number heat transfer of a 450 rib roughened rectangular duct with two channel orientations'*, International Journals of Heat Transfer 50 (2007) 4063 – 4078. ‘

[10] Shyy Woie Chang, Tong-Miin Liou, Yu Po –‘ *Coriolis and rotating buoyancy effect on detailed heat transfer distributions in a two-pass square channel roughened by 450 ribs at high rotation numbers',* International Journals of Heat Transfer 53 (2010)1349-1363.

[11] Apurba Layek, J S Saini, S C Solanki – ‘*Heat transfer and friction characteristics for artificially roughened duct with compound turbulators'*, International Journals of Heat Transfer 50 (2007) 4845 – 4854.

[12] Smith Eiamsa-ard, Pongjet Promvonge- ‘*Thermal Characteristics of Turbulent rib- grooved channel flows'*, International Communications in Heat and Mass Transfer 36 (2009) 705-711

[13] Mohammad Al-Qahtani, Yong-Jun Jang, Hamn-Ching Chen and Je-Chin Han- ‘*Prediction of Heat Transfer in Rotating Two Pass Rectangular Channels With 45-deg Rib Turbulators*', Journals of Turbomachinery, April 2002, Vol.124 pp1-9

[14] Mohammad Al-Qahtani, Yong-Jun Jang, Hamn-Ching Chen and Je-Chin Han – ‘*Flow and heat transfer in Rotating two pass rectangular channel (AR=2) by Reynolds stress turbulence model',* International Journal of Heat and Mass Transfer 45 (2002) 1823 – 1838.