Thermal boundary layer flow effectiveness enhancement of gas turbine blade internal cooling using CFD

[1]Siddesh S, [2]Prof. K S Madhu,
[1]PG Scholar, [2]Assistant Professor
[1]Thermal Power Engineering, Department Of Mechanical Engineering
[1]Rajarajeshwari College Of Engineering, Bengaluru, India

Abstract : A Computational Fluid Dynamics study will be carried out to obtain heat transfer data for a single pass and two-pass rectangular channel (aspect ratio 2:1) with smooth and ribbed surfaces for two channel orientations. The V-shaped ribs are placed on the leading and trailing surfaces of 4 different arrangements of 45 deg V-shaped ribs are studied. The Reynolds numbers for study are selected as 5000, 15000 and 40000 and the rib height to hydraulic diameter ratio is 0.092, the rib pitch-to-height ratio is 8. The CFD studies are carried out for the parallel 35 deg V-shaped rib arrangements, inverted 35 deg V-shaped rib arrangements, parallel inlet and inverted outlet and inverted inlet and parallel outlet to determine the better heat transfer and augmentation of the flow.

***Index Terms* – Heat Transfer, Gas Turbine Blade, CFD Analysis**

I. INTRODUCTION

GAS TURBINE THEORY

Gas turbine engine extracts power from the flow of combustion gases. Energy is extracted in the form of shaft power and thrust. The gas turbines are described thermodynamically by the Brayton cycle as shown in figure 1.2, in which air is compressed isentrophically, combustion occurs at constant pressure and expansion over the turbine occurs isentrophically back to the starting pressure.

THREE SPOOL TURBOFAN ENGINE

A three spool engine is one that has three sets of compressors before the combustor and three sets of turbines behind it. A spool is made up of a compressor and a corresponding turbine used to extract the power from the exhaust gasses to turn the compressor.
Each spool is given a name. N1, N2, and N3. N1 is the large fan section in front of the engine. N2 is the intermediate pressure compressor section. And N3 is the high pressure compressor section. Each section of the compressor wants to rotate at its own speed, and if allowed to do so as in a triple spool engine, it is able to operate more efficiently. It can turn at its optimum speed, all modern engine have 2 sets of compressors (HP and IP) and a fan section providing a vast majority of the thrust.

EXTERNAL AIR COOLING

Film Cooling

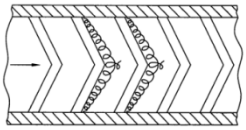
In film cooling there is an opportunity for direct interaction between coolant and mass flow. The secondary air bleed out from the compressor routed to internal passages of blade or vane is come out through single row or multiple rows of holes called film cooling holes located around the vane or blade. This secondary air forms a thin cooled film over the surface that protects the blade material from hot gas stream especially near the leading edge where high temperature is encountered. The flow coming out from the film cooling holes will not alter the in viscid characteristics of the flow but it will modify the turbulence, transition and boundary layer characteristics. The major complexity associated with film cooling is that the flow come out from the film cooling holes interact with main flow. This causes the separation of upstream boundary layer. If multiple rows of holes are used, the boundary layer never reaches a state of equilibrium.

Rib Turbulators

The location of ribs in the smooth surface interrupts the hydrodynamic and thermal boundary layers. Downstream of each rib the flow separates, recirculates and impinges on the channel surfaces, the effect of impinges are the main reasons for heat transfer enhancement in such test section. Due to the existence of ribs effective heat transfer surface increases, Ribs are manmade protrusions which are placed in a controlled way along the walls. The rib induces a separation in the flow and hence causes an increase in the frictional loss. The enhancement of the heat transfer has thus a drawback in the increased pressure drop, which sometimes can be several times higher than for a smooth channel. Phenomenon of Heat Transfer Enhancement using Ribs:

a) Laminar Sub-layer breakage.
b) Increasing local near wall turbulence.
c) Decreasing the thermal resistance.

In this thesis V Shaped Rib Turbulators at 45 degrees orientation are studied and its arrangement in a single pass



V-Shaped Rib Turbulators

II. LITERATURE REVIEW

A number of traditional cooling concepts are used in various combinations to adequately cool the turbine vanes and blades. Gas turbine heat transfer and cooling technology by Han, J. C., et al. Provides a detailed description of turbine blade heat transfer and cooling technology. The author compiled a comprehensive review of gas turbine cooling technology including techniques to enhance the heat transfer in internal cooling passages. The book also includes a numerous studies that have been conducted over the years on a wide range of rib configurations in various cooling channels using many experimental techniques.

Early studies investigated cooling channels with orthogonal ribs. Han, J. C. has performed an experimental study on the ribbed channels with orthogonal square ribs. Three different ribbed channels with different aspect ratios were considered. Also the rib spacing and Reynolds number were taken into account. The work compiled a detailed study on the effect of rib spacing on the heat transfer performance of the ribbed channels. The author has also provided us with a heat transfer and friction correlation. This paper provides us with the experimental results which were compared with the numerical study obtained through the numerical analysis.
Iacovides, H. [15] Performed the computation work on the ribbed rectangular passages. In his study both stationary and rotating rectangular passages were considered. The computation work was based on the orthogonal square ribs. The work mainly concentrated on the effect of turbulence model on the channel performance. A differential stress model was developed and it proved to yield better results than the standard k-e model.

Iacovides, H. and Raisee, M. [16] Performed a computational study on a rib roughened passage using low Reynolds number turbulence model. A standard k-e and standard k-w turbulence models were used. The periodic boundary condition was employed and the parameters such as Nusselt number are obtained. The differential stress model was also developed. The DSM model provided an improved heat transfer predictions after flow reattachment and over the ribs. However the model failed to predict the effect of Reynolds number. Lin, Y. L., et.al [20] performed a numerical study of flow and heat transfer in a duct with 45 degree angled ribs. Both rotating and non-rotating duct was considered. The analysis was performed on a three dimensional flow. With a 3D analysis, the secondary flow was predicted. These results also show how the nature of the fluid flow affects surface heat transfer. Also the secondary flow has pronounced effects on heat transfer.

Han and Chen provided a summary of experimental and numerical research into turbine blade internal cooling passages with rib turbulators. They highlighted the ability of Reynolds averaged Navier-Stokes (RANS) models to resolve the complex three dimensional flow physics that exist in rotating turbine blade internal coolant passages.

III. PROBLEM DESCRIPTION & METHODOLOGY

.

METHODOLOGY

In this study, simulation approach using commercial CFD software is used. 3-D geometrical. The flow and heat transfer pertaining arrangement for cooling enhancement is carried out using CFD simulation software Fluent, Different type of rib configuration for different flow rates. The theory behind the CFD and the relevant Partial Differential Equations are described below.

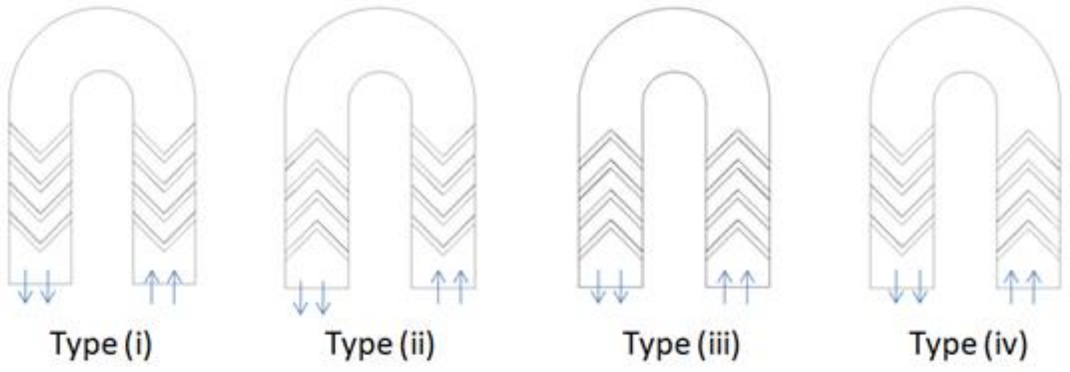
MATHEMATICAL BACKGROUND AND CFD

Computational Fluid Dynamics, usually and from here on abbreviated to CFD, is a numerical method to compute the dynamics of a fluid. In brief, it is implemented by dividing a computational domain into small cells where the flow is modeled and flow equations are solved. Since almost all fluid flows are turbulent, different CFD methods are used in order to simulate turbulence. These can be divided in different categories where some of them are; Turbulence models for Reynolds-Averaged Navier-Stokes (RANS) equations, Large Eddy Simulation (LES), Direct Numerical Simulation (DNS) and Lattice Boltzmann method (LBM). The LBM differs from the other methods by simulating the movements of particles and aims over the hydro dynamics of the Navier-Stokes equation. RANS on the other hand, uses the Navier Stokes equations as a starting point and aims at solving them. DNS does not use a turbulence model; it computes all the turbulent velocity fluctuations and therefore demands both small time steps and cells that require substantial computer resources. LES focus on the large eddies in the flow and requires quite large computer resources. However, the most common way to simulate turbulence is to calculate the time averaged properties of the flow, such as mean pressure, mean velocity and so on, which in most cases give sufficient information about the flow. This method has a modest computer demand and is conducted with RANS-models.

OBJECTIVES OF THE WORK

a) To study the channel aspect ratio effect (W/H = 2) on the heat transfer for the ribbed channels at different Reynolds number. The Reynolds number is in range of 5000, 15000 and 40,000.

b) To study the different arrangements of V-Shaped Ribbed turbulators in a two pass rectangular duct with 180 degree U bend. And different arrangements are as show in Figure 3.1.



Arrangements of V-Shaped Ribs in Rectangular Channel.

Type (i): First Pass: - V-Shaped Rib Arrangement
Second Pass: - Inverted V-Shaped Rib Arrangement

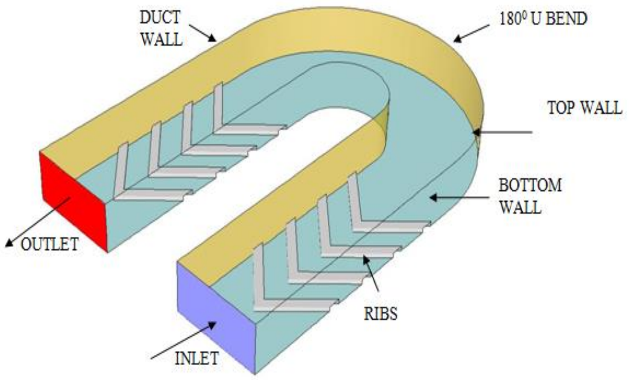
Type (ii): First Pass: - Inverted V-Shaped Rib Arrangement
Second Pass: - Inverted V-Shaped Rib Arrangement

Type (iii): First Pass: - Inverted V-Shaped Rib Arrangement
Second Pass: - V-Shaped Rib Arrangement

Type (iv): First Pass: - V-Shaped Rib Arrangement
Second Pass: - V-Shaped Rib Arrangement

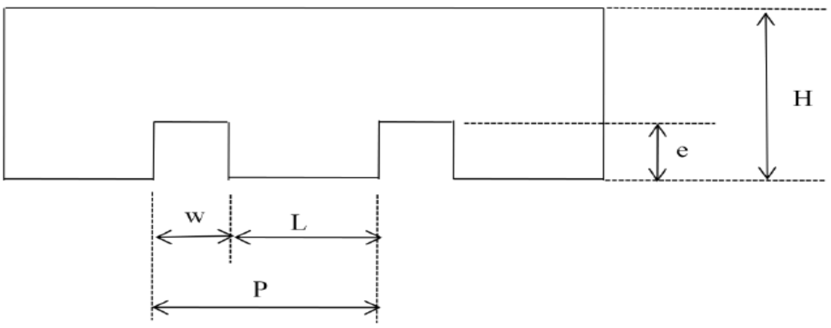
IV. CAD AND GRID DETAILS

CAD Geometry



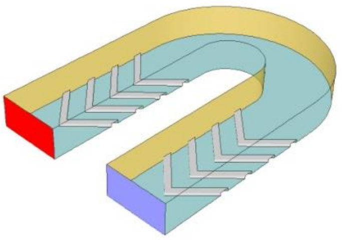
CAD Model of Rectangular Channel and V-Shaped Ribs

V-Shaped Ribs geometry is simplified by considering the eight ribs on the bottom surface of duct. Inlet and Outlet are placed at a distance equal to pitch.

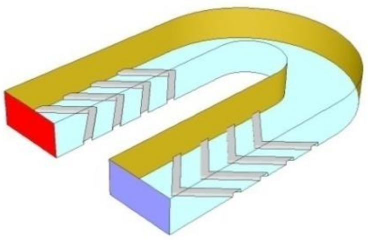


Parameters	Symbol	Range of Values
Channel Aspect Ratio	W/H	2
Rib height to Hydraulic diameter ratio	e/D	0.094
Rib Width to Rib height ratio	w/e	2
Rib Spacing	P/e	10
Reynolds Number	Re	5000,15000,40000

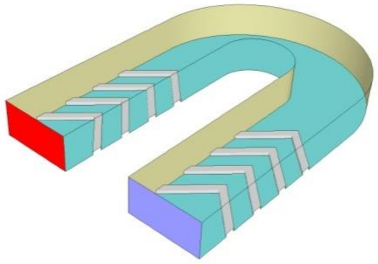
W = Width of Rectangular Channel = 25.4 mm, Height of Rectangular Channel = 12.7 mm.,D = Hydraulic Diameter of the channel = 16.933 mm. Rib Height (e) = 1.1938 mm,w = Width of rib = 2.3876 mm



Type (i)



Type (ii) & Type (iv)-reverse flow



Type (iii)

Type (i) shows the Rib Arrangement in straight manner hitting v notch and inverted in other pass
Type (ii) shows the Rib Arrangement Straight manner hitting v notch and following same
Type (iii) the Rib Arrangement hitting two edges of v and following the same (flow in reverse fashion)
Type (iv) shows the Rib Arrangement hitting two edges of v and following inverted hitting v notch.

Flow Physics and Boundary Condition

Boundary Condition Set up is followed as shown in Table 3.2. Velocity is calculated for different Reynolds number accordingly and defined as inlet. Ribs are of Copper material which is coupled and all other wall is considered as Aluminium. Fluid properties are discussed in chapter 3 and incompressible ideal gas is considered as fluid material. K-Epsilon Realizable turbulence model is been used for the solution of the problem and First order upwind scheme is chosen due to convergence issues with other models. Rib Wall temperature is known and is about 360 K. And air inlet temperature is 300K, Operating Pressure is 101325 Pa.

Boundary Conditions for CFD Analysis

	Boundary Condition Type	Parameter Defined
INLET	Velocity Inlet	Velocity in m/s Inlet Temperature
OUTLET	Pressure Outlet	0
Ribs	Wall	Copper Wall with Heat Flux - Coupled
Bottom Wall	Wall	Copper Wall with Heat Flux - Coupled
Top Wall	Wall	Aluminum
Duct Wall	Wall	Aluminum
Turbulence Model	-	K-Epsilon - Standard
Upwind Scheme	-	First Order Upwind

MATERIALS AND MODELING

Material chosen for square channel is aluminum and the fluid is considered to be incompressible air. Rib materials selected as same of channel. In this work the inlet velocity is varied as 3.79 m/s, 11.34 m/s, 30.25 m/s respectively as the Reynolds number varies as 5000, 15000, 40000.Inlet temperature of the fluid is assumed to be room temperature which is 300K

Fluid Properties

Properties of Air at Atmospheric Conditions (101325Pa)	Temperature at 300K
Density (ρ) kg/m^3	1.1526
Specific Heat Cp(J/kg-K)	1007
Thermal Conductivity k (W/m-K)	0.026102
Dynamic Viscosity μ (kg/m-s)	1.8858e-05
Kinematic Viscosity γ (m^2/s)	1.6362e-05
Molecular Weight (kg/kg mol)	28.97

V. RESULTS AND DISCUSSION

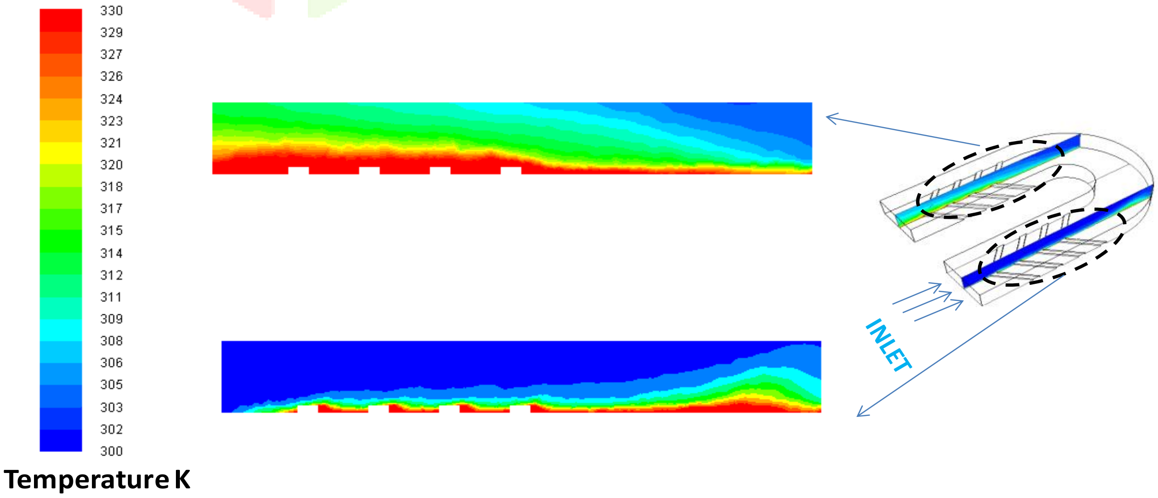
CFD ANALYSIS

Computational fluid dynamic study of the system starts with building desired geometry and mesh for modeling the domain. Generally, geometry is simplified for the CFD studies. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modeling starts with defining the boundary and initial conditions for the domain and leads to modeling the entire system domain. Finally, it is followed by the analysis of the results.

Details of Case Study

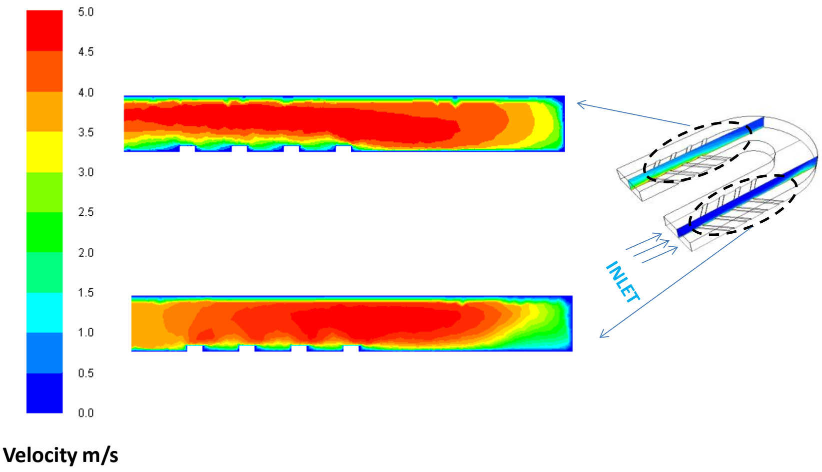
	Model Type	Reynolds Number Re	Velocity inlet m/s	Inlet Temperature K	Ribs Wall Temperature K
Case 1	Model 1	5000	3.79	300	360
Case 2		15000	11.34		
Case 3		40000	30.25		
Case 4	Model 2	5000	3.79		
Case 5		15000	11.34		
Case 6		40000	30.25		
Case 7	Model 3	5000	3.79		
Case 8		15000	11.34		
Case 9		40000	30.25		
Case 10	Model 4	5000	3.79		
Case 11		15000	11.34		
Case 12		40000	30.25		

Case 1: Total temperature and Velocity plots



Contour Plots of Total Temperature on Meridional Plane

For case 1 , the inlet velocity is 3.79 m/s and the temperature rise is from inlet to outlet is shown in figure 4.1 , the temperature gradient raise is quite high at V notch due to recirculation in the domain. The velocity attains peak of 5 m/s at middle of the flow, the obstructions of V notch makes the velocity to raise as shown in figure 4.2.

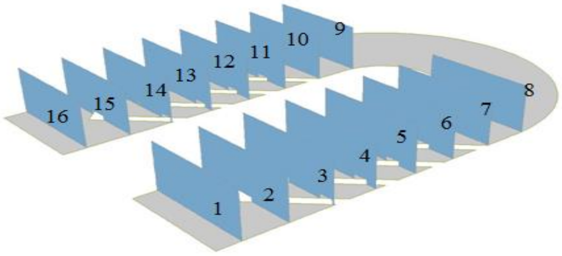


Contour Plots of Velocity magnitude on Meridional Plane

Temperature at	Case 1 K	Case 2 K	Case 3 K
Section 1	300.6309	300.2812	300.5443
Section 2	301.7564	300.7645	300.7733
Section 3	302.8271	301.3734	301.0853
Section 4	303.7242	301.9522	301.3879
Section 5	304.5943	302.5366	301.6932
Section 6	305.4893	303.1453	302.0085
Section 7	306.2992	303.7087	302.3163
Section 8	306.9401	304.1532	302.5468
Section 9	311.437	306.315	303.8076
Section 10	312.1456	306.6963	304.0263
Section 11	312.9001	307.2036	304.2951
Section 12	313.5794	307.7045	304.5779
Section 13	314.2906	308.2283	304.8795
Section 14	314.9515	308.7416	305.1667
Section 15	315.5825	309.2221	305.46
Section 16	316.0541	309.537	305.6391

Mass weighted Temperature Values

Mass Weighted Average Temperature at different Locations

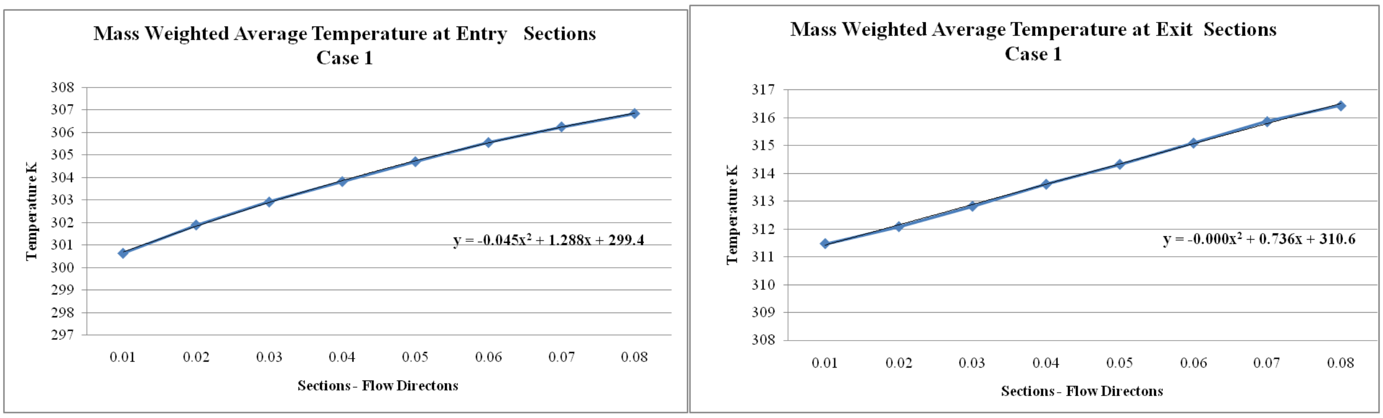


Sections in Flow Direction in Rectangular Channel

From Fluent post processing, the temperatures at different sections are extracted and tabulated by using this values further T_{bulk} calcualtions and HTC calculations are carried out in further section

T_{bulk} extraction using curve fit method

Using the below polynomial equation we can find the bulk temperature at required locations in the rectangular channel for Entry side and Exit side separately



Curve fitting of Mass Weighted Average of Temperature – Case1

In case1 as the Reynolds no. is 5000 and velocity is 3.79m/s the exit temperature is 316.0541K. In case2 as the Reynolds no. is 15000 and velocity is 11.34m/s the exit temperature is 309.537K. In case3 as the Reynolds no. is 40000 and velocity is 30.25m/s the exit temperature is 305.6391K .In which it is has high cooling effect in section 16 as compared to other cases

Htc and Nusselt number calculation:

a) Heat transfer coefficient is calculated for Rib entry side, wall entry side , rib exit side and wall exit side using following equation

$$\text{htc} = (\text{Wall Heat Flux})/(T_{wall} - T_{bulk}) \qquad (4.3)$$

b) Area weighted average htc was calculated using all the four htc's of the model

c) Nusselt Number Calculation

$$\text{Nu} = (\text{htc} \times \text{Dh} - \text{Hydraulic Diameter})/\ (\text{k}) \qquad (4.4)$$

Dh = Hydraulic Diameter m

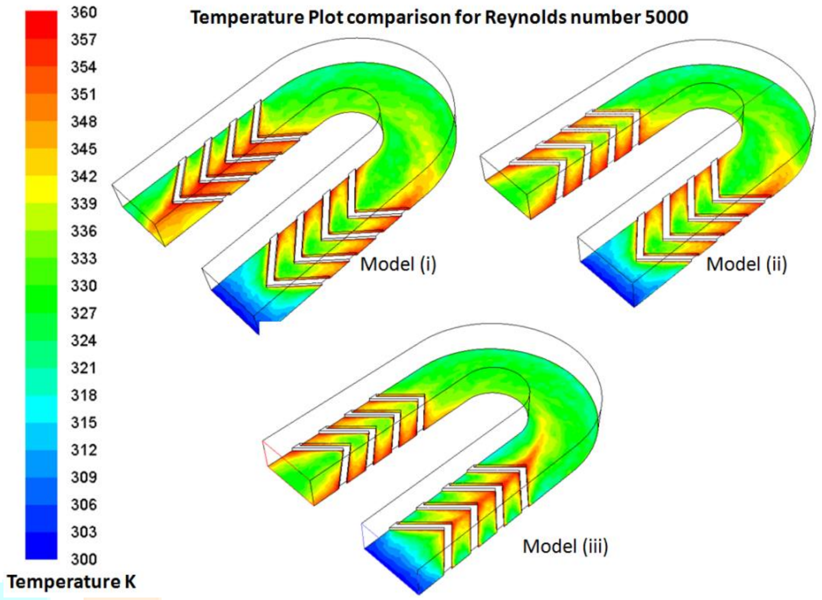
k = Thermal Conductivity of air W/m-K

Similar strategy has been used for case 2 and case 3, where mass flow rates were different from the first case and calculated values

Results and Nusselt Number – Model Type (i)

Model Type (i)	Case 1 Reynolds Number 5000	Case 2 Reynolds Number 15000	Case 3 Reynolds Number 40000
Mass Flow rate At Inlet	0.0014	0.0043	0.0116
Mass Flow At Outlet	0.0014	0.0043	0.0116
Mass Imbalance in %	0.0001	0.0001	0.0001
Heat Transfer Rate through Ribs (W)	5.504	10.492	15.211
Heat Transfer Rate through Wall (W)	18.08	31.21	46.181
Energy at Inlet	451.54	1350.7	3607.9
Energy at Outlet	475.13	1392.4	3669.4
Energy Imbalance in %	0.0007	0.0009	0.0026
htc at entry side rib (W/m^2-K)	82.879	137.41	194.2
htc at exit side rib (W/m^2-K)	80.27	133.41	197.63
htc at entry side bottom wall (W/m^2-K)	66.649	114.53	175.92
htc at exit side bottom wall (W/m^2-K)	67.271	114.48	182.52
Avg Htc of heating side rib +wall (W/m^2-K)	72.33	122.52	185.77
Nusselt Number	50.61	85.73	129.98

Comparison Plot: Surface Temperature



Surface Temperature comparison for Reynolds number 5000

From the Surface temperature rise is high in Model-1 compared to Model-2 and 3 the reason for this temperature distribution is due to inverted V notch in Model-1 at exit side and other models have reverse notch.

Comparison Plot: Total Temperature

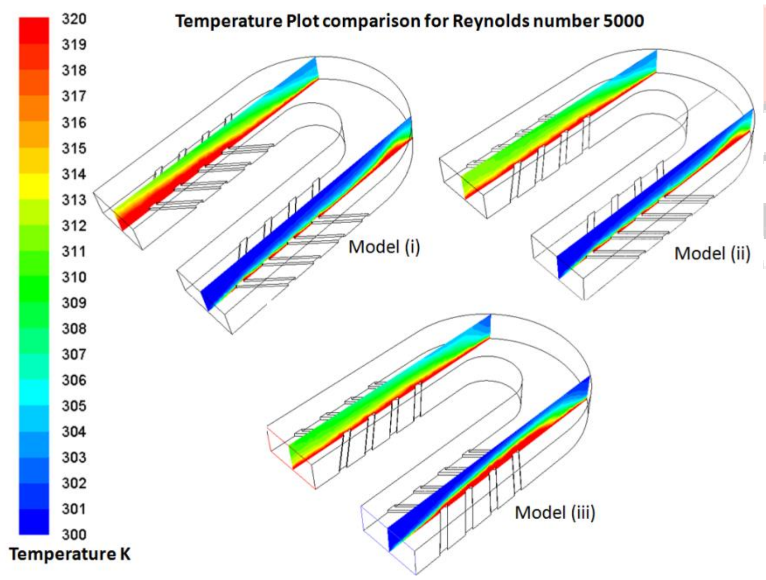
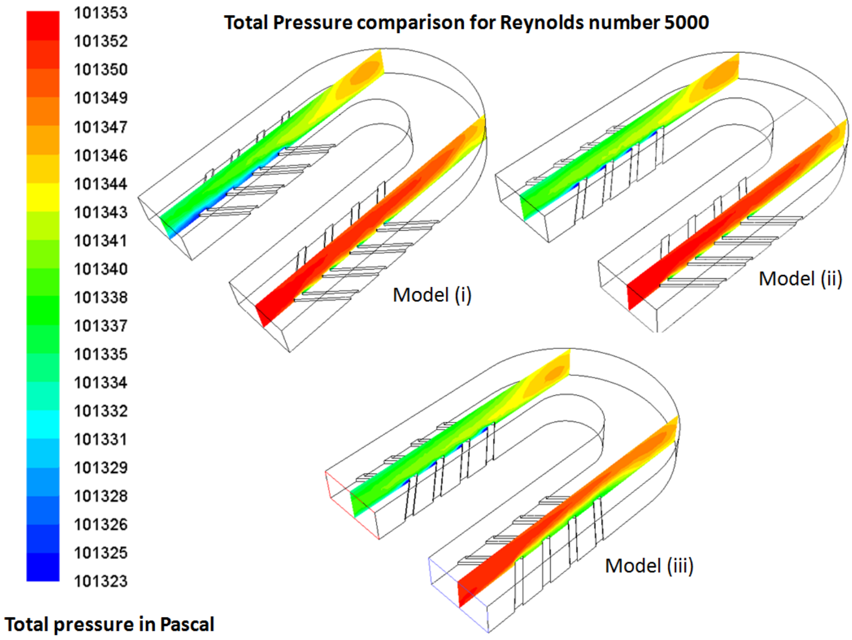


Figure 4.15: Temperature Plot comparison for Reynolds number 5000

From the above figure 4.15, the temperature rise gradient is low across cross section in Model-1 compared to Model-2 and 3 the reason for this temperature distribution is due to inverted V notch in Model-1 at exit side.

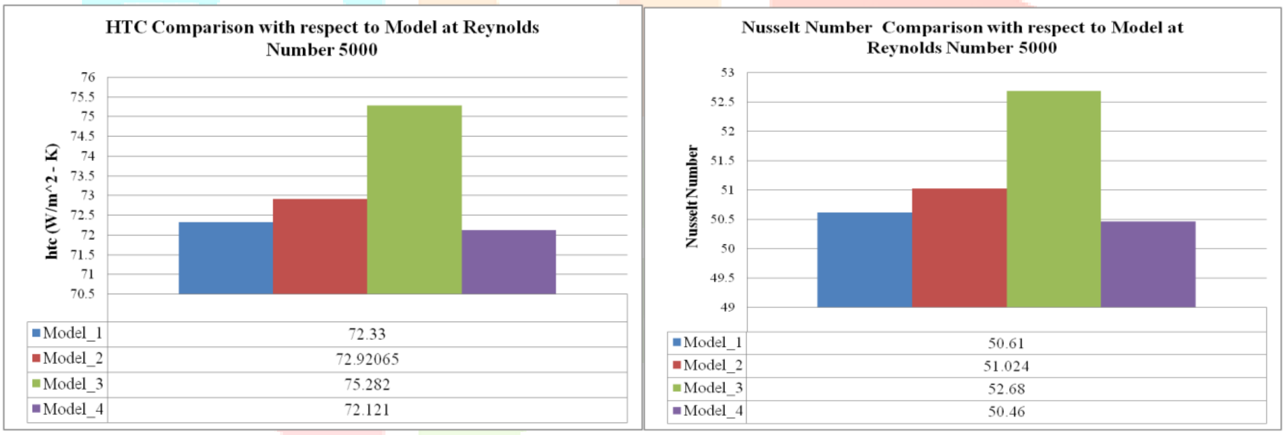
In case of Model-2 and 3 have smooth gradient throughout flow at exit side with V leads to higher surface htc.

Comparison Plot: Total Pressure



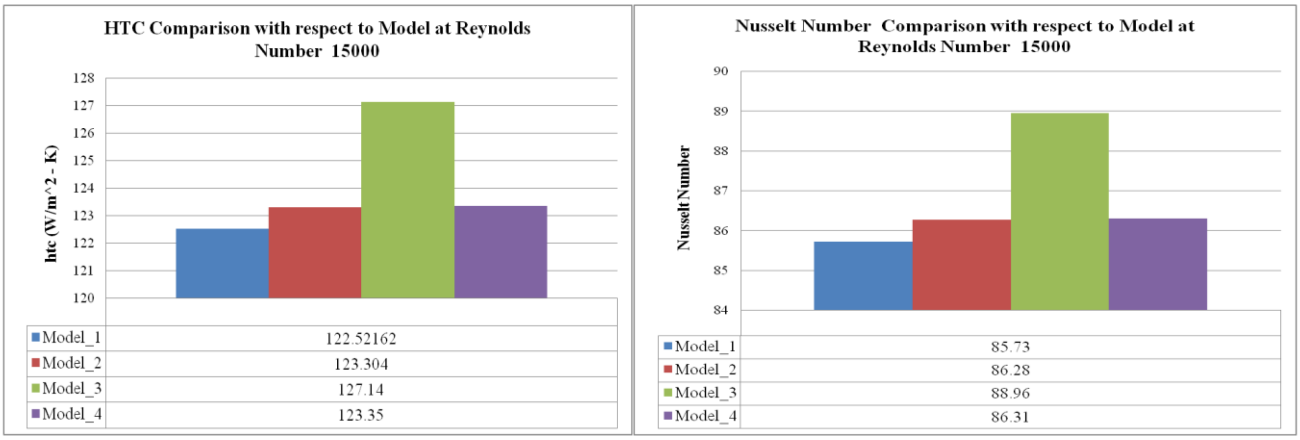
Total Pressure comparison for Reynolds number 5000

From the above figure 4.16 the Total Pressure drop across all the models with Reynolds number of 5000 is almost same, by changing the Rib positioning we can observe there is no pressure losses in the models. Similar trend can be seen for Reynolds number 15000 and Reynolds number 40000 for pressure drop and htc values.



HTC and Nusselt Number Comparison for Models with Reynolds number 5000

From the above Figure it is evident that the Model 3 predicts higher Heat Transfer Coefficient for Reynolds Number 5000. So the Rib Arrangements in model 3 is suggested for lower Reynolds Number 5000



HTC and Nusselt Number Comparison for Models with Reynolds number 15000

From the above Fig. it is evident that the Model 3 predicts higher Heat Transfer Coefficient for Reynolds Number 15000. So the Rib Arrangements in model 3 is suggested for Reynolds Number 15000

VI. CONCLUSION

From results, we can determine

- Model 3 with Inverted Rib Arrangement is best suited for flows with comparatively Low Reynolds Number 5000.
- Model 3 Proves to perform with same effect, and it is recommended for Reynolds Number around 15000
- Model 3 and 4 with best suitable for flows with high Reynolds Number around 40000
- Model 1 and 2 Performance is not efficient in comparison with other two models
- Henceforth it can be concluded that for Reynolds Number of range 5000 – 15000 Model 3 yields higher Heat Transfer Coefficient and High Nusselt Number which concludes as best suitable arrangement. Whereas for Reynolds Number of 40000 Model 3 and 4 performs better in comparison with Model 1 and 2.
- Entry side with inverted V ribs are yields higher htc from the CFD analysis. The model 3 and 4 with this type of arrangements resulted in achieving higher htc.

VII. FUTURE WORK

CFD can be a useful tool for studying different configurations with rib arrangements. It gives intricate details in flow field, near wall flow behavior. Thus providing a scope for W shaped rib arrangement studies and exploring with wedge, triangular, blunt type ribs arrangement with different angle of orientation 45^0, 65^0 ,90^0 and with varying flow Reynolds numbers and velocity parameters

REFERENCES

1. An experimental investigation of forced convection heat transfer in channels with rib Turbulators by means of liquid crystal thermography by Diego Cavallero, Giovanni Tanda, University of Genova, ITALY, 1999.
2. Agarwal, P., “Heat/ Mass Transfer in Smooth and Ribbed Rectangular Serpentine Passages of Different Aspect Ratios and Orientation”, Master’s Thesis, Department of Mechanical Engineering, Louisiana State University, USA, 2001.
3. Bonhoff, B.,et.al, “Experimental and Numerical Study of Developed Flow and Heat Transfer in Coolant Channels with 45 degree Ribs”, Journal of Heat and Fluid Flow , Vol. 20, pp. 311 – 319, 1999.
4. Bredberg, J., “On the Wall Boundary Condition for Turbulence Models”, Internal Report 00/4, Chalmers University, Sweden
5. Bredberg, J., “Turbulence Modeling for Internal Cooling of Gas Turbine Blades”, Doctoral Thesis, Department of Thermo and Fluid Dynamics, Chalmers University, Sweden 2002.
6. Cengel, Y. A., “Heat and Mass Transfer: A Practical Approach”, 2002, McGraw-Hill, 2nd Edition, pp. 451-482
7. Farrell, B. F., “Optimal excitation of perturbations in viscous shear flow”, American Institute of Physics, pp. 2093-2102, April 1988
8. Diego Cavallero, Giovanni Tanda "An Experimental Investigation of Forced Convection Heat transfer in channels with rib turbulators by means of Liquid Crystal Thermography" DITEC, Università degli Studi di Genova via all’Opera Pia 15a, I-16145 Genova, ITALY

9. Graham, A., Sewall, E., and Thole, K.A., “Flow Measurements in a Ribbed Channel Relevant to Internal Turbine Blade Cooling”, ASME Turbo Expo, Power for Land, Sea and Air, Vienna, June 14-17, 2004.
10. Han, J. C., Dutta, S., and Ekkad, S., “Gas Turbine Heat Transfer and Cooling Technology”, 2001, Rout ledge Press, 1st Edition, pp. 287-370, 2001.

.

.