iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 8 **Issue:** VIII **Month of publication:** August 2020

DOI: https://doi.org/10.22214/ijraset.2020.30852

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

Forced Convection Heat Transfer in Channels with Rib Turbulators using CFD

Gudepu Sridevi[1], Dr. V. Sridhar Patnaik[2], K. Ravindra[3], U. S. Ramakanth[4]
[1, 2, 3]M. Tech Student, Department of Mechanical Engineering, #Visakha Technical Campus, Vizag, AP
[4]Associate Professor, Lendi Institute of Engineering and Technology, Vizianagaram, AP

***Abstract:** To achieve high thermal efficiency in a gas turbine engine, the temperature of the turbine inlet gas must be increased. However, the penalty is a high heat load, which affects the durability of the turbine components. Therefore, improved cooling technologies such as film cooling and internal cooling are applied to the turbine blades. Inner cooling is achieved by circulating low enthalpy air in multiple flow channels inside the blade structure. To increase heat transfer from internal cooling, internal surfaces are generally hardened by angular ribs to move the boundary layer and increase turbulence.*

Computational fluid dynamics study was performed to obtain heat transfer data for a two-pass rectangular channel with two passes (2: 1 aspect ratio) with smooth and sheared surfaces for two-channel directions. V-shaped ribs are placed on the front and back surfaces. Seven different V-shaped rib arrangements with a 45-degree angle were studied. Reynolds numbers for the study are determined as 5000 and 40,000 and the aspect ratio of the rib to the hydraulic diameter is 0.094, the aspect ratio is 10, and the density ratio the refrigerant of the inlet wall is kept around 0.115 per test. CFD studies are conducted on arrays of parallelograms of V-shaped by 45 ° and produce a better increase in heat transfer than inverted ribs of V-shaped by 45 ° and crossed ribs of V-shaped by 45 ° to determine the best heat transfer and increase of flow.

CFD is used to predict the bulk temperature inside the channel channel through which the heat transfer coefficient is reached. Comparing the Nusselt number between the different matrices gives an idea of the improved heat transfer.

***Keywords:** Turbine Engine, ribs of V-shaped, Reynolds numbers, flow channels, CFD*

I. INTRODUCTION

The gas turbine engine as draws energy from the flue gas stream. Energy is drawn in the form of shaft and thrust force. Gas turbines are thermodynamically described by the Brighton Cycle, where air is compressed in an equal direction, combustion occurs at constant pressure, and expansion over the turbine occurs evenly again at initial pressure. Gas turbines play a vital role in today's world, and with increased energy demand, thermal efficiency and engine power production must be improved. One way to increase both the resulting energy and thermal efficiency is to increase the temperature of the hot gas inlet through the internal channels. The three major components of a gas turbine engine are the compressor, combustion chamber, and turbine. The compressor compresses incoming air at high pressure, the burner burns fuel and produces gas at high pressure, high temperature and high speed, and the turbine derives energy from the gas. The turbine section of the gas turbine engine is also tasked with producing a usable output shaft force to propel the fan. In addition, it must provide power to operate the compressor and all engine accessories. It does this by expanding the gas at a high temperature, high pressure and high speed and converting the gas energy into mechanical energy in the form of shaft energy. There are high pressure and low-pressure turbines. The high-pressure turbine is designed to extract work from the high-pressure fluid when it initially enters the turbine. The low-pressure turbine is designed to extract work from the fluid leaving the low-pressure turbine. The three-roll motor is the engine that has three sets of pre-combustion compressors and three sets of turbines behind. The pulley consists of a compressor and a corresponding turbine used to extract energy from the exhaust gases to rotate the compressor. Convection cooling is the simplest and one of the first to be used. In thermal cooling, the cooling air passes through very complex internal short channels and a different aspect ratio. Flow channels are commonly used in square or rectangular turbine blades, but the curved shape of the turbine blade can prohibit the efficient use of square or rectangular coolant flow channel. These ducts will contain rib turbines for more efficient heat transfer from the wall to the coolant. Part of this air was directed to affect the front and end edge of the blade. In some cases, this cooled air passes through the return channel 180 ° and is expelled from the back edge. Rib turbines only disturb the flow that passes near the channel wall, so the pressure drop penalty caused by rib turbines is reasonable for designing the internal cooling steps of the blade. The heat transfer rate in the rectangular coolant will pass through the rib turbines, which mainly depend on the engineering of the rib turbines, such as size, shape, distribution, attack flow angle and Reynolds flow number.

©IJRASET: All Rights are Reserved

The complexities in determining the heat transfer coefficient associated with internal cooling are listed below. Fluid mechanics of internal cooling ducts are very complicated due to bending and roughness. Mixing and turbulent separation can cause these refrigerated jets, cavities, etc.Each reel receives a name. N1, N2, and N3. N1 is the large portion of the fan in front of the engine. N2 is the middle pressure compressor section. The N3 is a high pressure compressor section. Each part of the compressor wants to rotate at its own speed, and if it is allowed to do it like a triple coil motor, it can work more efficiently. It can spin at top speed, and all modern engines have two sets of compressors (HP and IP) and a fan section that provides the majority of the thrust.

II. LITERATURE SURVEY

Various traditional cooling concepts are used in different combinations to properly cool the turbine blades and blades. Gas turbine cooling technology and heat transfer by Han, J. C., et al. (2001) provides a detailed description of the heat transfer and cooling technology in the turbine blade. The author has compiled a comprehensive review of the technology for gas turbine cooling that includes techniques for improving heat transfer in internal cooling channels. The book also includes many studies conducted over the years in a wide range of rib configurations in different cooling channels using many experimental techniques.The first studies examined cooling channels using orthogonal ribs. Han, J.S. (1988) performed a pilot study on grooved channels with orthogonal square ribs. Three different polygonal channels with different aspect ratios were considered. The distances between the sides and the Reynolds number were also taken into account. The work collected a detailed study on the effect of the rib spacing on the heat transfer performance of the slotted channels. The author has also provided us with heat transfer and friction correlation. This article provides us with the experimental results that were compared to the numerical study obtained through numerical analysis.[1] Han, J. C. et al. (1992) now performed multichannel experiments with angled ribs. The ribs are oriented to angles 30, 45, 60, and 90. The main objective of the study is the effect of the direction of the ribs on thermal performance. It was concluded that angled ribs had a higher heat transfer performance than orthogonal ribs. The friction and heat transfer correlations are derived from their studies. 4 4[2] Iacovides, H. (1998) did the math on polygon rectangular paths. The fixed and circular rectangular sections were considered in his study. The work of the calculation was based on the square orthogonal ribs. The work mainly focused on the impact of the turbulence model on channel performance. A differential tension model was developed that proved to have produced better results than the standard k-e model. [3] Bonhoff, B., et al. (1999) He conducted an experimental and numerical study in cooling channels with 45 ° ribs. A stationary channel is used in the study. Experimental and numerical results are obtained using Reynolds pressure model. The results were well compared. The case of periodic boundaries was used for numerical study. Speed distribution and heat transfer were recorded. [4] Iacovides, H. and Raisee, M. (2000) performed a computational study on a rough rib pathway using the Reynolds low number disorder model. Standard k-e and k-w standard turbulence models were used. The periodic limit condition was used and parameters such as the Nusselt number were obtained. The differential pressure model was also developed. The DSM model provided improved predictions of heat transfer after re-joining the flow and on the ribs. However, the model was unable to predict the impact of Reynolds' number.[5] Lin, Y. L., et.al (2001) conducted a numerical study of flow and heat transfer in a 45-degree channel with ribs. Rotary and non-periodic channels were considered. The analysis was performed in 3D flow. With 3D analysis, secondary flow was predicted. These results also explain how the nature of the fluid flow affects the heat transfer on the surface. Moreover, the secondary flow has clear effects on heat transfer. 5 5 [6] Agarwal, P. (2001) conducted a detailed study of heat transfer and flow in serpentine cooling channels. The channel was constructed with a aspect ratio of 1: 4 and 4: 1. The ribs were used at an angle of 90 and 45 degrees. Comparisons were made between the angle direction and the effect of channel aspect ratios. Slanted ribs showed that the heat transfer performance was higher than orthogonal ribs. Also, the wider channels proved to be better than the narrow channel.[7] conducted by Bridberg, c. (2002) An Intensive Study on Modeling Turbulence in Internal Cooling of Gas Turbine Blades. The fixed and rotary channels are taken into consideration. It was concluded that the standard k-w model produced an accurate method for simulating complex geometric figures.[8] Wright conducted to. M. (2008) Experimental Analysis on 3: 1 Rectangular Channel with Angled Ribs. Spacing between 10 and 20 sides was considered, the width of the sides was adjusted and their effect on performance was recorded. The influence of Reynolds number is also taken into account. The study concluded that the angled channel has the effect of heat transfer more than the soft channel. Heat transfer and friction links were obtained. Many researchers such as Al-Qahtani, a. M. (2002), Han, J. These works were useful for studying the effect of rib spacing, rib width, and flow parameters on heat transfer performance. The works gave an idea of how to change the heat transfer when the channel hardened. The amount of particles present hardens the matrix by increasing the content of fly ash and B4c particles. Therefore, the observation was more resistant to corrosion. The MMC welding machine wears a lot with a smaller fraction of weight of fly ash and B4c particles, and the corrosion increases linearly over time. [9-11].Han and Chen presented a summary of numerical and experimental research on the internal cooling steps of a turbine blade using rib inverters.

They highlighted, on average, the ability of the RANS models to solve the 3D complex. Flow physics in the internal cooling paths of the rotary turbine blade. Murata and Mochizuki compared laminar heat transfer and turbulence in a fixed square channel with angled or transverse rib turbines. The channel heat transfer is simulated numerically using the second order finite difference method. In turbulent cases using 60 ° and 90 ° ribs, reflux at the midpoint between the ribs and the unstable reverse flow against the rib formed high values of heat transfer at the average time. In the case of laminar flow, the low-torque fluid near the grooved wall has a lower impact on the flow field. Viswanathan and Tafti performed separate spiral simulations (DES) for turbulent flow and heat transfer in a two-step internal cooling channel. The simulation was applied to a 90 ° channel fixed channel. The analyser used a generic system that does not compress direct digital simulation and a great simulation of turbulence disorder using the DES version of the k-modelo model. "The flow in the space between the ribs is dominated by the heat transfer enhanced by the vortex in front of the rib, in the anchorage area behind the rib and at the intersection of the rib with the smooth walls."

III. METHODOLOGY

The ribs are reflected at an angle of 450 around the centre line of the channel to create two directions of the W-shaped ribs by 45 °. In Figure 3.1, the first direction is called the Rib 450 in W shape while the second is called the Inverted Rib as the W 450. The conceptual views of the secondary flow vortices induced in the Rib can also be seen in Figure 1.1. The W-shaped rib 450 is assumed to create two opposite rotating rotations when the liquid approaches the W-shaped leg, divided into two streams. "Each current along the rib moves toward the outer walls and then returns to the midline forming an opposite vortex. The fluid near the surface that approaches the inverted rib in the shape of the W450 begins to converge from the outer walls toward the central line of the channel. This creates a stagnation zone with fluid drop before Traveling over the rib and creating anti-roll swirls, the recession area expects the 45_W rib to perform better than upside-down ribs in non-periodic conditions.

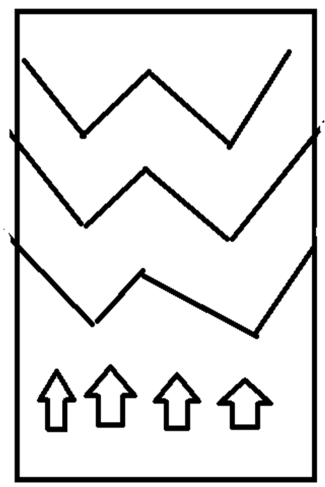


Figure 1.1: W-Shaped Rib Turbulators

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. To study the different arrangements of W-Shaped Ribbed turbulators in a two pass rectangular duct with 180 degree U bend. And different arrangements are as show in Figure 1.2.

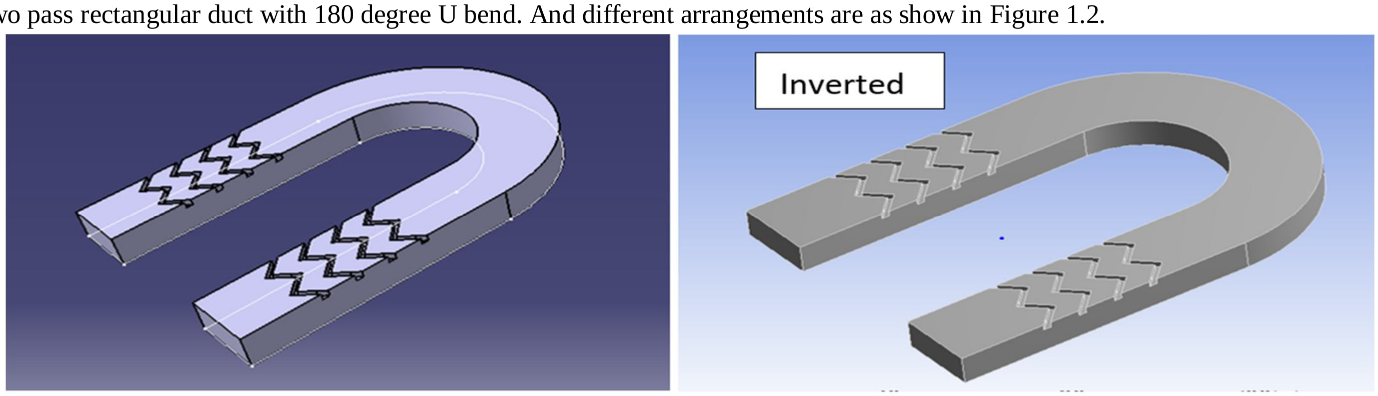
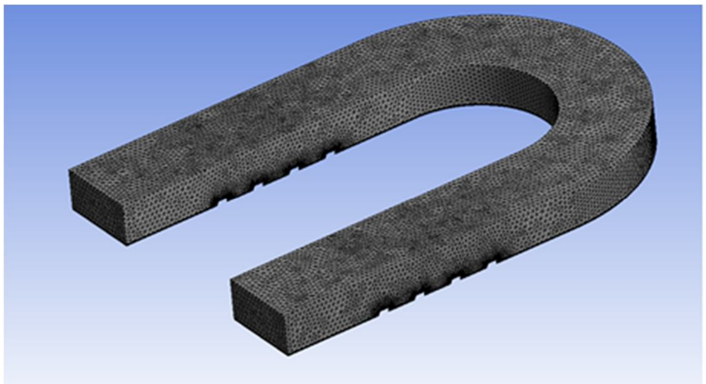


Figure 1.2: Arrangements of W-Shaped Ribs in Rectangular Channel.

©IJRASET: All Rights are Reserved

Computational Fluid Dynamics, Generally Below is an abbreviation CFD, is a numerical method for calculating fluid dynamics. In short, it is implemented by dividing the arithmetic field into small cells where flow is formed and solving flow equations. Since almost all fluid flows are turbulent, different CFD methods are used to simulate turbulence. They can be divided into different categories where some are located; Turbulence models for Navier-Stokes equations with Reynolds Average (RANS), Big Eddy Simulation (LES), Direct Numerical Simulation (DNS) and Lattice Boltzmann Method (LBM). LBM differs from other methods by simulating particle movements and targets on the hydraulic dynamics of the Navier-Stokes equation. On the other hand, RANS uses Navier Stokes equations as a starting point and aims to solve them. DNS does not use the disturbance model; It calculates all fluctuations of turbulent speed and therefore requires small steps and time cells that require large computing resources. LES focuses on large swirls in flow and requires very large computing resources. However, the most common method of simulating turbulence is the computation of the flow average time properties, such as mean pressure, average velocity, etc., which in most cases provide sufficient information about the flow. This method has modest demand on a computer and is done with RANS models. Tool used to generate Tetrahedral mesh in ansys shown in Fig 1.3, in Gambit software. Tetrahedral mesh elements are used and the total mesh size of 2 Million is used to capture the Ribs near wall and rectangular flow channel efficiently.



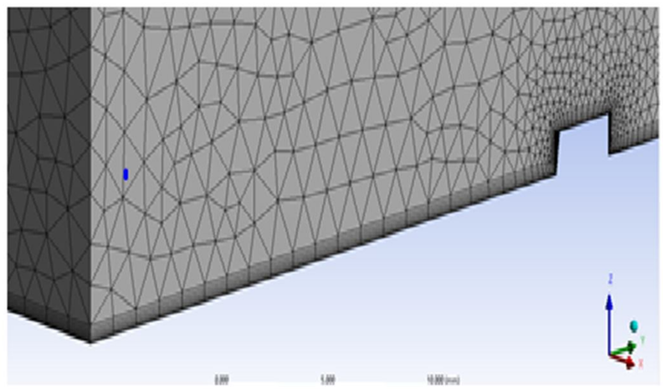


Figure 1.3: Mesh showing Prism Layers

Boundary Condition Set up is followed as shown in Table I. 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 450 K. And air inlet temperature is 300K, Operating Pressure is 101325 Pa.

Table I
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	Aluminium
Duct Wall	Wall	Aluminium
Turbulence Model	-	K-Epsilon - Standard
Upwind Scheme	-	Second Order Upwind

©IJRASET: All Rights Reserved

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

IV. NUMERICAL ANSLYSIS – RESULTS AND DISCUSSIONS

Three models arrangements are studied with respect to range of Reynolds numbers. Ribs wall temperature is around 360 K and the inlet air temperature is around 300 K. The analysis is carried out shown in Table II, cases heat transfer coefficient and Nusselt Numbers are determined.

Table II
Details of Case Study

	Model Type	Reynolds Number Re	Velocity Inlet m/s	Inlet Temperature K	Ribs Wall Temperature K
Case 1	I	5000	2.88	300	450
Case 2	I	15000	8.63	300	450
Case 3	I	40000	25.87	300	450
Case 4	II	5000	2.88	300	450
Case 5	II	15000	8.63	300	450
Case 6	II	40000	25.87	300	450

Case 1 involves study with straight rib arrangements with both passes. 16 Sections are taken as shown below, 1-8 sections are from first pass and 9-8 are taken from second pass in the channel. Figure 1.4 Contours of velocity magnitude at mid plane of upstream and downstream of U-Flow Rectangular Channel Case 1. Figure 1.5 Contours of total temperature at mid plane of upstream and downstream of U-Flow Rectangular Channel Case 1

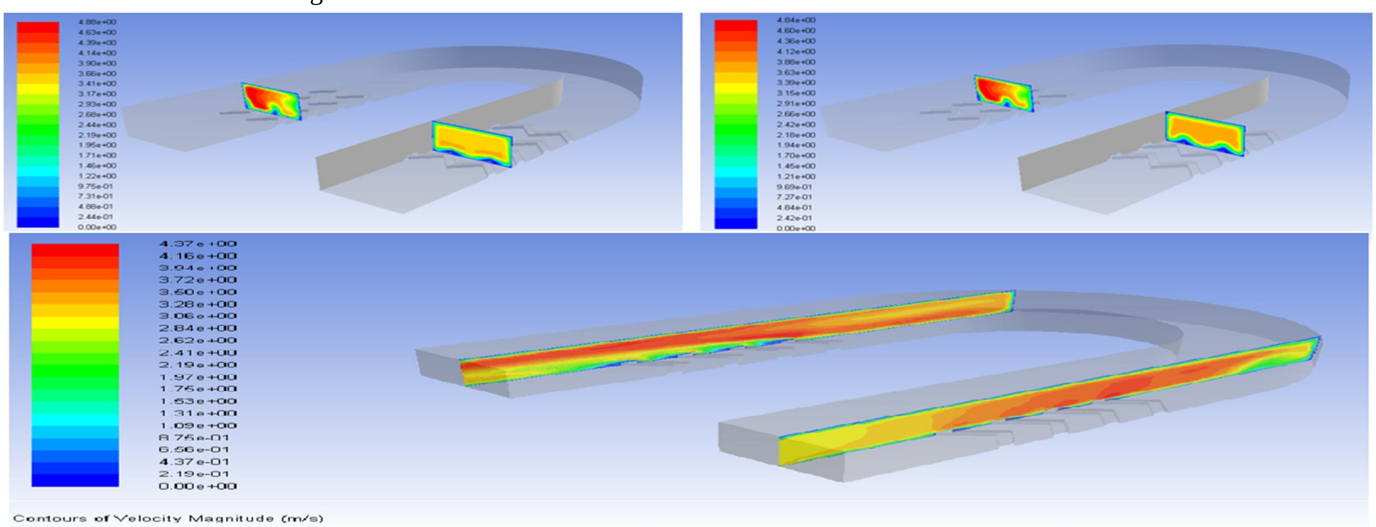


Figure 1.4 Contours of velocity magnitude at mid plane

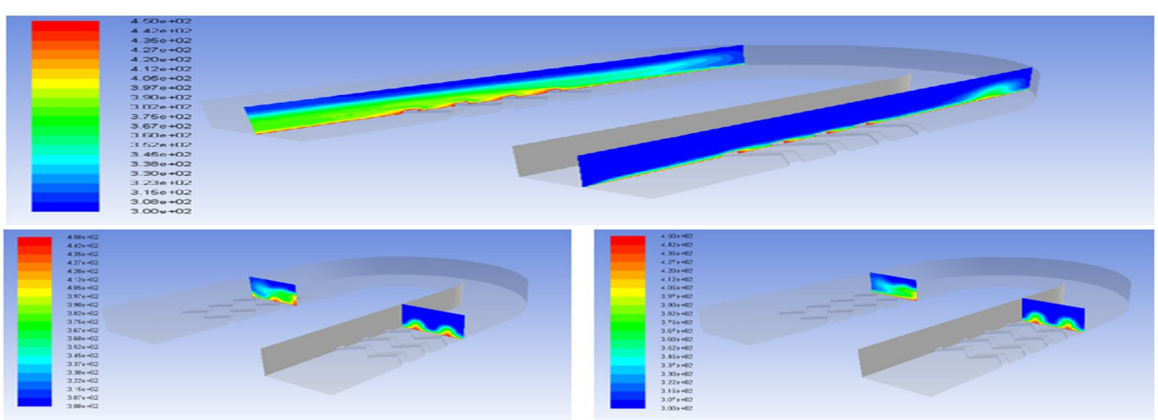
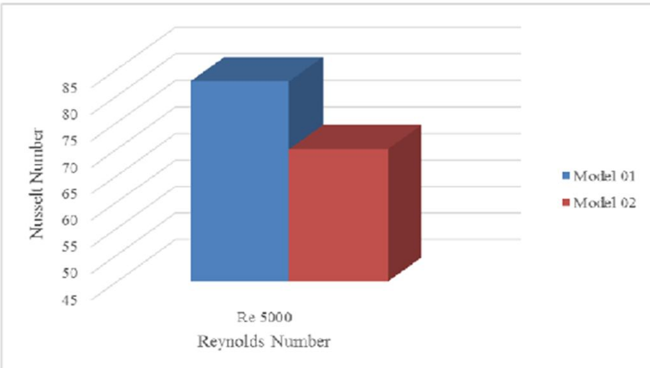


Figure 1.5 Contours of total temperature at mid plane

By the Nusselt Number predictions of cases 1-6, we can observe that for each respective Reynolds number Model TYPE I yields higher heat transfer in comparison with Model TYPE II. From this study we can recommend for rectangular duct flows the inverted W shaped ribs – Model TYPE I design is the best model for higher heat extraction from the surface. Figure 1.6 Nusselt Number Comparison for Reynolds Number 5000 and Reynolds Number 45000



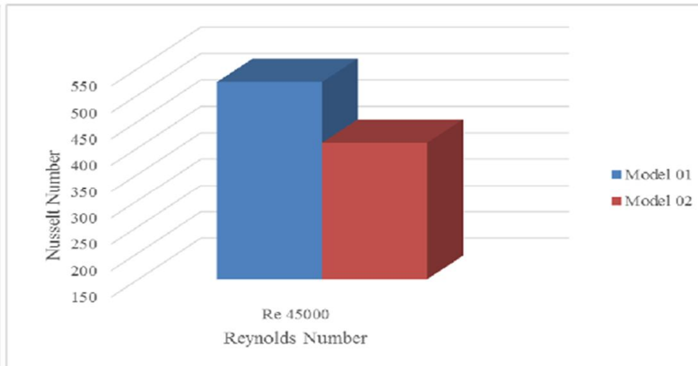


Figure 1.6Nusselt Number Comparison for Reynolds Number 5000 and Reynolds Number 45000

V. CONCLUSION AND FUTURE WORK

Model 1 with Inverted W Shaped Rib Arrangement is best suited for flows with comparatively Low Reynolds Number 5000. Model 1 Proves to perform with same effect, and it is recommended for Reynolds Number around 15000 Model 1 with best suitable for flows with high Reynolds Number around 40000.Henceforth it can be observed that for Reynolds Number of range 15000 Model 1 yields higher Heat Transfer Coefficient and High Nusselt Number which concludes as best suitable arrangement and flow velocity. Whereas for Reynolds Number of Re 5000 and Re 40000 Model 2 performs very worse in comparison with Re 15000 .

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 with discrete setup can be studied 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.

REFERENCES

[1] Han, J.C., “Heat Transfer and Friction Characteristics in Rectangular Channels with Rib Turbulators”, Journal of Heat Transfer, Vol. 110, pp. 321 – 328, May 1992.

[2] Iacovides, H., “Computation of Flow and Heat Transfer Through Rotating Ribbed Passages”, Journal of Heat and Fluid Flow, Vol. 19, pp. 393 - 400, 1998

[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] Iacovides, H., and Raisee, M., “Computation of Flow and Heat Transfer in Two Dimensional Rib Roughened passages, using Low Reynolds Number Turbulence 119

[5] Lin, Y.L., Shih, T. P., Stephens, M.A., and Chyu, M.K., “A Numerical Study of Flow and Heat Transfer in a Smooth and Ribbed U-Duct with and without Rotation”, Journal of Heat Transfer, Vol.123, pp. 219 – 232, April 2001

[6] 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.

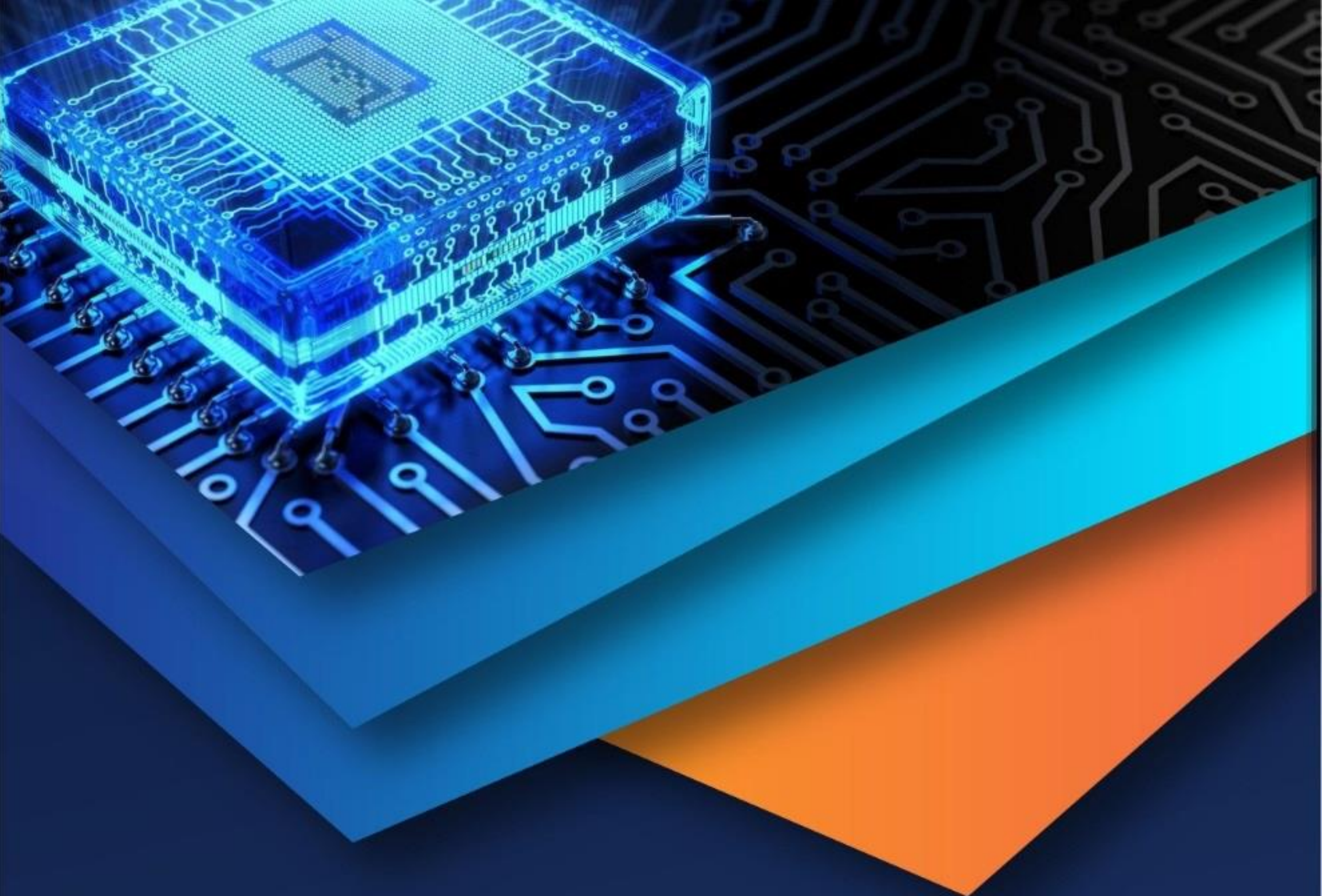
[7] Bredberg, J., “Turbulence Modeling for Internal Cooling of Gas Turbine Blades”, Doctoral Thesis, Department of Thermo and Fluid Dynamics, Chalmers University, Sweden 2002.

[8] 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.

[9] Uppada Rama Kanth, Putti Srinivasa Rao, Mallarapu Gopi Krishna, “Mechanical behaviour of fly ash/SiC particles reinforced Al-Zn alloy-based metal matrix composites fabricated by stir casting method”, Journal of Materials Research and Technology Volume 8, Issue 1, January–March 2019, Pages 737-744.

[10] U. S. Ramakanth & Putti. Srinivasa Rao, “Wear Behavior of Al 7075/Fa/Sic Hybrid Composites”, International Journal of Mechanical and Production Engineering Research and Development (IJMPERD) ISSN(P): 2249-6890; ISSN(E): 2249-8001 Vol. 9, Issue 2, Apr 2019, 331-338.

[11] U. Ramakanth & Putti. Srinivasa Rao, “Corrosion Studies on Ceramic Particulates Reinforced Hybrid Metal Matrix Composites”, International Journal of Mechanical Engineering and Technology (IJMET) Volume 10, Issue 01, January 2019, pp. 661–666.



10.22214/IJRASET

INDEX COPERNICUS 45.98

IMPACT FACTOR: 7.129

TOGETHER WE REACH THE GOAL

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089 (24*7 Support on Whatsapp)