Numerical Investigation of Heat Transfer Enhancement in Circular Pipe using various RIB Geometries

Chetankumar [1], Dr Pravin V Honguntikar[2]

M.Tech Student[1] Prof & Hod Mechanical[2]
M.Tech in Thermal Power Engineering, Poojya Doddappa Appa College of Engineering Kalaburagi-585102

***

Abstract - Heat transfer enhancement techniques can be divided into two categories passive and active. In passive heat transfer enhancement an object which does not use external energy, such as groove inside the tube, has the duty of increasing the heat transfer rate. Forced convection heat transfer is the most frequently employed mode of the heat transfer in heat exchangers or in various chemical process plants. During the last two decades, computational fluid dynamics (CFD) has become a very powerful tool in the process of industries not only for the research and development of new processes but also for the understanding and optimization of existing one. Perturbations or interruptions provided in the passage of heat exchanger generate the vortices downstream. The formation of these natural vortices, augment local heat transfer abruptly. The effect on convective heat transfer enhancement and friction characteristics by providing Flat conical and right angle ribs inside a circular pipe is computationally investigated in detail.Different variations of height, width and pitch of the ribs are used to optimize the rate of heat transfer through the pipe. Liquid water is employed as the working fluid. Input parameters of Reynolds Number ranging from 5000-60000 with axial flow along the pipe and constant heat flux of 50 W/cm2 to the pipe surface is used. After validation with the existing literature, Realizable k-ε turbulent model with enhanced wall function is used in commercial CFD software ANSYS FLUENT. The outcome of the investigation shows that the ribs provided on the inside of the pipe surface enhance the turbulence in the flow and produce recirculation which disturb the thermal boundary layer behind the ribs and thus help in enhancing the rate of heat transfer through the pipe.

1.1 Introduction

Enhancement of heat transfer in engineering applications had been a subject of interest in many research studies. Two different techniques for heat transfer enhancement are generally used; first, fluid additives like nanoparticles are used, second, geometry modification are made by roughening the heat transfer surfaces using ribs, grooves or wires or applying helical corrugated tubes. These modified geometries create the chaotic and good mixing in fluid flow due to the secondary flow regions which appear near the wall and cause to reduce the thickness of thermal boundary layer in a manner that increases the heat transfer rate.

Expanding interest of energy required either more generation of energy or finds an elective way which decreases the energy utilization. The change, usage and recovery of energy in each field for example either business or local involve a heat transfer process. Some significant zones in which forced convection heat transfer happen are recorded as power sector, refrigeration and air conditioning, thermal processing of chemicals, electrical machines and electronic gadgets, squander heat recuperation in manufacturing, inside cooling of engine; turbo-machinery frameworks and some more. Rib turbulators are playing a significant role in this regard. There are a few strategies, for example, jet impingement, film cooling, rib turbulators, shaped internal cooling sections, dimple cooling, appeared in Figure below, to cool gas turbine sharp edged blade. The jet impingement is utilized to cool the leading edge, pin fin cooling at the trailing edge and rib turbulators are utilized cool the internal entries. The present investigation centres on the thermal enhancement of heat utilizing rib turbulators or turbulence promoters.Introduction to turbulent flowIn General The flow is two types Laminar and turbulent flow where in the laminar flow is mostly smooth and straight with no obligations where the flow is mostly linear with no disturbances turbulence flow is formed mostly due to the separation caused to non linear surfaces with which high speed velocities formed over the structures form a cone like structure .

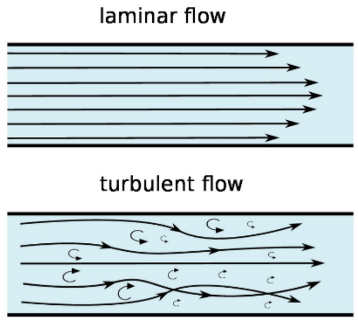


Figure 1.3 Laminar and Turbulent Flow

1.2 Introduction to Turbulence modelling

The number of problems possess the turbulence in the flow conditions there are most possible situations where the flow is turbulent due to the velocity or the geometric conditions the computational fluid dynamics has more of the K epsilon and k omega transport equation to determine the type of flow with which the exclusion of the extended surfaces.

The set of mean flow equations are closed by this computational process. There are four different types of methods used in turbulence modelling

1. **RANS :** RANS Stands For Reynolds averaged Numerical solution This method is used to calculate the flow simulation of the turbulence the RANS is used the flow problem with complex problems can be rectified and easily solved using this method this method is mainly used in sst of K-w and Standard of K-epsilon model.
2. **DNS :** Dns means Direct numerical solution the scale of all residuals in naiver stokes equation is solved using this method
3. **LES:** LES means Large Eddy simulation this type of simulation generally occurs in large scale simulations like dam break problems and cyclone simulation with large eddy flows

4. **DES:** DES stands for Dethatched Eddy simulation is the combination of RAS and LES which generally computes the Eddy simulations For most bigger problems.

2. Literature Review

Pressure drop and heat transfer predictions often are accurate even in complex geometries. Thus CFD has become the state of the art in thermal engineering like in heat exchanger design.

Aubin et al. [1] investigated the effect of the modeling approach, discretization and turbulence model on mean velocities and turbulent kinetic energy and global quantities such as the power and circulation numbers. The results have been validated by laser Doppler velocimetry data.

Rahimi et al. [2] reports experimental and CFD investigations on friction factor, Nusselt number and thermal hydraulic performance of a tube equipped with the classic and three modified twisted tape inserts. Uses of the artificial grooved tubes are widely used in modern heat exchangers, because they are very effective in heat transfer augmentation. Bilen et al. [3] experimentally investigated the surface heat transfer and friction characteristics of a fully developed turbulent air flow in different grooved tubes. Tests were performed for Reynolds number range of 10000 to 38000 and for different geometric grooved shape (circular, trapezoidal, and rectangular) tubes.

Kumar and Saini [4] presents the performance of a solar air heater duct provided with artificial roughness in the form of a thin circular wire in arc shaped geometry has been analyzed using CFD. The effect of arc shaped geometry on heat transfer coefficient, friction factor, and performance enhancement were investigated covering the range of roughness parameter (from 0.0299 to 0.0426) and working parameter (Re from 6000 to 18000). Different turbulent models have been used for the analysis and their results were compared. Renormalization group (RNG) k-e model based results have been found in good agreement and accordingly this model was used to predict heat transfer and friction factor in the duct.

Xiong and Chung [5] analyzed with the help of CFD solver to isolate the roughness effect and solve the 3-D Navier-Stokes equations for the water flow through the general rough micro tubes with diameter D = 50 m and length L = 100 m. The model has a potential to be used for direct simulations of 3-D surface roughness effects on the slip flow.

Craft et al. [6] reports computations of the fluid and heat transfer from a row of round jets impinging onto a concave semicircular surface designed to reproduce important flow features found in internal turbine blade cooling applications.

Iacovides et al. [7] discussed the results of a combined experimental and numerical study of flow and heat transfer in a straight duct, with ribs of square cross-section along the two opposite walls, in a staggered arrangement and at an angle of 45º to the main flow direction.

Chaube et al. [8] studied a computational analysis of heat transfer augmentation and flow characteristics due to artificial roughness in the form of ribs on a broad, heated wall of rectangular duct for turbulent flow has been carried out.

Karagoz and Kaya [9] presents CFD investigation of the flow field and heat transfer characteristics in a tangential inlet cyclone which is mainly used for the separation of a two phase flow. Finite volume based Fluent® software was used and the RNG k-e turbulence model was adopted for the modeling highly swirling turbulent flow.

Rigby and Evans [10] reports single and multiphase flow dynamics around bluff bodies in liquid cross-flows have been investigated using CFD modeling.

Li et al. [11] investigated the periodicities of convection heat transfer in channels with periodically grooved parts are studied numerically using an unsteady state model. The governing equations are discretized using SIMPLE algorithm with QUICK scheme.

Eiamsa-Ard et al. [12] presents the applications of a mathematical model for simulation of the swirling flow in a tube induced by loose – fit twisted tape insertion.

Zimparov [13, 14] investigated a simple mathematical model following the suggestions of Smithberg and Landis has been created to predict the friction factors for the case of a fully developed turbulent flow in a spirally corrugated tube combined with a twisted tape insert.

Goto et al. [15, 16] investigated the condensation and evaporation heat transfer of R410A inside internally grooved horizontal tubes. The measured local pressure drop, heat transfer coefficients were compared with the predicted values from previous correlations proposed by the authors.

Promvonge [17] investigated that the snail entry with the coiled square-wire provides higher heat transfer rate than that with the circular tube of under the same conditions.

Promvonge [18] studied thermal augmentation in circular tube with twisted tape and wire coil turbulators. Also he presents that heat transfer enhancement can create one or more combinations of the following conditions that are favorable for the increase in heat transfer rate with an Selvaraj, P., et al.: Computational Fluid Dynamic Analysis on Heat Transfer and ... 1126 THERMAL SCIENCE: Year 2013, Vol. 17, No. 4, pp. 1125-1137 undesirable in increase in friction: (1) Interruption of boundary layer development and

undesirable in increase in friction: (1) Interruption of boundary layer development and rising degree of turbulence (2) increase in heat transfer area (3) generating of swirling and/or secondary flows. To date, several studies have been focused on passive heat transfer enhancement methods reverse/swirl flow devices (rib, groove, wire coil, conical ring snail entry, twisted tape, winglet, etc.,) form an important group of passive augmentation technique.

Chiu and Jang [19] numerical and experimental analyses were carried out to study thermal hydraulic characteristics of air flow inside a circular tube with different tube inserts. Zhang et al. [20] experimental study on evaporation heat transfer of R417A flowing inside horizontal smooth and two internally grooved tubes with different geometrical parameters were conducted with the mass flow rate range from 176 to 344 kg/s. Based on the experimental results, the mechanism and mass flow rate, heat flux, vapor quality and enhanced surface influencing the evaporation heat transfer co-efficient were analyzed and discussed.

Li et al. [21] studied the turbulent heat transfer and flow resistance in an enhanced heat transfer tube, the DDIR tube, were studies experimentally and numerically, water was used as the working fluid with Re between 15000 and 60000. The numerical simulations solved the 3-D Reynolds-averaged Navier-Stokes equations with the standard k-e model in the commercial computational fluid dynamics code, Fluent. The numerical results agree well with the experimental data, with the largest discrepancy of 10% for the Nusselt numbers and 15% for friction factors.

Karwa et al. [22] carried out an experimental investigation of heat transfer and friction for the flow of air in rectangular ducts with repeated rib roughness on a broad wall. Friction factor and heat transfer coefficient correlations have been developed based on the law of wall similarity and heat-momentum transfer analogy.

Karwa [23] presents results of an experimental investigation of heat transfer and friction in rectangular ducts with repeated rectangular cross- -section ribs on one broad wall in transverse, inclined, v-continues and v-discrete pattern.

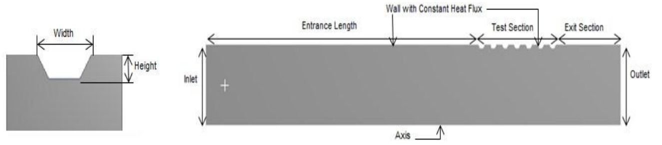
Tanda [24] carried out an experimental study on heat transfer in a rectangular channel with transverse and V-shaped broken ribs. Local heat transfer coefficients were obtained at various Reynolds numbers within the turbulent flow regime.

Vicente et al. [25] carried out experimental study on corrugated tubes in order to obtain their heat transfer and isothermal friction characteristics. The use of water and ethylene glycol as test fluids has allowed covering a wide range of turbulent fluid flow conditions. Reynolds number from 2000 to 90000 and Prandtl number from 2.5 to 100. Helical-wire-coils fitted inside a round tube have been experimentally studied in order to characterize their thermodynamic behavior in laminar, transition and turbulent flow. Experimental correlations of Fanning friction factor and Nusselt number as functions of flow and dimensionless geometric parameters have been proposed by

Garcia et al.[26]. In the present study the pipe flow with three types of grooved tubes (circular, square, trapezoidal) at constant wall flux condition was studied numerically for Reynolds numer ranges from 5000 to 13500 and groove depth ws fixed to investigate the effect of the groove shapes o heat transfer. The thermal hydraulic performane for all the cases was also performed. The present work a CFD modeling was carried out in order to find out the heat transfer in a tube equipped with grooves.

3.1. Geometrical Model Description

The analysis is performed in 2-D, considering the flow to be axisymmetric about the axis of the pipe. The trapezoidal ribs were protruded circumferentially along the length of the pipe at different pitches for different cases. Figure 1(a) shows the geometry of rib and figure 1(b) shows the schematic of the flow with different boundaries. The diameter of the pipe is 10 mm; entrance length is taken as 200 mm considering the fully developed flow at the test section. Length of the test section is 100 mm while the exit section length is 50 mm.



(a) (b)

Figure 1. Trapezoidal rib along the fluid flow.

3.2. Discretization

A structured grid was adopted for the analysis. As wall gradient has a significant role in heat transfer problems a finer mesh is adopted by proper biasing factor near walls to capture the properties of flow in viscous sub layer. The nearest wall distance governed by y^+ approach was considered which lies below 5. A grid independent test was performed to verify the results. Grid test was performed with width of the ribs 0.5mm, height of the ribs 0.25mm and pitch of the ribs 5mm. Grid sizes employed was 64929, 172959 and 458976. Pressure and Velocity distribution was considered for grid independency test. Grid with 458976 numbers of elements was adopted for the simulation after the grid independent test as the y+ value lied below 1 for this.

3.3. Boundary Conditions and Material Selection

Inlet was given selected as velocity inlet with different velocities to account for the desired Reynolds number. For turbulent specification hydraulic diameter of 10mm and the turbulent intensity of 3% were selected. At outlet zero gauge pressure outlet was selected. The upper edge was selected as wall with no slip condition and the test section length was given a constant heat flux of 50 W/cm^2. With centerline as axis the axisymmetric solver was selected. Properties of water are shown in table 1.

Table 1. Properties of the liquid water

Properties	Specifications
Density (kg/m3)	998.2
Specific Heat (J/kg-K)	4182
Thermal Conductivity (W/m-K)	0.6
Viscosity (kg/m-s)	0.001003

3.4. Solver Settings

For pressure velocity coupling SIMPLE scheme was used. Pressure equation has been discretized using Standard. Momentum, Turbulent Dissipation Rate, Turbulent Kinetic Energy and Energy equations has been discretized using second order upwind scheme.

4. Results

4.1 Introduction

This chapter Shows the Results obtained from the Boundary Conditions and setup Done in Chapter 3

There are 4 Designs of ribs done to see the Characteristics of the Outlet temperature and Reynolds Number

Case 1 Square Ribs

I. Re=5000

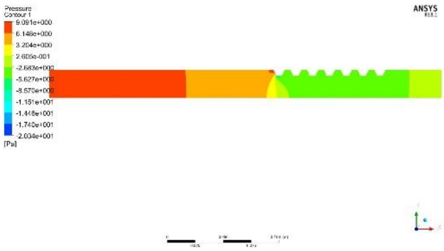


Fig 4.1 Pressure Distribution in Pipe with Square Ribs at Re=5000.

The above Figure Represents the Pressure Distribution of water at Re=5000 where blue color represents low pressure area and red color represents high pressure area left side of the picture the colored bar indicates the respected value to the Color significant in the contour from the Figure 9.09 pa is occurring at inlet and 0.260 at outlet.

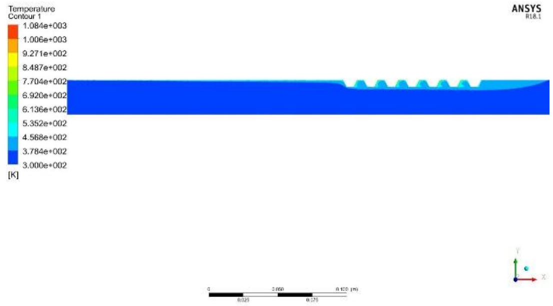


Fig 4.2 Temperature Distribution in Pipe with Square Ribs at Re=5000.

The above Figure Represents the Temperature Distribution of water at Re=5000 where blue color represents low pressure area and red color represents high pressure area left side of the picture the colored bar indicates the respected value to the Color significant in the contour from the Figure 300K is occurring at inlet and 325.739K at outlet.

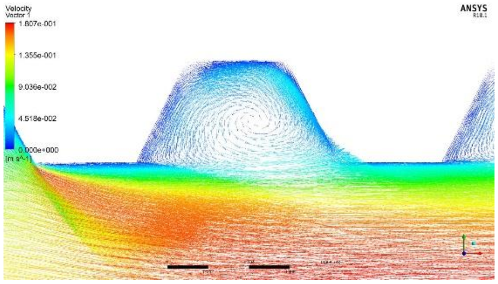


Fig 4.3 Velcoity Vector Colse up view at ribs in Pipe with Square Ribs at Re=5000.

The above Figure Represents the Vector representation pf the flow direction and highlighted recirculation in the Flow of water in a square ribbed Pipe at Re =5000 we can observe from the Figure that there is a clear recirculation Happening at Rib area which enhances the heat transfer.

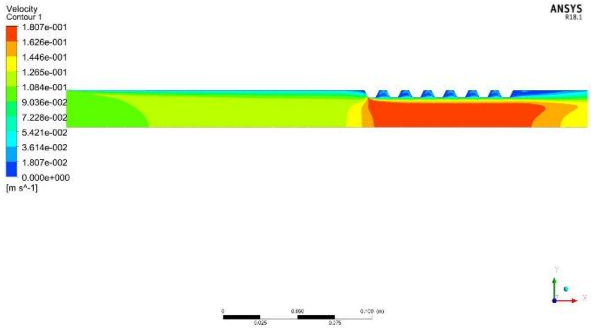
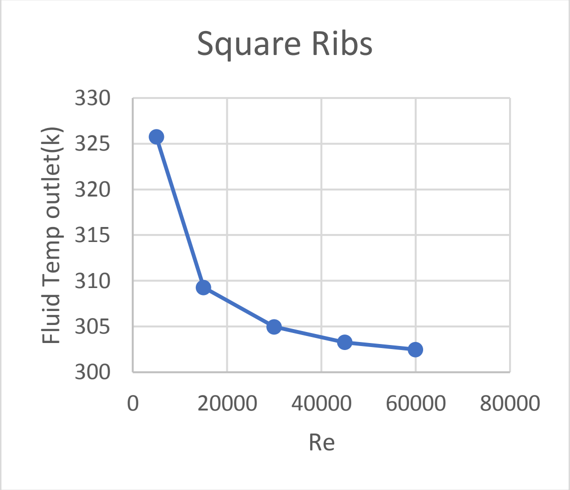


Fig 4.4 Velocity Distribution in Pipe with Square Ribs at Re=5000.

The above Figure Represents the Velocity Distribution of water at Re=5000 where blue color represents low pressure area and red color represents high pressure area left side of the picture the colored bar indicates the respected value to the Color significant in the contour from the Figure 0.1 m/s is occurring at inlet and 0.144 at outlet.

Table 1 Outlet Temperature of Square ribs.

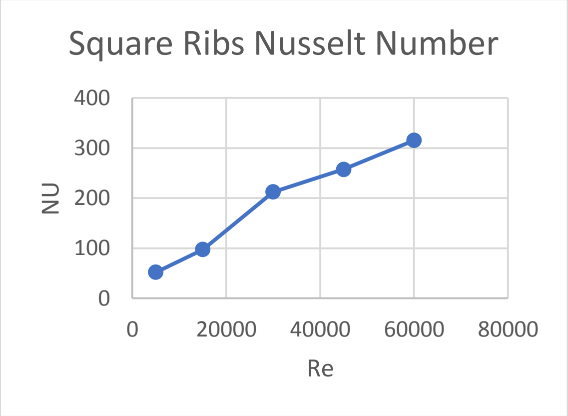
RE	Outlet Temperature of water using Square Ribs (K)
5000	325.739
15000	309.265
30000	304.965
45000	303.257
60000	302.464



Plot 1 Reynolds Number Vs outlet Temperature

Table 2 Nusselt Number of Square Ribs.

Reynolds Number	Square Ribs Nusselt Number
5000	52.2254
15000	97.558
30000	212.3314
45000	257.447
60000	315.324



Plot 2 Reynolds Number Vs Nusselt Number of Square Ribs.

Case 2 Smooth Ribs

I. Re=5000

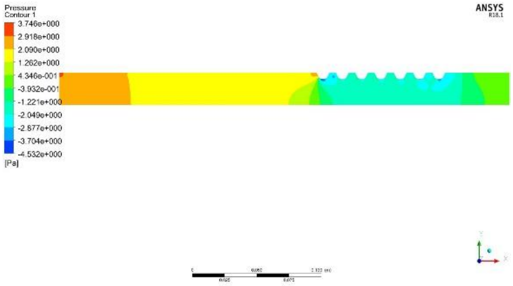


Fig 4.5 Pressure Distribution in Pipe with Smooth Ribs at Re=5000.

The above Figure Represents the Pressure Distribution of water at Re=5000 where blue color represents low pressure area and red color represents high pressure area left side of the picture the colored bar indicates the respected value to the Color significant in the contour from the Figure 3.746 pa is occurring at inlet and 0.434 pa at outlet.

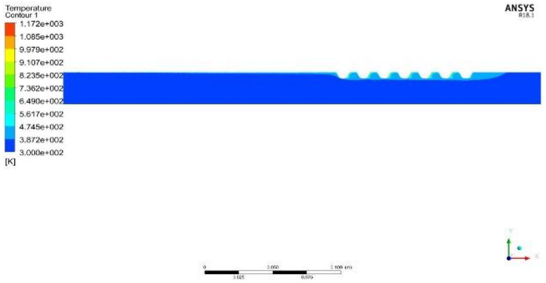


Fig 4.6 Temperature Distribution in Pipe with Smooth Ribs at Re=5000.

The above Figure Represents the Temperature Distribution of water at Re=5000 where blue color represents low pressure area and red color represents high pressure area left side of the picture the colored bar indicates the respected value to the Color significant in the contour from the Figure 300K is occurring at inlet and 320.906K at outlet.

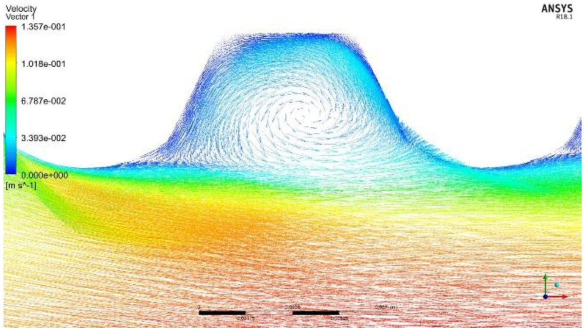


Fig 4.7 Velcoity Vector Colse up view at ribs in Pipe with Smooth Ribs at Re=5000.

The above Figure Represents the Vector representation pf the flow direction and highlighted recirculation in the Flow of water in a Smooth ribbed Pipe at Re =5000 we can observe from the Figure that there is a clear recirculation Happening at Rib area which enhances the heat transfer.

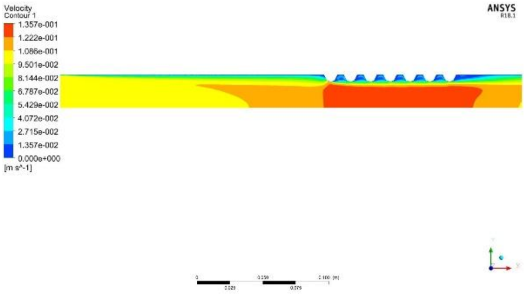
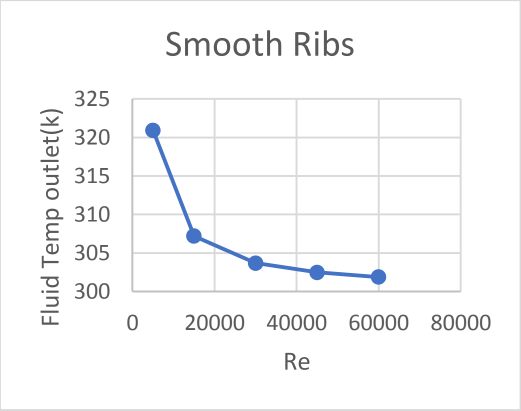


Fig 4.8 Velocity Distribution in Pipe with Smooth Ribs at Re=5000.

The above Figure Represents the Velocity Distribution of water at Re=5000 where blue color represents low pressure area and red color represents high pressure area left side of the picture the colored bar indicates the respected value to the Color significant in the contour from the Figure 0.1 m/s is occurring at inlet and 0.122 at outlet.

Table 3 outlet Temperature of Smooth Ribs.

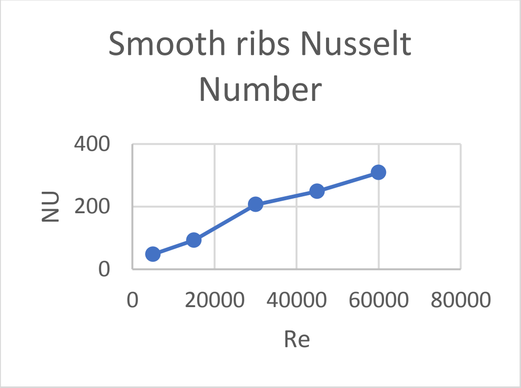
RE	Smooth Ribs
5000	320.906
15000	307.205
30000	303.685
45000	302.488
60000	301.892



Plot 2 Re Vs Fluid Temp outlet (K) of Smooth Ribs.

Table 3 Nusselt Number of Snooth Ribs

Reynolds Number	Smooth ribs Nusselt Number
5000	48.0012
15000	92.7774
30000	206.114
45000	248.662
60000	308.221



Plot 3 Re Vs Nusselt Number of Smooth Ribs

Case 3 Wedge Ribs

II. Re=5000

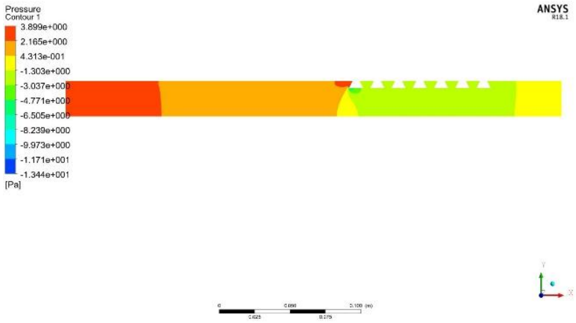


Fig 4.9 Pressure Distribution in Pipe with Wedge Ribs at Re=5000.

The above Figure Represents the Pressure Distribution of water at Re=5000 where blue color represents low pressure area and red color represents high pressure area left side of the picture the colored bar indicates the respected value to the Color significant in the contour from the Figure 3.899 pa is occurring at inlet and 0.4313 pa at outlet.

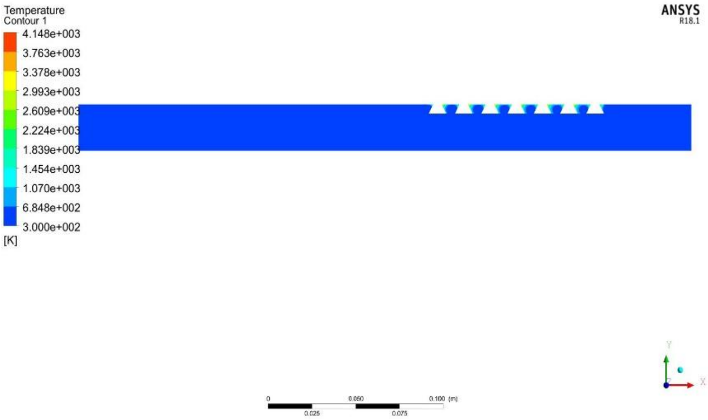


Fig 4.10 Temperature Distribution in Pipe with Wedge Ribs at Re=5000.

The above Figure Represents the Temperature Distribution of water at Re=5000 where blue color represents low pressure area and red color represents high pressure area left side of the picture the colored bar indicates the respected value to the Color significant in the contour from the Figure 300K is occurring at inlet and 326.528 K at outlet.

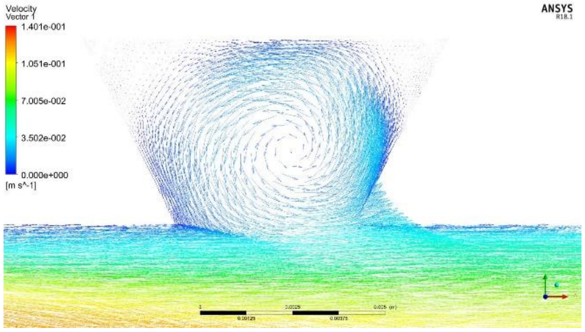


Fig 4.11 Velocity Vector Close up view at ribs in Pipe with Wedge Ribs at Re=5000.

The above Figure Represents the Vector representation pf the flow direction and highlighted recirculation in the Flow of water in a Wedge ribbed Pipe at Re =5000 we can observe from the Figure that there is a clear recirculation Happening at Rib area which enhances the heat transfer.

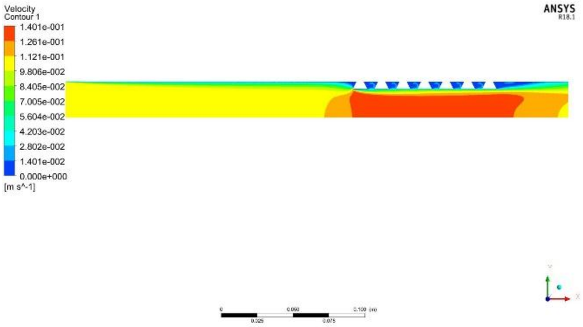
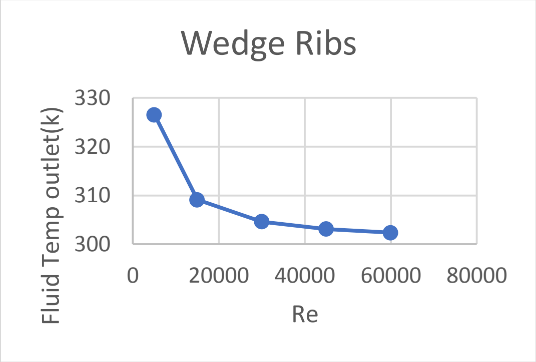


Fig 4.12 Velocity Distribution in Pipe with Wedge Ribs at Re=5000.

The above Figure Represents the Velocity Distribution of water at Re=5000 where blue color represents low pressure area and red color represents high pressure area left side of the picture the colored bar indicates the respected value to the Color significant in the contour from the Figure 0.1 m/s is occurring at inlet and 0.1261 at outlet.

Table 5 Outlet temperature Of Wedge ribs

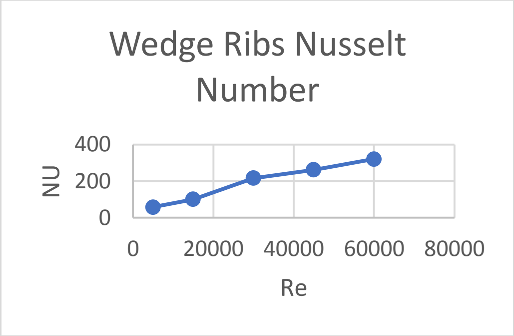
RE	Wedge ribs Outlet Temperature (K)
5000	326.528
15000	309.103
30000	304.619
45000	303.109
60000	302.366



Plot 5 Re Vs Fluid outlet Temp using Wedge Ribs.

Table 6 Nusselt Number of Wedge Ribs

Reynolds Number	Wedge Ribs Nusselt Number
5000	57.1458
15000	100.2547
30000	215.6687
45000	261.22554
60000	320.225



Plot 4 Re Vs Nusselt Number of Wedge Ribs.

Case 4 Funnel Ribs

III. Re=5000

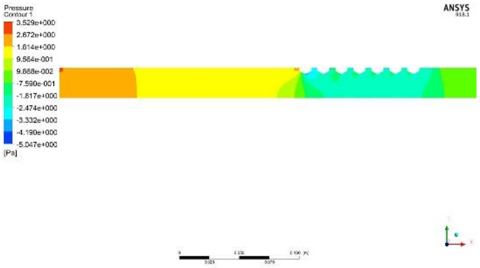


Fig 4.9 Pressure Distribution in Pipe with Funnel Ribs at Re=5000.

The above Figure Represents the Pressure Distribution of water at Re=5000 where blue color represents low pressure area and red color represents high pressure area left side of the picture the colored bar indicates the respected value to the Color significant in the contour from the Figure 3.529 pa is occurring at inlet and 0.9868 pa at outlet.

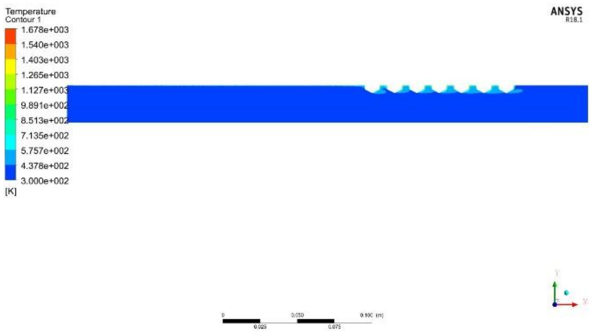


Fig 4.10 Temperature Distribution in Pipe with Funnel Ribs at Re=5000.

The above Figure Represents the Temperature Distribution of water at Re=5000 where blue color represents low pressure area and red color represents high pressure area left side of the picture the colored bar indicates the respected value to the Color significant in the contour from the Figure 300K is occurring at inlet and 321.901 K at outlet.

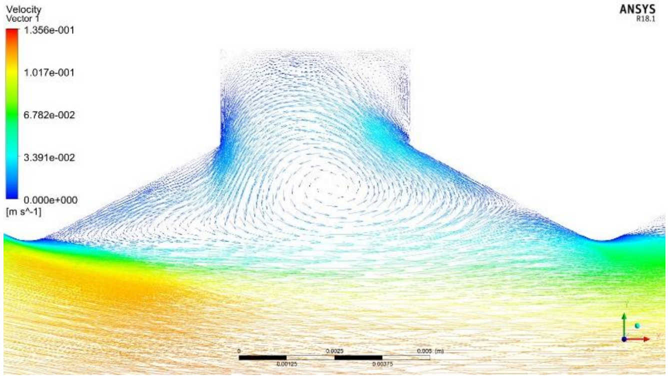


Fig 4.11 Velocity Vector Close up view at ribs in Pipe with Funnel Ribs at Re=5000.

The above Figure Represents the Vector representation pf the flow direction and highlighted recirculation in the Flow of water in a Funnel ribbed Pipe at Re =5000 we can observe from the Figure that there is a clear recirculation Happening at Rib area which enhances the heat transfer.

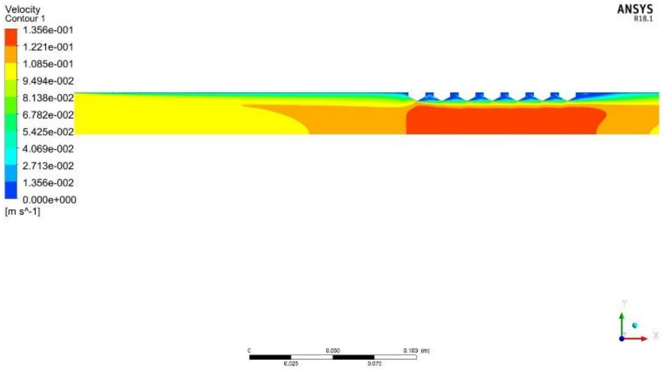
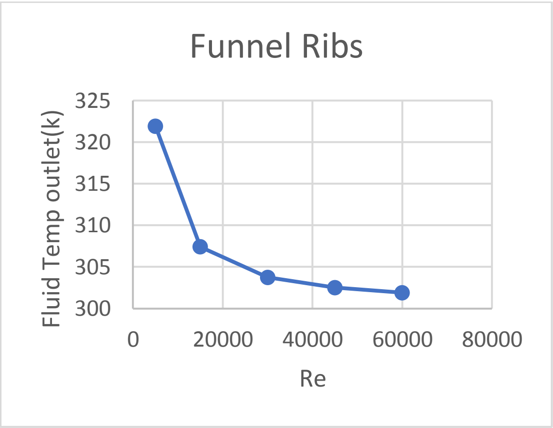


Fig 4.12 Velocity Distribution in Pipe with Funnel Ribs at Re=5000.

The above Figure Represents the Velocity Distribution of water at Re=5000 where blue color represents low pressure area and red color represents high pressure area left side of the picture the colored bar indicates the respected value to the Color significant in the contour from the Figure 0.1 m/s is occurring at inlet and 0.13561 at outlet.

Table 7 Outlet Temperature of Funnel Ribs

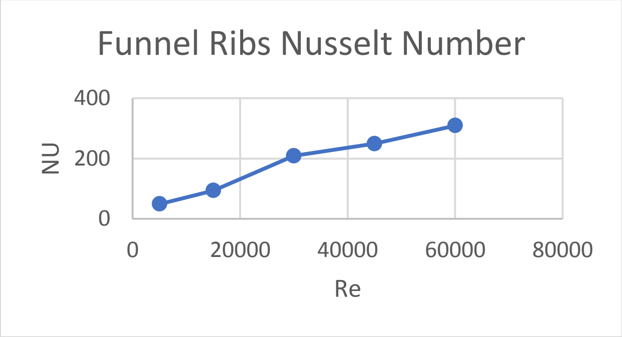
RE	Funnel Ribs
5000	321.909
15000	307.409
30000	303.743
45000	302.51
60000	301.896



Plot 7 Re vs Outlet Temp of Funnel Ribs

Table 8 Nusselt Number of Funnel Ribs.

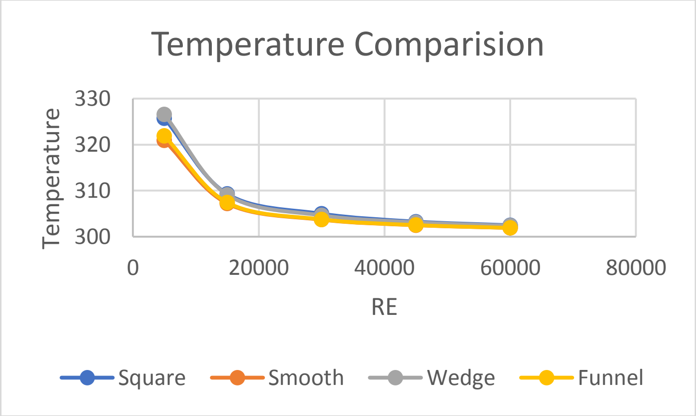
Reynolds Number	Funnel Ribs Nusselt Number
5000	49.54
15000	93.884
30000	208.9997
45000	249.2554
60000	309.22455



Plot 8 Re vs NU with Funnel Ribs

Table 9 Comparison Of Outlet Temperatures

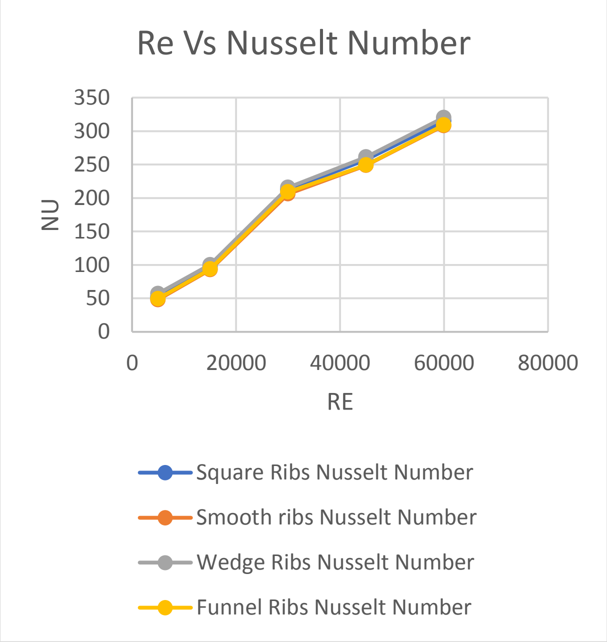
Reynolds Number	Square	Smooth	Wedge	Funnel
5000	325.739	320.906	326.528	321.909
15000	309.265	307.205	309.103	307.409
30000	304.965	303.685	304.619	303.743
45000	303.257	302.488	303.109	302.51
60000	302.464	301.892	302.366	301.896



Plot 9 Comparison Of Re Vs Outlet temperature between different Ribs

Table 10 Comparision of Nusselt Number

Reynolds Number	Square Ribs Nusselt Number	Smooth ribs Nusselt Number	Wedge Ribs Nusselt Number	Funnel Ribs Nusselt Number
5000	52.2254	48.0012	57.1458	49.54
15000	97.558	92.7774	100.2547	93.884
30000	212.3314	206.114	215.6687	208.9997
45000	257.447	248.662	261.22554	249.2554
60000	315.324	308.221	320.225	309.22455



Plot 10 Comparision of Re vs Nu for All rib Types

5. Conclusion

The immergence of a recirculation zone just behind the ribs disturbs the thermal boundary layer which enhances the rate of heat transfer. Nusselt number enhances with reduction in the pitch and by increasing the height and width of the ribs, resulting augmentations in convective heat transfer. Friction factor decreases with increase in pitch and reducing the height and width of the ribs. Friction factor varies very little as compared to Nusselt number by increasing Reynolds number. From the Simulation Discussed in the Results and the Values obtained the designed Ribs named Square, Smooth, Wedge and Funnel at which having the wedge Ribs will sures Increase the Heat transfer.

References

[1] Aubin, J., et al., Modeling Turbulent Flow in Stirred Tanks with CFD: The Influence of the Modeling Approach, Turbulence Model and Numerical Scheme, Int. J. Experimental Thermal and Fluid Science, 2 (2004), 5, pp. 431-445

[2] Rahimi, M., et al., Experimental and CFD Studies on Heat Transfer and Friction Factor Characteristics of a Tube Equipped with Modified Twisted Tape Insert, Int. J.Chemical Engineering and Processing, 48 (2009), 3, pp. 762-770

[3] Bilen, K., et al., The Investigation of Groove Geometry Effect on Heat Transfer for Internally Grooved Tubes, Int.J.Applied Thermal Engineering, 29 (2009), 4, 753-761

[4] Kumar, S., Saini, R. P., CFD Based Performance Analysis of a Solar Air Heater Duct Provided with Artificial Roughness, Int.J.Renewable Energy, 34 (2009), 5, pp. 1285-1291

[5] Xiong, R., Chung, J. N., A New Model for Three-Dimensional Random Roughness Effect on Friction Factor and Heat Transfer in Micro Tubes, Int. J. Heat and Mass Transfer, 53 (2010), 15-16, pp. 3284-3291 [6] Craft, T. J., et al., Modeling of Three-Dimensional Jet Array Impingement and Heat Transfer on a Concave Surface, Int. J. Heat and Fluid Flow, 29 (2008), 3, pp. 687-702

[7] Iacovides, H., et al., Flow and Heat Transfer in Straight Cooling Passages with Inclined Ribs on Opposite Walls: an Experimental and Computational Study, Int.J. Experimental and Thermal Fluid Science, 27 (2003), 3, pp. 283-294

[8] Chaube, A., et al., Analysis of Heat Transfer Augmentation and Flow Characteristics Due to Rib Roughness over Absorber Plate of a Solar Air Heater, Int. J.Renewable Energy , 31 (2006), 3, pp. 317-331 [9] Karagoz, I., Kaya., F., CFD Investigation of the Flow and Heat Transfer Characteristics in a Tangential Inlet Cyclone, Int. J. Communications in Heat and Mass Transfer, 34 (2007), 9-10, pp. 1119-1126

[10] Rigby, G. D., Evans, G. M., CFD Simulation of Gas Dispersion Dynamics In Liquid Cross Flows, Int. J. Applied Mathematical Modeling, 22 (1998), 10, pp. 799-810.

[11] Li, L., et al., Numerical Study of Periodically Fully-Developed Convection in Channels with Periodically Grooved Part, Int. J. Heat and Mass Transfer, 51 (2008), 11-12, pp. 3057-3065

[12] Eiamsa-Ard, S., et al., 3D Numerical Simulation of Swirling Flow and Convective Heat Transfer in a Circular Tube Induced by Means of Loose Fit Twisted Tapes, Int. J. Communications in Heat Mass Transfer, 36 (2009), 9, pp. 947-955

[13] Zimparov, V., Prediction of Friction Factors and Heat Transfer Coefficient for Turbulent Flow of Corrugated Tubes Combined with Twisted Tape Inserts, Part 1 – Friction Factors, Int. J. Heat and Mass Transfer, 47 (2004), 3, pp. 589-599

[14] Zimparov, V., Prediction of Friction Factors and Heat Transfer Coefficient for Turbulent Flow of Corrugated Tubes Combined with Twisted Tape Inserts, Part 2 – Heat Transfer Coefficient, Int. J. of Heat and Mass Transfer, 47 (2004), 2, pp. 385-393

[15] Goto, M., et al., Condensation Heat Transfer of R410A Inside Internally Grooved Horizontal Tubes, Int. J. Refrigeration, 24 (2001), 7, pp. 628-638

[16] Goto, M., et al., Condensation Heat Transfer of R410A Inside Internally Grooved Horizontal Tubes, Int. J Refrigeration, 26 (2003), 4, pp. 410-416

[17] Promvonge, P., Thermal Enhancement in a Round Tube with Snail Entry and Coiled – Wire Inserts, Int. J. International Communications in Heat and Mass Transfer, 35 (2008), 5, pp. 623-629

[18] Promvonge, P., Thermal Augmentation in Circular Tube with Twisted Tape and Wire Coil Turbulators, Int. J. Energy Conversion and Management, 49 (2008), 11, pp. 2949-2955

[19] Chiu, Y-W., Jang., J.-Y., 3D Numerical and Experimental Analysis for Thermal-Hydraulic Characteristics of Air Flow Inside a Circular Tube with Different Tube Inserts, Int. J. Applied Thermal Engineering, 29 (2009), 2-3, pp. 250-258

[20] Zhang, X., et al., Heat Transfer Characteristics for Evaporation of R417A Flowing Inside Horizontal Smooth and Internally Grooved Tubes, Int. J. Energy Conversion and Management, 49 (2008), 6, pp. 1731-1739

[21] Li, X.-W., et al., Turbulent Flow and Heat Transfer in Discrete Double Inclined Ribs Tube, Int. J. Heat and Mass Transfer, 52 (2009), 3-4, pp. 962-970

[22] Karwa, R., et al., Heat Transfer Coefficient and Friction Factor Correlations for the Transitional Flow Regime in Rib Roughened Rectangular Ducts, Int. J. Heat and Mass Transfer, 61 (1999), 1, pp. 1597-1615 [23] Karwa, R., 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. J. Communication in Heat and Mass Transfer, 30 (2003), 2, pp. 241-250

[24] Tanda, G., Heat Transfer in Rectangular Channels with Transverse and V-Shaped Broken Ribs, Int. J. Heat and Mass Transfer, 47 (2004), 2, pp. 229-243

[25] Vicente, P. G., et al., Experimental Investigation on Heat and Frictional Characteristics of Spirally Corrugated Tubes in Turbulent Flow at Different Prandtl Numbers, Int. J. Heat and Mass Transfer, 47 (2004), 4, pp. 671-681

[26] Garcia, A., et al., Experimental Study of Heat Transfer Enhancement with Wire Coil Inserts in Laminar-Transition-Turbulent Regimes at Different Prandtl Numbers, Int. J. Heat Mass Transfer, 48 (2005), 21-22, pp. 4640-4651

[27] Wang, L., Sunden, B., Performance Comparison of Some Tube Inserts, Int. J. Communication in Heat and Mass Transfer, 29 (2002), 1, pp. 45-56

[28] Zimparov, V., Enhancement of Heat Transfer by a Combination of Three Start Spirally Corrugated Tubes with a Twisted Tape, Int. J. Heat Transfer, 44 (2001), 3, pp. 551-574

[29] Kothandaraman, C. P., Subramanyan, S., Heat and Mass Transfer data Book, New Age International Publishers, New Delhi, 2010 [30] ***, Fluent 6.3 ® April 7(2009)