CFD Based Heat Transfer Analysis of Discrete Arc Shaped Roughened Solar Air Heater For Different Relative Roughness Pitch

Shailesh Kumar Ram[1] Prof. Hemant KumarTandan[2], Prof. Dharmendra Rathore[3]

[1]M-Tech Research Scholar, [2]Research Guide and [3]HOD Deptt. of Mechanical Engineering

Bagula Mukhi College of Technology, Bhopal (M.P.)

Abstract- The present numerical study conducted on Computational Fluid Dynamics (CFD) using ANSYS 16.1 SOFTWARE in order to inspect the heat transfer characteristics of a solar air heater having rectangular duct roughened with discrete arc shaped rib roughness. Discrete arc shaped roughness is investigated with different relative roughness pitch (P/e) with an angle of attack 60^o corresponding to the flow direction. SIMPLE algorithm is used on the finite volume method for the computational numerical study of fluid flow; k–ε turbulence model is used for solving governing equations. Reynolds numbers ranging from 6000-14000 during the investigation of discrete arc shaped rib roughness. The different parameters used during present investigation are roughness height as 2mm, Hydraulic diameter (D_h) as 0.0461, relative roughness pitch (P/e) varies from 8 to 12(three values) and angle of attack (α) as 60^o. The roughness design should be selected to create the turbulence inside the duct with minimum use of energy to flow the fluid over the geometry. The discretisation used in arc shaped roughness improves the thermal performance of solar air heater with proper acceleration of flow. The enhancement in the value of Nusselt number has been reported with increase in Reynolds number. It is also found that the thermal performance varies with different relative roughness pitch. 3Diamentional CFD analysis is also compared with the experimental study of arc shaped roughness and found satisfying agreement of numerical results with the experimental results for investigated range of parameters.

Keywords: Solar air heater; artificial roughness; Turbulence; CFD; Thermal performance.

I. INTRODUCTION

Solar air heater is a thermal system use to increase the thermal characteristics of flowing air and deliver the air with moderate temperature for drying and heating purpose. Heat transfer rate between absorber plate and flowing air in solar air heater is low, hence an effort has been made to increase the heat transfer rate through heated absorber plate by using roughness geometries over the surface. Artificial roughness increases heat transfer because of developing turbulence to the viscous sub layer. It increases the turbulence in the flow and increases the re-attachment points. Many researchers have proposed and investigated various roughness geometries for the enhancement of heat transfer rate and proposed that creating artificial roughness on the absorber plate could substantially enhance the heat transfer of solar air heater. Prasad and Mullick [1] was first introduced transverse shape of roughness to improve the heat transfer characteristics of solar air heater. Prasad and Saini [2] and Kays [3] uses small diameter wires and investigated for the different operating and geometrical parameters. Experimental investigation over transverse rib roughness was studied by Gupta et al. [4] using different operating parameters. Broken type of transverse rib was investigated by Sahu & Bhagoria [5] to analyse the heat transfer characteristics of solar air heater. Momin et al. [6] investigated the effect of V-shaped roughness on heat transfer characteristics of solar air heater duct. It was found that V-shape rib roughness gives better performance with an angle of attack of 60^o. Lanjewar et al. [7] uses W shaped rib roughness which increases the number of secondary flow developed due to multi inclination of the geometry having 60^o angle of attack. Enhancement in Nusselt number is reported as compare to smooth plate. The concept of discretising the W-shaped of geometry was investigated by A. Kumar et al. [8] and the effect of operating parameters are compared with the result of smooth surface under similar operating conditions. Performance of solar air heater using arc shaped of roughness was investigated by Saini et al. [9] with relative roughness pitch (P/e) of 10. Further CFD analysis of solar air heater with arc shaped roughness geometry was studied by Kumar et al. [10]. 2D investigation of artificial roughness using CFD in a solar air heater is carried out by Yadav et al. [11]. They use square sectioned transverse rib roughness in a rectangular duct is using ANSYS FLUENT 12.1 software with RNG k-ε turbulence model. Using square sectioned transverse ribs with different values of relative roughness pitch investigated using CFD is conducted by Yadav et al. [12] on the solar air heater. Yadav et al. [13] predicts the effect of roughness having transverse wire on heat transfer enhancement in solar air heater. In this analysis also FLUENT 12.1 software is used with the RNG k–ε turbulence model. 2D CFD investigation using different ribs as rectangular, square, chamfered, triangular etc. is performed by Chaube et al. [14]. The heat flux of 1100 W/m^2 is provided on the absorber plate and rib, rest of the surrounding surfaces is kept adiabatic. Karmare et al. [15, 16] has done experimental as well as numerical study and analyze the effect of heat transfer performance of a rectangular duct with metal grit ribs as roughness. A 3D numerical analysis has been performed using CFD on solar air heater roughened with metal ribs of circular, square and triangular cross-section.

Number of researches has been done experimentally to analyze the performance of different geometries on the performance of solar air heaters, but experimentation needs a lot of time to analyze even a single parameter and it also involves so much cost. Hence present study involves CFD analysis using ANSYS SOFTWARE 16.1 with roughness element in form of discrete arc shaped ribs for different range of parameter. The heat transfer and fluid flow analysis is performed using 3-D model of discrete arc shaped roughness. The ribs are provided on the one side of rectangular duct whereas other sides are kept smooth. In order to validate the CFD results, the obtained data is compared with the experimental results of arc shaped roughness obtained from the correlation developed by Kumar et al. [10].

II. COMPUTATIONAL DOMAIN AND CFD ANALYSIS

The computational domain used in CFD analysis is designed in such a way that the dimensions of domain must be similar to the dimensions used in experimental test rig of Kumar et al. [10]. As per the experimental investigation the dimensions used for computational domain in CFD analysis as the height (H) of domain is 25mm, width (W) 300mm and length of test section is 1000mm as shown in Fig 1.

As per the recommendation provided in ASHRAE Standard 93-77 the inlet section should be provided to make the flow fully developed. The dimensions of roughness element and operating parameters are shown in Table.1.

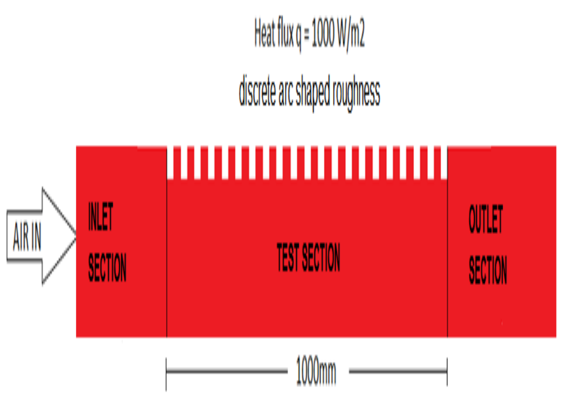

Fig.1. Solution domain of W-rib roughness

Table.1. Operating dimensions to formulate the computational domain

PARAMETERS	VALUE
Duct Height (H)	25mm
Duct Width (W)	300mm
Hydraulic mean diameter (D_h)	46.1 mm
Duct aspect ratio (W/H)	12:1
Length of Test Section	1000mm
Rib height (e)	2mm
Relative roughness pitch (P/e)	08,10,12
Angle of attack (α)	60°
Reynolds number (Re)	4000-14000
Uniform Heat at bottom Surface	1000W/m^2

Table.2. Thermo-physical properties

PROPERTIES	WORKING FLUID	ABSORBER SURFACE (aluminium)
Density (ρ) Kg/m^3	1.225	2719
Specific heat (Cp) J/Kg.K	1006.43	871
Viscosity (μ) N/m^2	1.789 x 10^{-5}	--
Thermal Conductivity (K) W/m.K	0.0242	202.4

2.1. The assumptions made to solve the mathematical model while CFD analyses

In order to analyze the artificial roughness domain using CFD following assumption has been considered

i. Steady Flow
ii. Incompressible flow
iii. Two-dimensional flow.
iv. Constant thermo physical properties of the fluid.
v. Neglecting conduction resistance of the heated absorber plate.
vi. Neglecting viscous dissipation in the energy equation as significance of viscous dissipation only for flows at high velocities.

1.2. Boundary condition

The rectangular computational domain of solar air heater is divided into three sections as inlet, test and exit sections. Air is used as a working fluid solar air heater duct and the absorber plate is made up of aluminum. The thermo physical properties of flowing fluid and absorber plate material are shown in Table 2. A constant heat flux of 1000 W/m^2 is generated over the top wall of test section. Uniform velocity is taken as the inlet boundary condition for the inlet section and constant pressure (1.013 X 10^5 Pa or 0 gauge) is considered as outlet boundary condition applied at the exit. The upper roughened surface of domain is maintained at no-slip boundary condition and all the surrounding wall is maintained as adiabatic.

Table.3. Boundary conditions

LOCATION	BOUNDARY TYPE	BOUNDARY DETAIL	
Air inlet	Velocity inlet	Velocity	Reynolds number
		1.745455	6000
		2.327273	8000
		3.490909	12000
		4.072727	14000
Outlet	Pressure outlet	Gauge Pressure 0	
Left side, Right side, Bottom	Wall	No slip, Adiabatic insulated wall	
Absorber plate	Wall	Heat flux = 1000 W/m^2 Material = Alluminium	

2.3. *Grid/ Mesh Generation*

ANSYS 16.1 SOFTWARE is used for creating non-uniform tetrahedral meshing module on the computational domain of solar air heater. A very fine mesh size is used by increasing the number of nodes and element at the boundary of smooth rectangular solar air heater duct to resolve the laminar sub layer. Different numbers of cells are provided to obtain the mesh. The grid independence test is carried out by varying elements from 381550 to 577102 in three steps for Re = 14,000, P/e = 10 and e/D = 0.0461. A marginal variation is obtained in heat transfer coefficient when the number of cells varies from 350000 cells to 500000. Hence, increasing the number of cells beyond this value has negligible effect on the results.

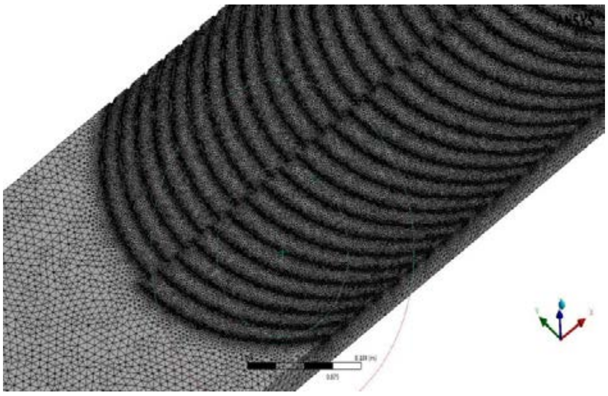

Fig.2. Grid generation over the computational domain

2.4. *Method of solving the domain*

The CFD analysis of solar air heater duct with discrete arc shaped artificial roughness is carried out using CFD code ANSYS FLUENT 16.1. The steps involving for the numerical simulation using CFD in solar air heater is as follows:

- ✓ Developing model
- ✓ Computational domain
- ✓ Create 3-D model of the domain
- ✓ Generating mesh
- ✓ Set up and compute the solution
- ✓ Examine the results

Initial step is to develop the model, in present study 3-D discrete arc shaped model is selected to analyze because it is time saving and obtained results exact to the experimental testing as compare to the 2-D model. A 3-D computational domain is prepared like as shown in Fig.1. Non-uniform grid or mesh is generated over the domain in order to capture the gradients by using CAD model. Computational domain is prepared in ANSYS modeler 16.1. The properties of Material, operating conditions, boundary conditions, solver controls and convergence monitors are set up in solver. Finite volume method is used to discretize the governing equations of continuity; momentum and energy. RNG K-ε turbulence model is used to solve the flow equations.

III. RESULTS AND DISCUSSION

3.1 *Validation of setup*

For the validate the numerical solution, the computational analysis have been carried out for forced convection in a smooth rectangular duct having fully developed turbulent flow and it is compared with the exact solution obtained with the Dittus Boelter equation. In CFD analysis, Nusselt number for smooth duct (Nu_s) of a solar air heater can be obtained by the Dittus–Bolter

equation (McAdams, 1942),

$$Nu_s = 0.024\ Re^{0.8}\ Pr^{0.4}$$

It is obtained from the results that the present study of CFD analysis over smooth duct has good agreement with the exact solution for the Nusselt number

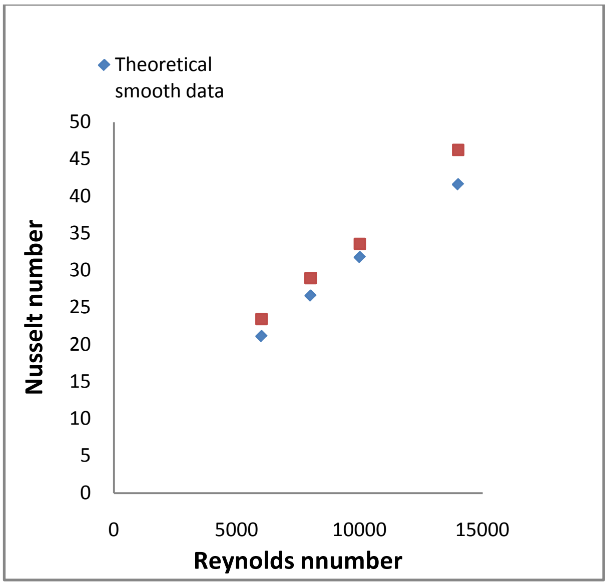

Fig.2. Comparison of computational and formulated value of Nusselt no. vs Reynolds no. for smooth plate

In order to validating the accuracy of numerical solutions CFD analysis of present roughness geometry i.e. discrete arc shaped is also compared with the experimental results obtained from the correlations developed by Kumar et al. [10] under similar parametric and operating condition. Less than ± 10% deviation is obtained between the numerical and experimental results as shown in Fig .3.

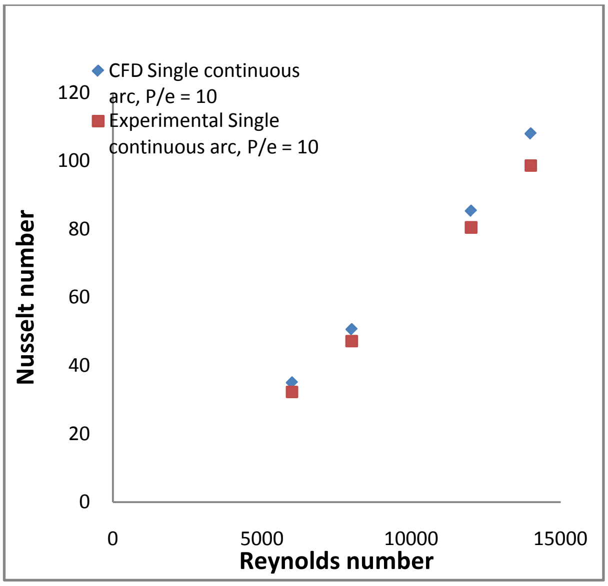

Fig.3. Comparison of computational and experimental results of Nusselt no. vs Reynolds no.

3.2. Heat transfer in roughened solar air heater duct

Providing artificial roughness on the rectangular duct of solar air heater affects the heat transfer and fluid flow characteristics. Nusselt number has been found to be low just at the vicinity of rib because heat transfer takes place at that rib area is due to conduction only, while at the point where the flow get reattached with the heated surface Nusselt Number will get increased. The increase in the value of Nusselt number is due to the variation in flow pattern. The presence of roughness along the flow direction creates turbulence and the fluid flow also separates from the wall. Heat transfer decreases with the separation of flow whereas vortices generated because of turbulence makes the fluid to mix and hence increasing heat transfer. Temperature profile of arc shaped rib roughness and discrete arc shaped rib roughness is shown in the Fig.4 and 5.

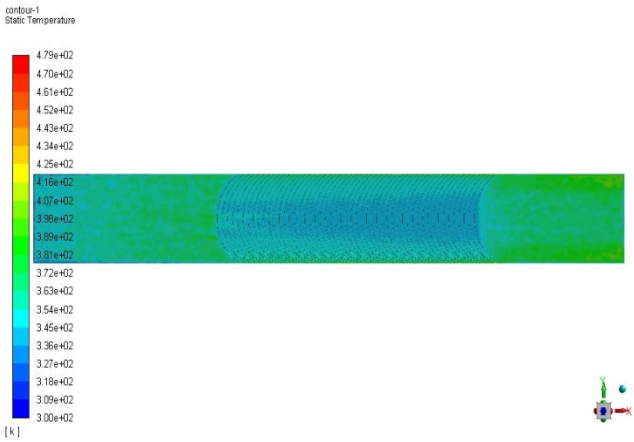

Fig.4. Temperature contour of computational domains of Arc shaped roughness

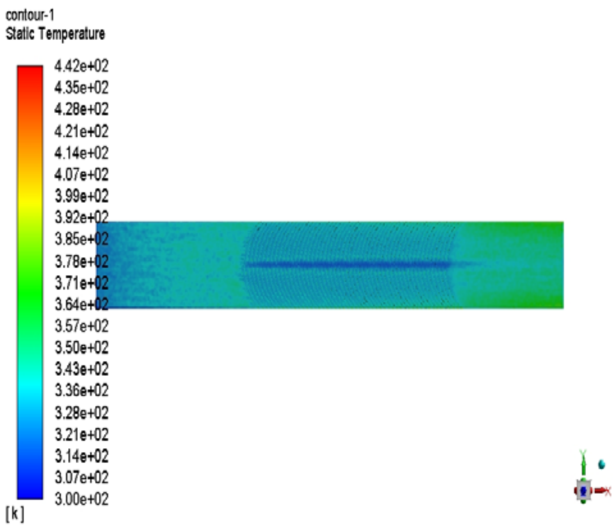

Fig.5. Temperature contour of computational domain of discrete arc shaped roughness

2.4. Comparison with earlier reported roughness

The enhancement in the value of Nusselt number by using Discrete arc shaped roughness is found to be more than that of earlier reported Arc shaped roughness geometry as well as smooth surface simulated under similar parametric and operating conditions as shown in Fig.6. This is because of more flow separation over the roughness and discretization accelerates the flow in discrete arc shaped roughness as compare to the arc shaped roughness. Heat transfer and Nusselt number increase with the increase in turbulence which is affected intensity of flow as well as flow separation provided in the geometry which is more in discrete arc shaped rib roughness as compared to arc shaped rib roughness.

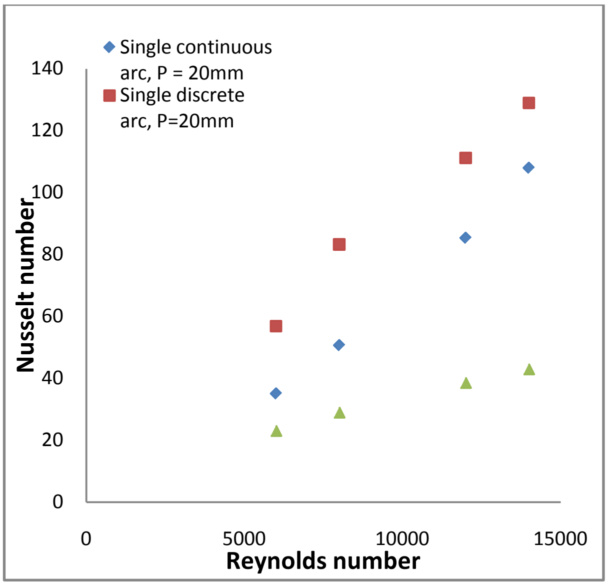

Fig.6. Comparison of Nusselt number Vs Reynolds number for W-rib and V-rib roughness

2.5. Comparison between various ribs at different relative roughness pitch

The variation in the value of Nusselt number for different relative roughness pitch (P/e) of Discrete arc shaped roughness has been displayed in graph for the range of Reynolds number varies from 6000-14000 shown in Fig.10. It is observed from the analysis that for any particular value of Reynolds number, Nusselt number attains maximum value corresponding to relative roughness pitch (P/e) value of 10 and on increasing this value as 12 or decreasing the relative roughness pitch as 8 decreases in Nusselt number has been observed. It is due to the fact that flow separation occurs on a rib at relative roughness pitch (P/e) as 12 decreases and the reattachment points also decreases with the heated surface because of less number of elements. It is observed that the nusselt number decreases on either side of relative roughness pitch (P/e) of 10. By decreasing the relative roughness pitch (P/e) as 8 decreases the exposed heated surface to the air and also decrease the reattachment of air with the heated surface due to lower value of pitch. The temperature contour of Discrete arc shaped roughness having relative roughness pitch (P/e) as 8, 10 and 12 is shown in Fig.7, 8 and 9.

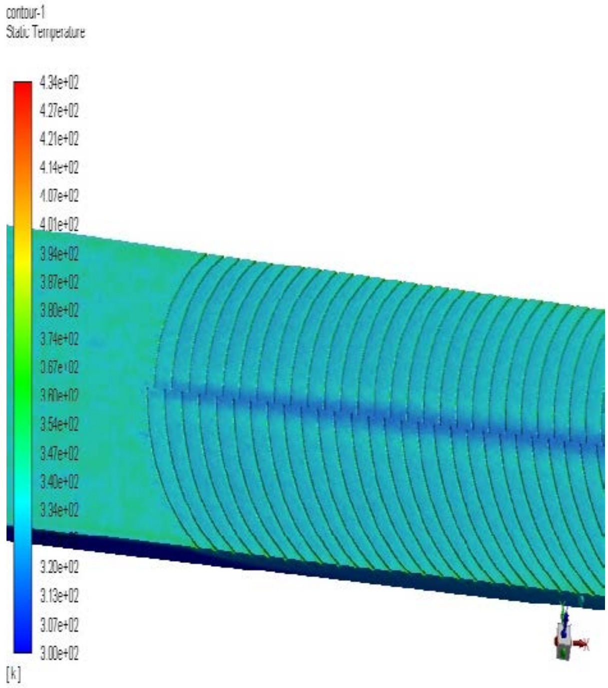

Fig.7. Temperature contour of Discrete arc shaped roughness having P/e = 8

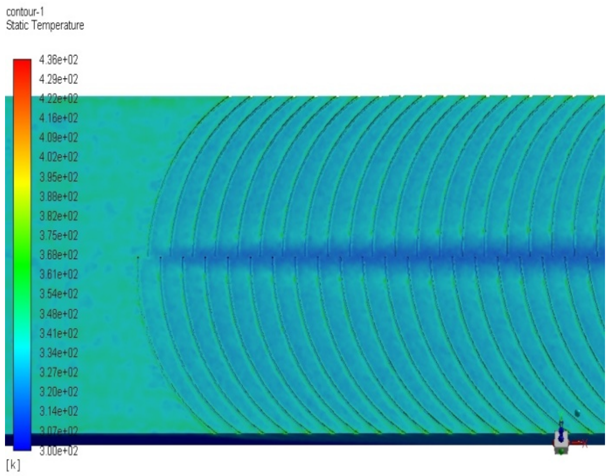

Fig.8. Temperature contour of Discrete arc shaped roughness having P/e = 10

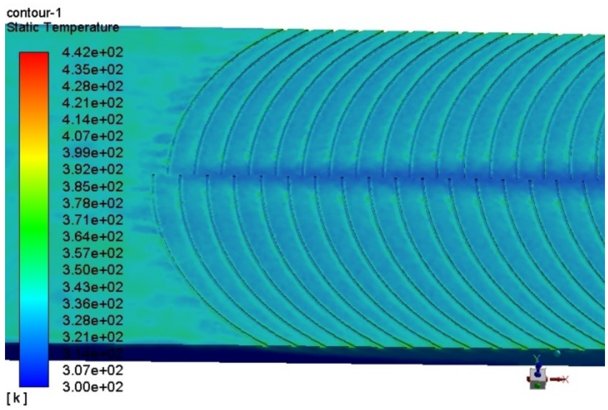

Fig.9. Temperature contour of Discrete arc shaped roughness having P/e = 12

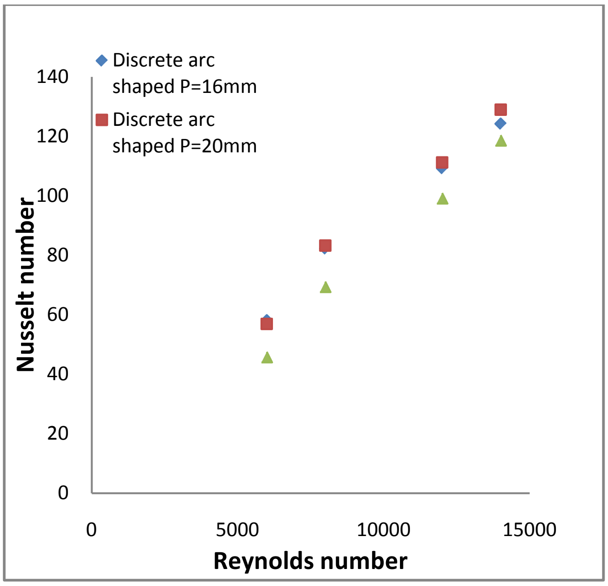

Fig.10. Comparative graph of W-rib roughness with different relative roughness pitch

The present numerical study of using Discrete arc shaped roughness with different relative roughness pitch (P/e) in a rectangular duct of solar air heater shows that the Discrete arc shaped rib roughness having relative roughness pitch (P/e) of 10 gives better performance as compare to the Single arc shaped roughness under similar operating conditions as well as the smooth rectangular duct of solar air heater. The overall comparative chart of Discrete arc shaped roughness is shown in Fig.11.

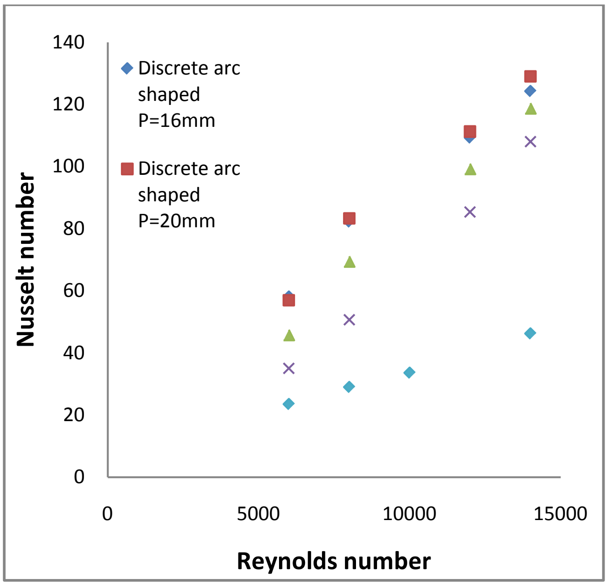

Fig.10. Comparative chart of Discrete and Single arc shaped roughness

IV. CONCLUSION

The present numerical study using CFD carried out by using discrete arc shaped roughened solar air heater to evaluate fluid flow and heat transfer characteristics. It shows that the CFD results found in better agreement with experimental results. As we increase the Reynolds number the thickness of boundary layer reduces which improves the convective heat transfer between the heated plate and the air by decreasing the convective resistance, it results increase in Nusselt number. Discrete arc shaped roughness gives better heat transfer rate as compare to the single arc shaped roughness because of more separation of flow in discrete arc shaped roughness under similar operating conditions. The maximum value of Nusselt number for discrete arc shaped roughness is obtained with the relative roughness pitch (P/e) of 10. It starts decreasing on either side of the relative roughness pitch of 10. The maximum increment in Nusselt number is found to be 2.79 times as compare to the smooth surface at Re = 14000. The maximum increment in Nusselt number is found to be 1.19 times as compare to the arc shaped roughness at Re = 14000.

REFERENCES

[1] Prasad K, Mullick SC. Heat transfer characteristics of a solar air heater used for drying purposes. Applied Energy 1983;13(2):83–93

[2] Prasad BN, Saini JS. Effect of artificial roughness on heat transfer and friction factor in a solar air heater. Sol Energy 1988; 41:555–60.

[3] Kays WB. Convective Heat and Mass Transfer. Mc Graw Hill Book Co., New York, 1966; pp. 197–198.

[4] Gupta D, Solanki SC, Saini JS. Heat and fluid flow in rectangular solar air heater ducts having transverse rib roughness on absorber plates. Sol Energy 1993; 51 (1):31–7.

[5] Sahu M M, Bhagoria J L. Augmentation of heat transfer coefficient by using 90° broken transverse ribs on absorber plate of solar air heater. Renewable Energy 2005; 2063–2075.

[6] Momin AME, Saini JS, Solanki SC. Heat transfer and friction in solar air heater duct with v-shaped rib roughness on absorber plate. Int J Heat Mass Transf 2002;45:3383–96.

[7] Lanjewar A, Bhagoria JL, Sarviya RM. Experimental study of augmented heat transfer and friction in solar air heater with different orientations of w-rib roughness. Exp Therm Fluid Sci 2011;35:986–95.

[8] Kumar A, Bhagoria J L, Sarviya R M, 2008. In: International 19th National & 8th ISHMT-ASME Heat and Mass Transfer Conference Heat Transfer Enhancement in Channel of Solar Air Collector by using Discrete W-shaped Artificial Roughened Absorber.

[9] Saini S K, Saini R P. Development of correlations for Nusselt number and friction factor for solar air heater with roughened duct having arc-shaped wire as artificial roughness. Sol. Energy 2008; 82, 1118– 3110.

[10] Kumar S, Saini RP. CFD based performance analysis of a solar air heater duct provided with artificial roughness. Renew Energy 2009; 34:1285–91.

[11] Yadav AS, Bhagoria JL. Modeling and simulation of turbulent flows through a solar air heater having square-sectioned transverse rib roughness on the absorber plate. Sci World J 2013.

[12] Yadav AS, Bhagoria JL. Numerical investigation of flow through an artificially roughened solar air heater. Int J Ambient Energy 2013.

[13] Yadav AS, Bhagoria JL. A CFD (computational fluid dynamics) based heat transfer and fluid flow analysis of a solar air heater provided with circular transverse wire rib roughness on the absorber plate. Energy 2013;55:1127–42.

[14] Chaube A, Sahoo PK, Solanki SC. Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater. Renew Energy 2006;31(3):317–31.

[15] Karmare SV, Tikekar AN. Heat transfer and friction factor correlation for artificially roughened duct with metal grit ribs. Int J Heat Mass Transf 2007;50:4342–51.

[16] Karmare SV, Tikekar AN. Analysis of fluid flow and heat transfer in a rib grit roughened surface solar air heater using CFD. Sol Energy 2010;84(3):409–17.