Computational Investigation for Thermal Performance of Solar Air Heater for Dimple Shape Roughness

Sanjeev Kumar Joshi[1] Saurabh Aggarwal [2] Gaurav Kumar[3]
Uttaranchal University, Dehradun
ersanjeevjoshi@gmail.com

***Abstract* -** In this present study, computational 2D studies are carried out to investigate the Thermal performance of solar air heater duct and pressure drop characteristics for different flow problems. This study focus on rectangular duct with dimple shape rib roughness inserted in solar air heater. The dimples are oriented in one side of a wall and other wall of a duct is insulated, the fluids taken were air. The dimple height to duct hydraulic diameter ratio varies from 0.03 to 0.06 for different dimple height; the ratio of relative roughness pitch varies from 6 to 18, and Reynolds number varies from 6,000 to 18,000.Analysis is done using ANSYS FLUENT.

I. INTRODUCTON

The heat transfer enhancement is essential to converse and recovery of energy and heat transfer enhancement technique in order to obtain more compact and efficient heat exchangers. Heat exchangers are needed in many industrial fields. In order to reduce the energy consumption and increase economic benefit, high-performance heat transfer components for thermodynamic process are become more and more popular for many applications. Some examples are steam generation and condensation in power and cogeneration plants; sensible heating and cooling of viscous media in thermal processing of chemical, pharmaceutical, and agricultural products; refrigerant evaporation and condensation in air conditioning and refrigeration; gas flow heating in manufacturing and waste-heat recovery; and cooling of electrical machines and electronic devices. Improved heat exchange can significantly improve the thermal efficiency in such applications as well as the economics of their design and operation. The need to increase the thermal performance of heat exchangers, thereby effecting energy, material, and cost savings as well as a consequential mitigation of environmental degradation had led to the development and use of many heat transfer enhancement techniques.

A. Solar air heater

Solar air heater is the simplest device which is used to convert the solar energy into heat energy In solar air heater heat generated by solar energy is collected over a collector and that heat is then taken away by the fluid flowing i.e. air in the duct of solar air heater The heat carried away by air is then used for various purposes and in many applications such as crop drying space heating The efficiency of solar air heater is low due to low convective heat transfer between the absorber plate and the fluid flowing inside the duct So to increase the thermal efficiency of solar air heater many investigators put forth their views Several methods have been used by various investigators to increase efficiency. Some of these are Use of artificial roughness on absorber plate use of fins electro hydrodynamic method packed bed etc.

B. Heat transfer through dimple roughness

Heat transfer is enhanced by the different types of artificial roughness using some enhancement techniques and the dimples are used as passive techniques. The dimple Roughness enhancement method is one of the most effective ways to improve the heat transfer performance with small increase of pressure drop which are smaller than other types of heat augmentation devices, such as rib tabulators.

C. Effect on the thermo hydraulic performance

The thermal efficiency of solar air heater having smooth plate collector is very low due to low convective heat transfer coefficient between the absorber and the air flowing in the duct The use of artificial roughness on underside of absorber plate surface is an effective way to enhance heat transfer to the flowing air in the duct, expense of pressure drop. Several investigators have investigated that artificial roughness is provided by fixing wires, ribs, wire mesh or expanded metal mesh by roughness in shaped geometry. In a smooth plate air heater a thin sub layer develops adjacent to the wall turbulent layers where velocity is relative low. In this region heat transfer pre-dominated by conduction and beyond this heat transfer process is dominated by convection. Thus the objective is to increase the heat transfer between the absorber plate air flowing over the plate. Providing element improves thermal efficiency but it would result in frictional losses.

II. METHODOLOGY

In the present work numerical simulation has been carried out to investigate the effect of dimple shape roughness. The working fluid is air flowing through a constant heat flux rectangular duct with the dimple under the top surface of the duct contain heat flux while the bottom surface is insulated. Analysis type 2-Dimensional CFD study of fluid flow and heat transfer enhance characteristics in a rectangular duct in flow direction. A 2-Dimensional CFD study of fluid flow and heat transfer characteristics in a rectangular duct with dimple of increasing height in flow direction mounted on top surfaces. Here, the selection of suitable turbulence model has been made on the basis of the literature survey. The computational processes in solving the simulation using the ANSYS-CFD are presented in the following subsections.

A. Physical modelling

The dimple configuration (Fig. 3.1) and physical model that has been used in current analysis mainly consists of a long rectangular duct with inlet, test section and outlet. For a constant property fluid flowing with constant cross section area the velocity profile becomes independent on the stream-wise flow at some distance from the inlet. The flow will, after an entry length repeats itself in a periodic manner within each rib interval. The channel height has a direct influence in the analysis. The domain is halved and a symmetry boundary condition is employed to simplify of the analysis. Since the analysis is two dimensional, the channel width has no direct influence in the analysis. But the mass flow rate is calculated only on the basis of hydraulic diameter and hence the channel width is taken into consideration. Uniform heat wall 1200 w/m^2 of rectangular duct of 640mm length for turbulent flow with Reynolds number range (6000, 9000, 12000, 15000, 18,000) and relative roughness pitch p/e (6, 10, 14, 18) has been carried out with k-e turbulence model is selected by computational fluid dynamics software (Fluent 14.0 Solver).

B. Parameters range

The top walls of the test section are provided with constant heat flux 1200 w/m^2 and remaining walls are adiabatic. A number of dimples are mounted on the heated top and bottom wall of the duct to improve heat transfer coefficient between the hot walls and air. the total length of the rectangular duct is 640mm, entrance length is 245mm and exit length is 115mm. The range of the Reynolds number is 6000, 9000, 12000, 15000, 18000. The flow under consideration is expected to attain a fully-developed flow condition. Air enters the rectangular duct at an inlet temperature, T_{in} of 300K. The rectangular duct is 640 mm long and the cross-sectional area of the rectangular duct is 20mm × 100mm. The hydraulic diameter (D_h) of the duct is 33.33mm.

C. Governing equation

The governing equation have been developed for the given flow system under three distinct flow condition namely steady-laminar unsteady-laminar and turbulent flow. In view of the above , the section focuses on conservation equation pertaining to laminar and turbulent flow condition.

D. Mesh generation

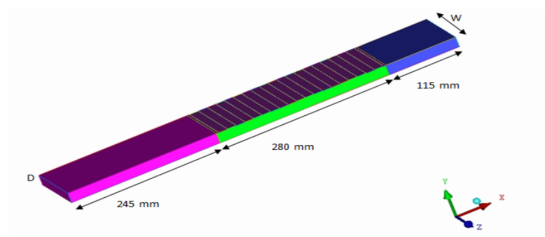
Mesh generation is one the most critical aspects of engineering simulation' Too many cells may result in long solver runs and too few may lead to in accurate results ANSYS Meshing technology provides a means to balance these requirements and obtain the right mesh for each simulation in the most automated way possible ANSYS Meshing technology has been built on the strengths of stand-alone class-leading meshing tools The strongest aspects of these separate tools have been brought together in a single environment to produce some of the most powerful meshing available.

E. Grid independence test

For grid independence test, a solution adaptive refinement method is used. When adaption is employed appropriately the resulting mesh is most favourable for the flow solution. Thus, to avoid wastages of the computational resources the addition of unnecessary cells was stopped. As shown in table 3.4 after 78390 numbers of nodes, the variation in the value of heat transfer is negligible. Hence, the further analysis using meshes with 78390 numbers of nodes.

F. Validation

The present numerical method and model is validated with respect to the result of numerical study carried out by Yadav and J.L.Bhagoria. The numerical data is used to find out the heat transfer from two-dimensional geometric modeling of rectangular duct with slit-ribs of 640mm×100mm×20mm dimensions have been observed and the results of Nusselt numbers and friction factors have been obtained in the variation of Reynolds numbers (6000, 9000, 12000, 15000, 18000). Results obtained from numerical investigation are plotted as a function of Nusselt number and Reynolds number as shown in Fig 3.3. Numerical results are compared with the experimental results and found deviation of ±10% which is considerable for numerical results obtained by using CFD (fluent) analysis



III. RESULT AND DISSCUSSION

The computational fluid dynamic analysis has been performed for artificial roughened solar air heater with dimple shaped roughness rib roughness on the absorber plate in this chapter the result of computational fluid dynamic analysis has been discussed. The results have been compared with the result of smooth duct operated under same condition To discuss the enhancement in heat transfer on account of artificial roughness. In the present investigation of the project heat transfer enhancement in two dimensional rectangular duct flows over dimple shape has been completed using ANSYS WORKBENCH 14.0. The heat transfer enhancement and friction characteristics of the rectangular ducts with or without dimple shape discussed with the help of heat transfer, nusselt number and friction factor. In ansys we using a hydraulic diameter D_h , density ρ, height of pitch e, roughness ratio, pitch to rib height ratio,viscosity, and Reynold number these value.

All the calculations were carried out at 6000, 9000, 12000, 15000, 18000, Reynolds number. Boundary conditions were defining with appropriate material property in fluent soft ware. In the solver, all flows were specified as steady state and incompressible. The realizable k-ε turbulence model with standard wall function was set for all models for turbulent flow.

Calculations are presented for seven different result in Re =6000, 9000, 12000, 15000, 18000, spread over the laminar and transitional flow regimes in a smooth channel. First, the dominant flow structure is presented in terms of instantaneous and mean coherent velocity distributions. This is then related to the wall shear stress and heat transfer augmentation.

Fig. 4.1 shows the comparison between the results of the smooth duct and dittous-boelter equation correlation. The maximum Nusselt numbers are 49.63 and 50.750 for respectively smooth duct and dittous-boelter equation correlation and Reynolds number in the range of 6000, 9000, 12000, 15000, 18000.

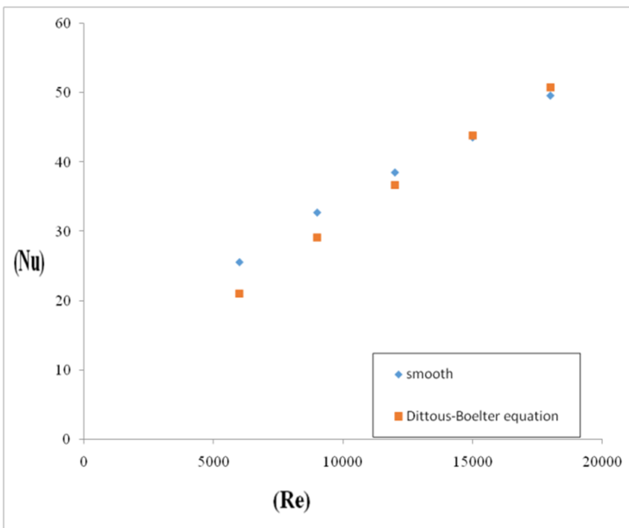


Fig.4.1. Variation of Nusselt number versus position at different Reynolds number for smooth duct.

The figure 4.2 shows the comparison of Nusselt number between the duct with different relative roughness slit-ribs pitch p/e = 6, p/e = 10, p/e = 14, p/e = 18 and smooth duct at the Reynolds number 6000-18000 In fig.4.2

We have been observed that the Nusselt number was increased with relative roughness pitch (p/e) increasing and maximum Nusselt number was observed at p/e = 18, reynold number 18000 and Nusselt number in smooth duct is lowest because no disruption in smooth duct.

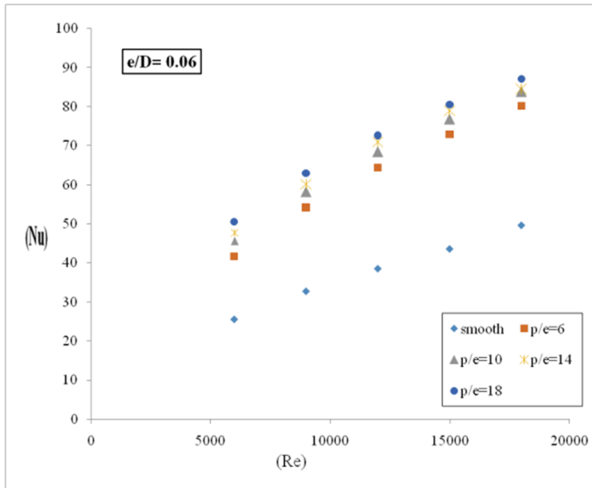
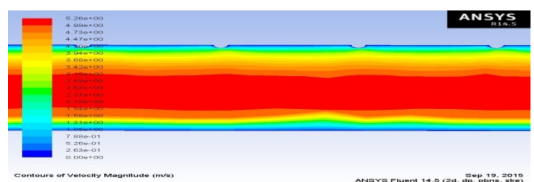


Fig.4.2. Comparison of Nusselt number for smooth duct with respect to the numerical investigation.

A. Velocity contours

Fig. 4.3 shows the mean streamline distribution at different Reynolds numbers in the center plane of the dimple. At the lowest Reynolds number, Re = 6000 and p/e = 6, there is no flow separation and recirculation in the dimples, whereas in all the other cases the flow in the dimple is characterized by a recirculation zone within the dimple. The flow separates at the leading edge of the dimple and the separated shear layer is drawn into the dimple cavity where it reattaches downstream. As the Reynolds number increases, Heat transfer is highest at the downstream rim of the dimples where the flow reattaches.



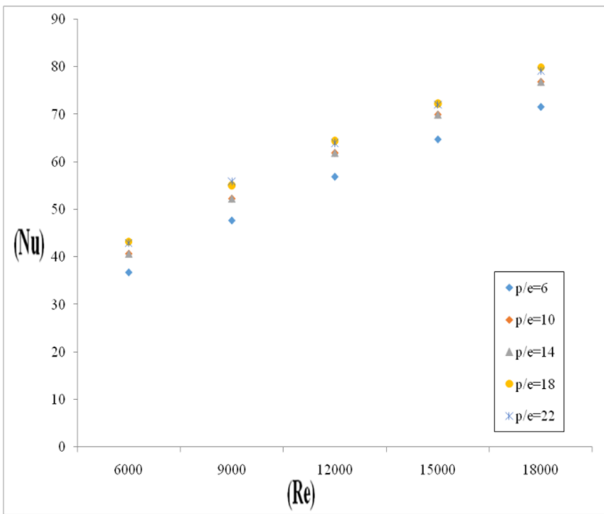
Variation of velocity magnitude aruond the slit-ribs at Re = 6000 for roughness angles p/e = 6.

B. Effect of relative roughness pitch (p/e)

In this numerical investigation, the computation analysis is observed in a rectangular duct with dimples. The dimples have a configuration of height of dimple is 1mm, 1.64mm, and 2mm e/D_h = 0.03, 0.049, and 0.06. The analysis is calculated for a constant heat flux 1200 w/m^2 is applied at top wall of the rectangular duct for The Reynolds number range is 6000,9000, 12000, 15000, 18000 and the velocities corresponding to the Reynolds number is 2.655881 m/s to 7.967644 m/s. The variations in temperature contour in have been shown in fig.4.7-4.9. The variation in the relative roughness pitch is p/e = 6, 10, 14, and 18mm according to this variation in the Nusselt number with Reynolds number as shown in figures 4.11 - 4.12 which explain the results of the Nusselt number at variation in the relative roughness pitch distance.

C. Effect of Reynolds number on fluid flow and heat transfer characteristics

The different Reynolds number (Re = 6000, 9000, 12000, 15000, 18000) with same constant heat flux conditions of top surface are investigated. For the simplicity, only the region around the dimple. The variation of Nu number of duct with dimple at different Reynolds number is shown in Fig.4.10-4.12. From the figure, it is observed that The Nu gradually increases and reached its maximum value that corresponds to flow reaches the reattachment point of the main recirculation vortex and the maximum nusselt number (nu = 86.06674) is observed at the Re=18000. Further, Nu decreases because of growth in the boundary layer. The variation of heat transfer coefficient is observed again when the flow arrives at the next dimple. From the Fig 4.10 - 4.12, it is also observed that Nusselt number increases with increase in Reynolds number.



Variation of Nusselt number with Reynolds number (6000 - 18000) for different pitch roughness (p/e = 6, 10, 14, 18, 22) at e/D = 0.03.

IV. CONCLUSION

A significant amount of research activity in numerical investigation way has been carried out during the past few decades which concisely devotes to the understanding of heat transfer and fluid flow through ducts with dimple. In this present examination, numerical expectation has been conducted to study heat transfer practices of a dimple shaped roughness rib duct of a solar air heater having triangular rib roughness on the absorber plate. The effect of Reynolds number (6000, 9000, 12000, 15000, and 18000), relative roughness pitch (p/e = 6, 10, 14, and 18), and relative roughness height ratio (e/D= 0.03, 0.049, 0.06) on the heat transfer and friction factor have been studied and which are discussed with the help of Nusselt number and friction factor as plots. In the work the k-e model has been taken to simulate turbulent flow in the rectangular duct.

The main conclusions can be summarized as follows:

From the results, it is clear that k-ε standard model could provide results with acceptable engineering accuracy for the analysis of flow and heat transfer patterns in dimple shape.

1. The heat transfer increases and friction factor decreases with the increasing in the value of Reynolds number.
2. The Nusselt number increases with the relative roughness pitch (p/e) increases.
3. The highest Nusselt number (Nu=87.066) is obtained at roughness height e/D=0.06, Reynolds number Re = 18000 and pitch ratio p/e = 18 which has been discussed in plots.
4. The lowest friction factor is obtained at the value of roughness height e/D=0.06, p/e = 6 and Re = 6000.
5. highest hydraulic thermal performence (THP = 2.0061) was observed at Re=18000 where p/e=18.

V. REFERENCES

[1] Singh A.P., Varun, Siddhartha (2014), "Effect of artificial roughness on heat transfer and friction characteristics having multiple arc shaped roughness element on the absorber plate", Solar Energy Vol.i105,i pp. 479i –i 493.i

[2] Kumar A.,i Saini R.P.,i Saini J.S.,i (2014),i "Ai review of thermohydraulici performance of artificially roughened solar air heaters",i Renewable andi Sustainable Energy Reviews Vol. 37,i pp. 100i –i122.i

[3] Prasad B.N., Aruni K. Behura,i Prasad L.,i (2014),i "Fluid flows andi heat transfer analysis for heat transfer enhancement in three sided artificially roughened solar air heater",i Solar Energy Vol. 105,i pp.i27i –i35.i

[4] Alam T Saini R.P, Saini J.S.(2014), "Effect of circularity of perforation holes in V-shaped blockages on heat transfer and friction characteristics of rectangular solar air heater duct",i Energy Conversion and Management Vol.86,i pp. 952i –i 963.i

[5] Bekele A., Mishra M., Dutta S., (2014),i "Performance characteristics of solar air heater with surface mounted obstacles",i Energy Conversion and Management,Vol.i 85,i pp.603–611.i

[6] Yadavi A.S., Bhagoriai J.L., (2014), "Ai CFD based thermo-hydraulic performance analysis of an artificially roughened solar air heater having equilateral triangular sectioned rib roughness on the absorber plate, International Journal of Heat and Mass Transfer, Vol. 70,i pp. 1016i –i1039.i

[7] Alam T., Sain R.P., Sain J.S., (2014), "Experimental investigation on heat transfer enhancement due to V-shaped perforated blocks in ai rectangular duct of solar air heater", Energy Conversion and Management, Vol.81,pp.374–i 383.i

[8] Yadav A.S., Bhagoria J.L., (2014), "Ai numerical investigation of square sectioned transverse rib

roughened solar air heater", International Journal of Thermal Sciences, Vol. 79,i pp. 111i – 131.i

Sanjeev Kumar Joshi has done B.E in Mechanical Engineering (Regional Engineering College Silchar) and M.Tech in Thermal Engineering. Presently he is working as a Assistant Professor in Uttaranchal University, Dehradun.His research interests include heat transfer enhancement techniques; solar energy utilization etc.

SaurabhAggarwal received his B.Tech degree in Mechanical Engineering and M.Tech in Stress and Vibration Analysis from MANIT Bhopal. He is working as an Assistant Professor in Uttaranchal University, Dehradun. His research interests include Heat transfer,solar energy, Materials and Vibrations.

Gaurav Kumar received his B.Tech degree in Mechanical Engineering and pursuing M.Tech in Thermal Engineering from Uttaranchal University.