International Journal of Engineering Sciences & Research Technology

(A Peer Reviewed Online Journal)
Impact Factor: 5.164

IJESRT

IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

ALTERNATING UP AND DOWN DISCRETE V-RIBS ABSORBER PLATE WITH VARYING PARAMETERS: A REVIEW

Deepika Rathore*[1] & Nilesh Sharma[2]
[1]Research Scholar, Department of Mechanical Engineering, Lakshmi Narain College of Technology & Science (RIT), Indore
[2]Assistant Professor, Department of Mechanical Engineering, Lakshmi Narain College of Technology & Science (RIT), Indore

DOI: https://doi.org/10.29121/ijesrt.v9.i12.2020.6

ABSTRACT
Heat transfer enhancement is a subject of considerable interest to researchers as it leads to saving in energy and cost. Because of the rapid increase in energy demand in all over The world, both reducing energy lost related with ineffective use and enhancement of energy in the meaning of heat have become an increasingly significance task for design and operation engineers for many system. This solar energy can be properly utilized by converting it first into thermal energy using solar collectors and then using this energy in various applications. Among the various solar collector systems solar air heater is the simplest and cheapest way to convert the solar energy into thermal energy. Heat transfers enhancement technology has been improved and widely used in heat exchanger application; such as refrigeration, automotive, process industry, chemical industry, etc. One of the widely-used heat transfer enhancement technique is inserting different shaped elements with different geometries in channel flow. Flow simulation analysis of alternating up and down discrete v-ribs with varying parameters is planned as research problem. The Alternating Up and Down Discrete V-ribs is considered and the height, angle and pitch are varying parameters of the geometry to analyze the output variables in the terms of heat flux. The aim is to identify optimized parameters to achieve maximum Heat Flux. Flow simulation module can be used to predicts the outlet temperature, velocity, flow behavior to great accuracy due to application of thermal loading on the work piece.

KEYWORDS: Roughness height, Heat flux, Pitch, Fluid Temperature, Velocity, Pressure, Flow Simulation.

1. INTRODUCTION
Government created the Department of Non-conventional Energy Sources (DNES) in 1982. The phenomenon of Heat transfer Enhancement are Breaking of laminar sub layer, Creation of local wall turbulence and Decrease in the thermal resistance. Solar energy is available freely and an indigenous source of energy provides a clean and pollution free atmosphere. The simplest and the most efficient way to utilize solar energy are to convert it into thermal energy for heating applications by using solar collectors. Solar air heaters, because of their inherent simplicity are cheap and most widely used collector devices. Solar air heaters are being used for many applications at low and moderate temperatures. Some of these are crop drying, timber seasoning, space heating etc.

Solar energy can be properly utilized by converting it first into thermal energy using solar collectors and then using this energy in various applications. Among the various solar collector systems solar air heater is the simplest and cheapest way to convert the solar energy into thermal energy. Poor thermal conductivity of air leads to lesser heat transfer coefficient between absorber plate and air which leads to poor performance of solar air heater.

The thermal efficiency of solar air heaters has been found to be generally poor because of their inherently low heat transfer capability between absorber plate and air flowing in the duct. It has been found that main thermal resistance to the convective heat transfer is due to the formation of convective boundary layer on heat transferring surface. Efforts for increasing heat transfer have been directed towards artificially destroying or disturbing this boundary layer. The use of artificial roughness on a surface is an effective technique to enhance heat transfer to fluid flowing in the duct.

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

The primary forms of solar energy are heat and light. In recent years solar energy has been strongly promoted as a viable energy source. One of the simplest and most direct applications of this energy is the conversion of solar radiation into heat. Purpose of solar air heater is to convert the solar radiation into heat to satisfy energy needs but with some limitations it is not being used on grid scale because of its poor efficiency and higher initial cost. So there is a requirement of advancement in the flat plate collector to overcome its limitations so that it can be used as a replacement of conventional heaters and electric power consuming devices.

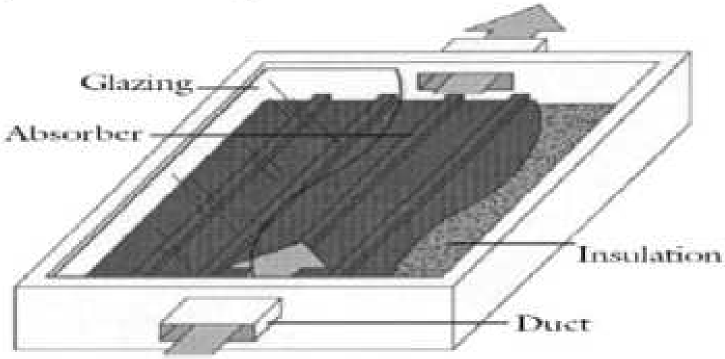

Figure *Error! No text of specified style in document.**.1 Flate Plate Collector***

The term 'flat plate' is slightly misleading in the sense that surface may not be truly flat it may be a combination of flat, grooved or of other shapes as the absorbing surface with some kind of heat removal device like tubes or channels. A flat-plate solar collector as shown in above consists of a water proof, metal or fiberglass insulated box containing a dark colored absorber plate, with one or more translucent glazing. The glazing covers reduce the convection and radiation heat losses to the environment.

Thermal performance may be increased by increasing convective heat transfer coefficient. There are two way for increasing heat transfer coefficient either increase the area of absorbing surface by using fins or create the turbulence on the heat transferring surfaces.

Artificial roughness is basically a transfer enhancement technique by which thermo hydraulic performance of a solar air heater can be improved. The thermal efficiency of solar air heater is generally poor due to low heat transfer co-efficient between the absorber plate and the air flowing in to the duct due to the formation of laminar sub layer on the absorber plate which acts as heat transferring surface. So there is a need to break the laminar sub layer therefore, artificial roughness has been used extensively for the enhancement of forced convective heat transfer, which further requires flow at the heat-transferring surface to be turbulent. However, energy for creating such turbulence has to come from the fan or blower and the excessive power is required to flow air through the duct. Therefore, it is desirable that the turbulence must be created only in the region very close to the heat transferring surface, so that the power requirement may be reduced.

Method and Software

***Computational Fluid Dynamics**:* Computational fluid dynamics (CFD) is a computer-based simulation method for analyzing fluid flow, heat transfer, and related phenomena such as chemical reactions. This project uses CFD for analysis of flow and heat transfer. Some examples of application areas are: aerodynamic lift and drag (i.e. airplanes or windmill wings), power plant combustion, chemical processes, heating/ventilation, and even biomedical engineering (simulating blood flow through arteries and veins). CFD analyses carried out in the various industries are used in R&D and manufacture of aircraft, combustion engines, as well as many other industrial products. It can be advantageous to use CFD over traditional experimental-based analyses, since experiments have a cost directly proportional to the number of configurations desired for testing, unlike with CFD, where large amounts of results can be produced at practically no added expense. In this way, parametric studies to optimize equipment are very inexpensive with CFD when compared to experiments.

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

After studying the basic steps in CFD to be followed to analysis the flow inside a duct. Now we can start the analysis of the solar air heater with actual data .following three steps are required to run the simulation-Preprocessing: 1-CAD modeling, 2-Meshing, 3-type of solver, 4- physical model, 5- material property, 6-boundary condition.

2. LITERATURE REVIEW

Kalpana Soni and Surendra Bharti, 2017, Heat transfer enhancement is a subject of considerable interest to researchers as it leads to saving in energy and cost. Because of the rapid increase in energy demand in all over The world, both reducing energy lost related with ineffective use and enhancement of energy in the meaning of heat have become an increasingly significance task for design and operation engineers for many system. A 3-dimensional CFD analysis has been carried out to study heat transfer and fluid flow behavior in a rectangular duct of a solar air heater with one roughened wall having combination of circular and square transverse wire rib roughness. The effect of Reynolds number, roughness height, roughness pitch, relative roughness pitch and relative roughness height on the heat transfer coefficient and friction factor have been studied. In order to validate the present numerical model, results have been compared with available experimental results under similar flow conditions.

A 3-dimensional CFD analysis has been carried out to study heat transfer and fluid flow behavior in a rectangular duct of a solar air heater with one roughened wall having combination of circular and square transverse wire rib roughness. The effect of Reynolds number, roughness height, roughness pitch, relative roughness pitch and relative roughness height on the heat transfer coefficient and friction factor have been studied. In order to validate the present numerical model, results have been compared with available experimental results under similar flow conditions. CFD Investigation has been carried out in medium Reynolds number flow (Re¼3800e18, 000).

Kalpana Soni and Surendra Bharti, 2017, Energy is the basic ingredient to sustain life and development. Work means moving or lifting something, warming or lighting something. There are many sources of energy that help to run the various machines invented by man. Average Nusselt number increases with an increase of Reynolds number. The maximum value of average Nusselt number is found to be 43.577 for relative roughness pitch of 5 and for relative roughness height of 0.06 at a higher Reynolds number, 10,000. Average friction factor decreases with an increase of Reynolds number. The maximum value of average friction factor is found to be 1.698 for relative roughness pitch of 5 and relative roughness height of 0.06 at a lower Reynolds number, 5000.

The conclusions are drawn from present analysis are Average Nusselt number increases with an increase of Reynolds number. The maximum value of average Nusselt number is found to be 43.577 for relative roughness pitch of 5 and for relative roughness height of 0.06 at a higher Reynolds number, 10,000. Average friction factor decreases with an increase of Reynolds number. The maximum value of average friction factor is found to be 1.698 for relative roughness pitch of 5 and relative roughness height of 0.06 at a lower Reynolds number, 5000.

Anil Singh Yadav and J.L.Bhagoria, 2013, This paper presents the study of heat transfer in a rectangular duct of a solar air heater having triangular rib roughness on the absorber plate by using Computational Fluid Dynamics (CFD). The effect of Reynolds number on Nusselt number was investigated. The computations based on the finite volume method with the SIMPLE algorithm have been conducted for the air flow in terms of Reynolds numbers ranging from 3000-18000. A commercial finite volume package ANSYS FLUENT 12.1 is used to analyze and visualize the nature of the flow across the duct of a solar air heater. CFD simulation results were found to be in good agreement with experimental results and with the standard theoretical approaches. It has been found that the Nusselt number increases with increase in Reynolds number.

In the present simulation governing equations of continuity, momentum and energy are solved by the finite volume method in the steady-state regime. The numerical method used in this study is a segregated solution algorithm with a finite volume-based technique. The governing equations are solved using the commercial CFD code, ANSYS Fluent 12.1. A second-order upwind scheme is chosen for energy and momentum equations. The SIMPLE algorithm (semi-implicit method for pressure linked equations) is chosen as scheme to couple pressure and velocity. The convergence criteria of $10^{-3}$for the residuals of the continuity equation, 10^{-6} for the residuals of the velocity components and 10^{-6} for the residuals of the energy are assumed. A uniform air velocity is introduced at

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

the inlet while a pressure outlet condition is applied at the outlet. Adiabatic boundary condition has been implemented over the bottom duct wall while constant heat flux condition is applied to the upper duct wall of test section.

The patterns of temperature contour at regions behind and ahead of the rib illustrate the overall temperature field and the degree of heat transfer. CFD predicts temperature contour pattern better at the regions ahead of the rib. The contour of stream function for the triangular shape of ribs inserted in a solar air heater duct. An observation of stream function contours reveals that vortex formation at top of the rib surface provides rolling action to the flow and hence reduces the friction.

In this present investigation, a numerical prediction has been conducted to study heat transfer and flow friction behaviors of a rectangular duct of a solar air heater having triangular rib roughness on the absorber plate. The main conclusions are:

Mr. Obaid Ul Haque Ansari and Dr. V.R. Kalamkar, 2017, In the present work the thermo hydraulic performance of solar air heater having absorber plate roughened with compound turbulators is analyzed using Computational Fluid Dynamics (CFD). The effect of absorber plate geometry on heat transfer and friction factor is studied by varying the relative roughness height (e/DH) from 0.025 to 0.040(4 values) and relative roughness pitch (P/e) from 8 to 15 (4 values). Range of Reynolds number selected for numerical simulation is 3000 to 18,000 (6 values). Different turbulent models have been used for the analysis of solar air heater and their results are compared with the experimental data available in the literature. The results obtained using K- ϵ Standard model have been found in good agreement with the experimental results and hence this model is used to predict the heat transfer and friction factor in the roughened duct for various geometries of an absorber plate. The thermo hydraulic performance parameter (THPP) is also evaluated for each plate to predict the overall performance and also for selecting the best geometry for the range of parameter investigated.

A numerical study of the flow of air in a rectangular duct with one roughened wall subjected to uniform heat flux and with the other three smooth walls being insulated has been performed. These conditions correspond to the flow in the duct of a solar air heater. The effect of relative roughness pitch, relative roughness height at constant relative groove position on the friction factor and heat transfer coefficient has been studied.
The major conclusions are:

1. As compared to the smooth duct, the introduction of compound turbulators yields Nusselt number from 1.3668 to 2.7041 times while the friction factor rises from 1.7162 to 2.7711 times for the range of parameters investigated.
2. The maximum heat transfer occurs for a relative roughness pitch of 10, while the friction factor goes on decreasing as the relative roughness pitch increases.
3. Both heat transfer and friction factor goes on increasing with increase in relative roughness height for the range of parameters investigated.
4. For selected range of roughness parameter, the range of the Thermo Hydraulic Performance Parameter for the roughened duct with compound turbulators comes out to be 1.0896 to 1.9456. The highest THPP obtained (1.9456) is greater to that of highest THPP (1.87) obtained by numerical simulation of transverse square ribs.
5. The Mean THPP for the range of Re = 3000 to 18,000 is maximum (1.8369) for Plate 14. Hence Plate 14 (P/e = 10 and e/DH = 0.040) has an optimized geometry of absorber plate for the range of roughness parameter investigated.

Rajeev Ranjan, M.K. Paswan and N. Prasad, 2017, In the present work the study of heat transfer and friction characteristics in rectangular duct of a solar air heater having isosceles right triangle rib shape on the absorber plate by using CFD (Computational Fluid Dynamics) The roughness has been placed in such way that the hypotenuse faces against the flow direction. The design parameters chosen for study are Reynolds number, roughness height and roughness pitch. The turbulence model used in CFD solutions is Renormalization group (RNG) k-ε. The roughness height (e) is chosen as 0.5mm, 1.0mm and 1.5mm at roughness pitch (P) of 5mm, 10mm, 15mm and 20mm. The relative roughness Pitch (P/e=3.33-40) and relative roughness height (e/D= 0.015-0.045) is taken. The effect of transverse wedge shape on heat transfer and friction factor was investigating covering the range of roughness parameter having Reynolds no varies from 3000 to 18000. ANSYS FLUENT is used to simulate turbulent airflow in artificial roughened solar air heater. Five different turbulence models is tested on the quality of obtained results .The RNG k-ε model yield better result for two dimensional flow through conventional solar air heaters. Thermo-hydraulic performance is also calculated. Correlations for the Nusselt number and friction factor have been developed.

CFD simulation of two-dimensional artificially roughened solar air heater duct along transverse ribs roughness with isosceles right triangle cross-section as a vortex generator in inlet section is carried out using the CFD software package ANSYS FLUENT (version 14.5). The general assumption, consider in the analysis are as follows:

1. The flow is steady, two dimensional, turbulent and fully developed.
2. The flow is single phase across the duct.
3. The walls in contact with fluid are assigned, no-slip boundary condition.
4. The duct wall, absorber plate and roughness material are homogeneous and isotropic. The thermodynamic properties of air and the absorber plate (aluminum) are considered constant.
5. The effect of radiation heat transfer and other heat losses are considered less.

On the basis of the results and discuss the following conclusion has been drawn:

1. CFD has been applied in design of solar air heater. The quality of the solutions obtained from CFD simulations are largely within the acceptable range providing that CFD is an effective tool for predicting the behavior and performance of a solar air heater.
2. Nusselet number increases with the increase of Reynolds number. Solar air heater with isosceles right triangle gives 3.06 times enhancement in Nusselt number in comparison to that of a smooth plate.
3. Roughened absorber plates perform better at lower Reynolds number and smooth plates should be preferred when operating at higher Reynolds number. The maximum average Nusselt number is 70 for relative roughness pitch 3.33and relative roughness height 0.045.
4. Maximum thermo-hydraulic performance is obtained for relative roughness pitch of 5 and relative roughness height 0.03.
5. The maximum enhancement of friction factor 3.45 times in comparison over the smooth plate.
6. Statistical correlations for Nusselt number and friction factor have been developed. These correlations have been found to predict the value within the error limits.

Navneeta Bisht and Vijay Singh Bisht, 2018, Solar air heater is a device which is used to trap incoming solar radiations from the sun but the efficiency of solar air heater is low because of low value of heat transfer coefficient of air so the roughness is provided at the bottom of the absorber plate so as to increase the value of heat transfer. So, here we have used the crown shape roughness to increase the heat transfer rate. The parameters used are: heat flux of 1000W/m2, Reynolds number (Re) ranging between 2000and 20,000 and the relative roughness pitch (p/e) used are 10, 15, 20, 25mm at various rib heights: 0.8, 1.2, 1.8 and 2.2 mm respectively. The duct size and depth are 640mm and 20mm, hydraulic depth is 33.33mm and angle of attack used are 350, 450 and 550.The heat transfer and friction factor values obtained are compared with those of smooth duct under similar flow conditions. Investigation shows that better performance is achieved by reducing the friction factor and increasing the Nusselt number. The average variation in the Nusselt number and friction factor in the range of 3.386632 - 4.823277 and 1.776364 - 1.937077 times of the results of Smooth Duct respectively.

A.Boulemtafes- Boukadoum and A. Benzaoui, 2014, In this paper, Author presented the results obtained by numerical analysis of flow and heat transfer in the air duct of a solar air collector, whose absorber is provided with artificial rectangular ribs. The analysis is based on CFD techniques and was performed using numerical software. This numerical analysis allowed us to found out the effect of absorber roughness on the air flow and heat transfer enhancement in solar air heaters. The first part of the analysis aims to validate the numerical model by comparing results to Karwa's experimental results. It have been compared between four turbulence closure models and from the results, it is clear that the k-w SST gives the best results. The second part is an approach to the solar air heater in real operating conditions. This analysis allowed author to visualize the separation and reattachment zones. It is also distinguished the over-speed area, where the velocity fluid reaches over 150% its initial velocity. Also plotted Nusselt number, hc and Ts evolution with Reynolds number. The global thermo

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

hydraulic performance parameter ET is a good indicator of the effect of artificial ribs in the heat transfer enhancement in solar air heaters. The curve representing its evolution with Reynolds number shows good performance for rectangular ribs used. Moreover, the geometric shape of roughness studied gave rise to friction factors and therefore not penalizing the thermal-hydraulic performance. So, it is recommended using this type of roughness to improve the thermal performance of solar air collectors.

Dr. A. Nagaraju , Dr. B. Chandramohan Reddy and Dr. K. Umamaheswari, 2016, A solar air heater utilizes solar thermal energy to heat the air and further it can be used efficiently in space heating and industrial process heating. An absorber plate of good thermal conductivity is exposed to green renewable solar energy and thus gets heated. The air thus gets heated by the phenomenon of forced convection by the absorber plate. Many attempts were made by the researchers to destroy the laminar sub layer thus formed, by adding artificial roughness to the absorber plate. The roughness obstructs the laminar flow of air near the surface thus create turbulence. The turbulence aid the enhancement of heat transfer between the absorber plate and the air. Here an attempt is made to analyze the heat transfer between smooth absorber plate surface and air flowing in the duct due to forced convection, also when artificially roughened absorber plate surface used instead through computational fluid dynamics simulation by using STAR-CCM+ software. There is appreciable increase in temperature at outlet in solar air heaters using artificially roughened absorber plates. The V-shaped rib roughness gives high rate of heat transfer.

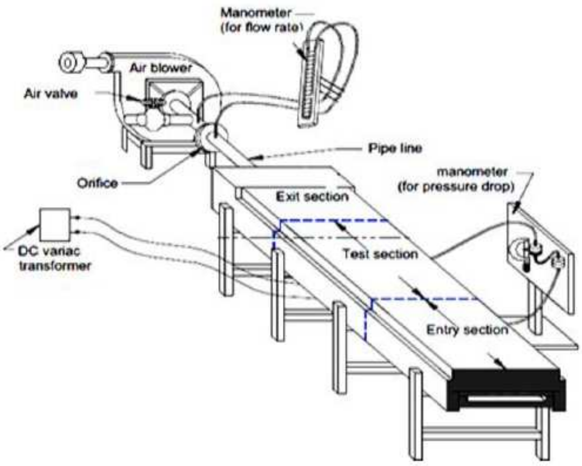

Figure *Error! No text of specified style in document.* ***Schematic of Solar Air Heater***

An attempt has been made to carryout CFD based analysis to fluid flow and heat transfer characteristics of a solar air heater having roughened duct provided with artificial roughness in V-shaped geometry.

Combined effect of swirling motion, detachment and reattachment of fluid which was considered to be responsible in the increase of heat transfer rate has been observed during CFD analysis.

There is appreciable increase in temperature at outlet in solar air heaters using artificially roughened absorber plates.

Compared to smooth absorber plate, there is no much pressure drop across the duct even in case of rough absorber plate.

The V-shaped rib roughness gives high rate of heat transfer thus this type of surface roughness plate can be recommended to be used in solar air heat.

Hrishikesh A. Dhandha and Anjali A. Vyas, 2015, This paper presents computational fluid dynamics (CFD) analysis to determine fluid flow and heat transfer characteristics of solar air heater. Artificial roughness is used in one side wall of solar air heater absorber plate to break laminar boundary sub layer. It enhances rate of heat transfer

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

from the absorber plate to flow of air stream. A CFD-based investigation of 3-dimensional forced convective fluid flow over solar air heater rectangular duct with one broad wall roughened by half perforated transverse baffles has been performed in ANSYS FLUENT software. The duct has width to height ratio of 7.77; the baffle relative roughness pitch ratio is 7.06; the relative baffle height ratio is 0.495.The Reynolds number of the analysis ranges from 3000 to 9000. The effect of roughness geometry [i.e., relative roughness height of baffle (e/h), relative roughness pitch of baffle (p/e), open area ratio (β)] on the heat transfer coefficient and Nusselt number predicted. Solar air heater with artificial roughness experimental model's 3-dimensional geometrical modelling made with aid of ANSYS Workbench. The results were predicted by ANSYS FLUENT 14.5 solver. Over the range of study the Heat transfer coefficient and Nusselt number obtained in range of 14.5-29 W/m2K and 35- 80 respectively for variable mass flow rate. Validation of results compared with performed experimental work and found to be in good agreement.

A Three- dimensional Computational fluid dynamics (CFD) analysis has been carried out to study heat transfer behavior in a rectangular duct of solar air heater having artificial roughness. There is a good agreement between the experimental and simulated results for outlet air temperatures. The Nusselt number of CFD results has maximum ±8.73 % over experimental results. Although there are some small discrepancies due to some experimental imperfectness matters, we still have a good confidence in the CFD simulation program that can be used in the future for complex solar collector problem. Considering present predicted CFD results following relevant conclusion are drawn. A kinetic energy turbulence module is taken for analysis in which heat transfer coefficient increase due to artificial roughness. Nusselt number is increase with increase in Reynolds number for range of simulation.

Vivek Rao, Dr. Ajay Gupta and Amit Kumar, 2015, In the present study, the thermo-hydraulic performance of four ribs–roughened rectangular duct is investigated. The aspect ratio of the duct was kept constant as 5. Symmetry and periodic boundary conditions are used to minimize the computational cost. Four rib configurations were tested: V-up rib, V-up Broken rib, V-down broken rib, and Multi V rib pointing upstream of the main flow direction. Profile boundary condition was created at the outlet of the test section and was applied to various inlet con ditions of main rib configuration. For all cases hydraulic diameter and angle of attack were kept constant to 33 mm an d 600. Only relative roughness pitch and Reynolds number were varied from 8-12 and 8000-15000.

The overall objective of this study was to investigate the effect of various dimensionless parameters on physics of heat transfer process in terms of Nusselt number using FEM simulations.

CFD simulation software, ANSYS (workbench mode) has been used for simulation. Fluid flow (FLUENT) module has been used to demonstrate detailed temperature distribution, velocity path-line between a pair of ribs on the ribbed surface. The secondary flow and vortices' caused by the inclination of the rib and broken condition of rib create a significant span wise variation of heat transfer coefficient due to turbulent associated with it. Y plus value is calculated to test the accuracy of the mesh. Solar air heater provided with Multi-V rib gives maximum Nusselt number of 145 at relative roughness pitch of 8 and Reynolds number of 15000. This simulation module predicts the velocity path-line and the temperature contour to predict the nature of flow of fluid and distribution.

Rajeev Ranjan, M.K.Paswan and N.Prasad, 2015, A numerical investigation of fluid flows through a solar air heater with semi-circular sectioned transverse rib roughness has been executed. The physical problem is represented mathematically by a set of governing equations, and the transport equations are solved using the finite element method. The effect of relative roughness pitch on average Nusselt number, average friction factor, and thermohydraulic performance parameter (THPP) has been investigated. This investigation is taken for relative roughness pitch of 7.5 and relevant Reynolds numbers in the range of $3800 \leq Re \leq 18,000$. The maximum thermohydraulic performance parameter is 1.616 and it is occurred for the relative roughness height of 0.06 and relative roughness pitch of 7.5.

A 2-dimensional CFD analysis has been carried out to study heat transfer and fluid flow behavior in a rectangular duct of a solar air heater with one roughened wall having semi-circular transverse wire rib roughness. The following conclusions are drawn from present analysis:

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

1. The maximum Friction factor ratio is found to be 4.08324 corresponding to relative roughness pitch (P/e) of 7.5 at a Reynolds number of 18,000.
2. It is found that the solar air heater roughened with semi-circular sectioned transverse rib roughness on the absorber plate with relative roughness pitch, P/e = 7.5 provide the better THPP of 1.6165 at a Reynolds number of 3,800.

Yadaba Mahanand, Abhijit Mahato and Debanshu Shekhar Khamari, 2018, The 2D heat transfer and fluid flow phenomenon through an artificially roughened solar air heater is investigated by means of a numerical model at a constant heat flux of 1000 W/m2. A modern CFD code ANSYS FLUENT v 14.5 is used to simulate fluid flow and heat transfer through the solar air heater. The duct wall, Absorber plate and roughness materials are assumed to be homogeneous & isotropic and also the thermal conductivity is independent of temperature. The present work show that the Renormalization-group k-epsilon model provides the results close to those, worked out from available empirical co-relation for two-dimensional steady flow solar air heaters. The maximum enhancement of average Nusselt number has been found to be 2.3104 times that of smooth duct for relative roughness pitch of 7.14 and for relative roughness height of 0.042.

The selection and validation of turbulence model is carried out by comparing the Nusselt number predicted by different turbulence model such as standard k-ε model, Renormalized group k-ε model (RNG) with empirical correlation available for smooth duct of the solar air heater i.e. Dittus-Boelter correlation.

The effects of Reynolds number, relative roughness pitch and relative roughness height on the heat transfer and fluid flow process are discussed. Key findings from the study are as follows:
1. The Renormalization-group (RNG) k-epsilon turbulence model predicted very close results to the experimental results, which yields confidence in the predictions done by numerical analysis in the present study.
2. The maximum enhancement of average Nusselt number has been found to be 2.3104 times that of smooth duct for relative roughness pitch of 7.14 and for relative roughness height of 0.042.
3. The maximum enhancement of average friction factor has been found to be 3.12 times that of smooth duct for relative roughness pitch of 7.14 and for relative roughness height of 0.042. The maximum enhancement of average friction factor occurs at a Reynolds number of 3800.

Dr. Ashwini Kumar, Abhijit Mahato and Dr. Aruna Kumar Behura, 2017, A CFD analysis is conducted through different turbulence models to study the performance of a solar air heater using corrugated absorber plate. A modern CFD code ANSYS FLUENT v 14.5 is used to simulate fluid flow and heat transfer through the solar air heater. Flow is assumed to be two-dimensional and the heat flux is considered at a constant value of 910 W/m2. The present work show that the Renormalization-group k-epsilon model provides the results close to those, worked out from available empirical co-relation for two-dimensional steady flow solar air heaters.

Prateek Minhas, Kumari Ambe Verma, Sudhanshu Dogra and Nitin Chauhan, 2017, Performance of the solar air heater can be enhanced by increasing the area of heat transfer. There are different methods used to enhance the performance of solar air heater i.e. using corrugated surfaces, extended surfaces, packed bed etc. The thermal efficiency of single pass solar air heater is less than the double pass solar air heater because of less area of heat transfer. By various literatures, it is concluded that maximum studies are done on solar air heater double pass integrated with extended surface and porous media but minimum studies are on corrugated surfaces. An efficient method is considered is to apply artificial roughness on the absorber plate of solar air heater to increase the thermal efficiency. The efficiency of solar air heaters can be influenced by different parameters such as depth of channels, number of channels, collector length, absorber plate type, material and number of glass cover, inlet air temperature and velocity of air.

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

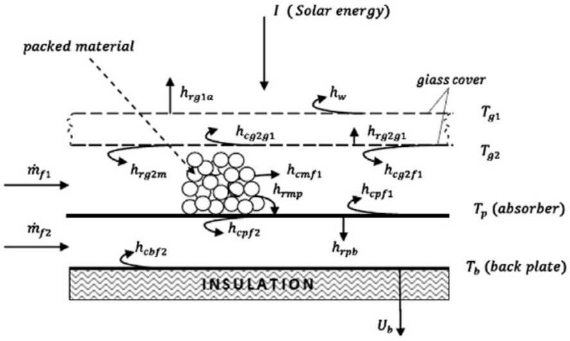

Figure 3 Schematic diagram of parallel flow packed bed solar air heater

Based on the review of literature solar air heater different approaches of mathematical and analytical were observed by many researchers. There are many studies have been carrying on to find the effects of different parameters on the solar air heater's performance. The rate of mass flow of air and the porosity of material of packing are assumed to be significant parameters that influence the performance of solar air heater with packed bed. For improvement in the performance of solar air heater various studies(mathematical and analytical)have been carried out on integrated fins, packed bed, corrugated absorber/collector plate in comparison to conventional system. There is a vast scope for study of using artificial roughness attached on both lower and upper side of absorber/collector plate of solar air heater in future. The information provided in this paper helps the new researchers in this research area.

Rohan Sonalkar and Parth Trivedi, 2018, The heat transfer rate in a solar air heater duct has been found to be low due to the formation of laminar sub-layer when air passes through the absorber plate. The use of artificial roughness on the underside of the absorber plate is an effective technique to break this laminar sub-layer and hence to enhance the rate of heat transfer. In the present work, Computational Fluid Dynamics (CFD) is used to analyze the air-flow and heat transfer rate in the duct of solar air heater. RNG k-epsilon turbulence model is selected by comparing the results obtained from different turbulence models. A detailed numerical analysis is carried out using this model, firstly, for a smooth plate and then with a series of NACA 0030 airfoils as artificial roughness arranged with an appropriate pitch on the absorber plate for Reynolds number ranging from 6000 to 18000. A constant solar radiation of 1000 W/m2 is provided on the absorber plate. Correlations for Nusselt number and friction factor have been developed which show a significant increase in the heat transfer rate in comparison to that for a smooth surface without noticeable friction losses.

In this paper, author have presented the results obtained by numerical analysis of air-flow and heat transfer in the duct of a solar air heater, whose absorber plate is provided with artificial roughness in the form of NACA 0030 airfoils. The analysis is based on CFD techniques and was performed using numerical software ANSYS FLUENT 16.2. This numerical analysis allowed us to found out the effect of roughness on the air flow and heat transfer enhancement in solar air heaters.

The first part of the analysis aims to validate the turbulence model by comparing different results. We found RNG k-epsilon model the best and hence used it for analysis. The second part is an approach to analyze the thermal performance of solar air heater in real operating conditions.

Author have plotted Nusselt number and friction factor variations with Reynolds number. The results are indicators of the effect of NACA 0030 airfoils in the heat transfer enhancement in solar air heater.

Moreover, the geometric shape of roughness of NACA 0030 airfoils gave rise to heat transfer with moderate friction and therefore not penalizing the thermal performance.

So, author recommends the use of this type of roughness to improve the thermal performance of solar air heater.

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

3. CONCLUSION

There is no doubt that a major focus of CFD analysis for solar air heater is to enhance the design process that deals with the heat transfer and fluid flow. In recent years CFD has been applied in the design of solar air heater. The quality of the solutions obtained from CFD simulations are largely within the acceptable range proving that CFD is an effective tool for predicting the behavior and performance of a solar air heater. It is also seen that Nusselt number increases with the increase of Reynolds number. There is a significant improvement in effective efficiency of solar air heater by providing different types of roughness elements on the absorber plate. Heat transfer distribution depends upon type of dimensionless parameter and highest at a specific value. Computational cost is saved by use of periodic and symmetry boundary condition.

REFERENCES

[1] Kalpana Soni and Surendra Bharti, "CFD Analysis of Solar Air Heater Provided With Circular and Square Section Transverse Rib Roughness on the Absorber Plate: A Review", International Journal of Computer Application (2250-1797) Volume 7– No.3, May – June 2017, PP74-79.
[2] Kalpana Soni and Surendra Bharti, "CFD Analysis of Solar Air Heater for Enhancement of Heat Transfer", International Journal of Engineering Sciences & Research Technology, 6(6): June, 2017, PP 430-444.
[3] Anil Singh Yadav and J.L.Bhagoria, "A CFD Analysis of a Solar Air Heater Having Triangular Rib Roughness on the Absorber Plate", International Journal of ChemTech Research, Vol.5, No.2, April-June 2013, PP 964-971.
[4] Mr. Obaid Ul Haque Ansari and Dr. V.R. Kalamkar, "CFD analysis of solar air heater having absorber plate roughened with compound turbulators", International Journal of Engineering Development and Research, Volume 5, Issue 3, PP 40-58.
[5] Rajeev Ranjan, M.K. Paswan and N. Prasad, "CFD based Analysis of a Solar Air Heater having Isosceles Right Triangle Rib Roughness on the Absorber Plate", International Energy Journal 17 (2017), PP- 57 – 74.
[6] Navneeta Bisht and Vijay Singh Bisht, "CFD Analysis of Solar Air Heater Roughened With Crown Shape Roughness", Asian Journal of Applied Science and Technology, Volume 2, Issue 3, July-September 2018, PP 73-82.
[7] A.Boulemtafes- Boukadoum and A. Benzaoui, "CFD Based Analysis of Heat Transfer Enhancement in Solar Air Heater Provided with Transverse Rectangulat Ribs", The International Conference on Technologies and Materials for Renewable Energy, Environment and Sustainability, Energy Procedia 50 (2014), PP 761 – 772.
[8] Dr. A. Nagaraju , Dr. B. Chandramohan Reddy and Dr. K. Umamaheswari, "CFD Analysis of Flow and Heat Transfer in Solar Air Heater Duct by using V-Shaped Inclined Continuous Ribs on Absorber Plate, International Journal of Engineering Research & Technology (IJERT), Vol. 5 Issue 11, November-2016, PP 227-230.
[9] Hrishikesh A. Dhandha and Anjali A. Vyas, "CFD Analysis of Solar Air Heater with Perforated Baffle on the Absorber Plate", International Journal of Modern Trends in Engineering and Research, Volume 2, Issue 7, [July -2015] Special Issue of ICRTET'2015, PP 1986-1991.
[10] Vivek Rao, Dr. Ajay Gupta and Amit Kumar, "CFD Based Heat Transfer Analysis of Solar Air Heater Duct Provided with Artificial Roughness", International Journal of Scientific & Engineering Research, Volume 6, Issue 5, May-2015, PP 28-35.
[11] Rajeev Ranjan, M.K.Paswan and N.Prasad, "CFD Investigation of Artificially Roughened Solar Air Heater for Performance Enhancement", International Advanced Research Journal in Science, Engineering and Technology Vol. 2, Issue 9, September 2015, PP59-64.
[12] Yadaba Mahanand, Abhijit Mahato and Debanshu Shekhar Khamari, "CFD Analysis of Semi-Circular Rib Roughened Solar Air Heater", International Journal of Advanced Mechanical Engineering, Volume 8, Number 1 (2018), PP 251-262.
[13] Dr. Ashwini Kumar, Abhijit Mahato and Dr. Aruna Kumar Behura, "CFD Analysis of Solar Air Heater Having Corrugated Absorber Plate", International Journal of Emerging Technology and Advanced Engineering, Volume 7, Issue 9, September 2017, PP 575-587.

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

[14] Prateek Minhas, Kumari Ambe Verma, Sudhanshu Dogra and Nitin Chauhan, "Study of Solar Air Heater Having Different Geometries on Both Sides of Absorber Plate", International Journal of Mechanical Engineering and Technology, Volume 8, Issue 7, July 2017, PP 1299–1310.
[15] Rohan Sonalkar and Parth Trivedi, "CFD based performance analysis of a roughened solar air heater duct having NACA 0030 airfoils as artificial roughness", International Research Journal of Engineering and Technology, Volume: 05 Issue: 03, Mar 2018, PP 2882-2887.