Evaluation of Solar Air Heater Performance with Artificial Rib Roughness over the Absorber Plate using Finite Element Modelling Analysis

K. Ravi Kumar1, P. Nikhil Varma1, N. Jagadeesh1, J. V. Sandeep1, Muralimohan Cheepu2*, D. Venkateswarlu3, B. Srinivas1

1Department of Mechanical Engineering, MVGR College of Engineering, Andhra Pradesh 535005, India
2Department of Mechatronics Engineering, Kyungsung University, Busan 48434, Republic of Korea
3Department of Mechanical Engineering, Marri Laxman Reddy Institute of Technology and Management, Telangana 500043, India

*Corresponding author E-mail: muralicheepu@gmail.com

Abstract. Among the different renewable energy resources, solar energy is widely used due to its quantitative intensity factor. Solar air heater is cheap, simple in design and has got wide range of applications. A modest solar air heater has a lower in heat transfer and thermal performance as it has heat transfer coefficient lower in between coated absorber plate and the carrier fluid. This low thermal performance can be reduced to a greater extent by introducing the artificially created roughness over the absorber plate of the solar heater. In the present study, the combination of various geometries and roughness’s on the absorber plate are reported. Methods have been developed and implemented in order to improve the rate of the heat transfer. A comparison is drawn among different geometries to select the most effective absorber plate roughness. For flow analysis k-ω SST model was used and the constant heat flux was taken as 1100 W/m². The Reynolds number is varied in a range from 3000 to 20000. The variation of different parameters temperature, Nusselt number, turbulence kinetic energy and heat transfer coefficient with Reynolds number were examined and discussed.

1. Introduction

In recent years, it has been found that energy consumption rate increasing gradually with the technology development in various field. Among the various sources of energy, a tremendous solar energy is the vastly and readily available source of energy. A clean source of solar energy is the non-pollutant, unlike other type of sources of energy such as coal, oil, fossil fuel and natural gas which cause more destruction to the environment and environment in expressions of pollution of water and air, wild life destruction, community health damage and totally global warming. Moreover, these fossil fuels are diminishing with the vast usage for various applications, hence the renewable energy arises as alternative and very clean basis to satisfy the energy anxieties and become a leading source of one of the main contributor in recent economic development [1]. Since from many years, energy from solar used as an eco-friendly fuel of energy reservoir and it has to be transformed into thermal energy to utilize it for several energy based applications. The solar energy gathering from the sun radiation using flat perpendicular surface equipment, there are various types of equipment and methods exists to convert solar energy into thermal
energy, among them flat plate solar collectors become most efficient for various solar air heater applications and water heater [2]. Generally, solar air heater can efficiently transforms solar radiation into heat and that can produces a heat to air. Now a days many solar heating systems have been developed in order to avoid the consumption of domestic sources of oil, gas, electric and other types of heat sources. Using solar air heaters supplying the hot air for drying the most useful applications for agriculture products, marine products, warm the buildings in cold countries to maintain the human comfortable environment [3]. The use of conventionally available regular solar air heaters determined to get initially deprived thermal efficiency because of the convective hot air heat transfer and its coefficient was low between flowing stream and absorber plate of the solar air heater. The hot air convective heat transfer amongst elegant air stream and air heater plate gradually increased due to the increasing of the frequency of turbulence via flowing the horizontal laminar viscous sublayer. However, to overcome these issues artificial roughness absorber plates have been developed. The use of artificially created roughness on intense observed surface of the plate is the efficient inactive technique, which is recycled to increase the heat flow transfer. Hence, the improvement in heat flow transfer resulted in an inexorably drop in high pressure loss of the air or fluid flow. It is very important to minimize the friction losses that can be controlled by creating the turbulence in the region which is closed to the duct surface of laminar sublayer [4]. The solar air heaters comprising of physics of air or fluid flow and the heat flow and these two flows are easily predicted through its performance of hydraulic, thermal and the combination of thermos and hydraulic efficiencies. Therefore the concert of thermal accompanies assessing heat transfer flow and coefficient of the heat transfer and are renovating it into thermal efficiency and non-dimensional Nusselt’s number. In addition, solar air heaters thermal performance can be anticipated trough Hotell-Whillier-Bliss advanced mathematical equations which are defined through Duf-fie and Beckman [5]. The developed heaters using artificial roughness techniques for solar air heater are in the arrangement of the very thin lines of various sizes, forms and different alignments on the bottom of the absorber plate which is among the most efficient technique and valued strategy enhances that the anticipated to progress the thermo-hydraulic performance [6]. The recent literature studies reported that the various experimental works were performed for knowing enactment valuation of developed solar air heater via artificially roughened. The manufacturing and application of theatrically solar air heaters of artificial roughened may be originate in research reports of various researchers [7-23]. The previous literature reports revealed that very few studies have been conducted on computational fluid dynamics (CFD) approach for evaluating the concert of artificially roughened solar air heater. The excellent outcome from the simulation results gains the importance and hasty development of powerful computers for utilising major resources and improvement of CFD software packages which can solved for industrial problems efficiently. It is found that CFD has already proven to be a valid simulation tool for fluid flow and structure flow studies [3,24]. It is well known that the computational simulation of flow structure computed by various mathematical equations solving and its solutions given the output to govern the flow dynamics. In simulation, the results are the complete description of the three-dimensional flow in the entire results flow domain of the pressure distribution, velocity flow and temperature variations with density and other related physical properties in values and units.

Yadav et al. [25], have been reported the conventional solar air heaters performance using a CFD based simulation studies. To analyze the nature of the flow Ansys Fluent simulation tool and RNG k-ε turbulence model were used and output of the results satisfy the experimental results. In overall, the outcome of the literature reports shows that the very few studies have been available on this issue using CFD simulation tool to analyses the artificial roughened solar heater absorber plates. In the present investigation, different geometry of the ribs roughness on the absorber plates were selected to find the variants in the output efficiency of the flows to assessing of heat transfer, along with hot air resistance in a solar air heater using conventional CFD simulation study. It is also aimed to study the influence of thickness of the roughness on the Nusselt number, also the friction factor with other studies by Ansys CFD tool.
2. Modeling of Geometry

Figure 1 illustrates the shape and sizes of the different types of ribs used in this study. The ribs geometry was constructed as per the schematics (shown figure 1) using commercially available software CREO 2.0. The geometry outlines with ribs was produced in x-y plane with suitable proportions are given in table 1 (in mm) and at that point exterior parts was made from the fill option for 2-dimensional geometry and volume was generated from the extrude option for 3-dimensional geometry. The surface area of the different shapes of ribs was made equal while constructing the remaining geometries. The previous studies proposed that an appropriate 2D model is capable to provide the output of the simulation effectively the characteristics of forced convection and efficiency of turbulent flow were the solar air heater of artificially roughened having a rib shape of circular shape on the absorber plate. Due to this, there is no need of using such a complicated 3D models to analyse the simulation results [26]. The computational domain of 2D view has been selected for this study the purpose of reducing the computational time. The model was divided into three sections such

![Figure 1: Schematic view of the shape and sizes of the ribs](image)

Figure 1 Schematic view of the shape and sizes of the ribs (a) square and (b) equilateral triangle shape ribs.

![Figure 2: Geometry and meshing domains of the ribs](image)

Figure 2 Geometry and meshing domains of the (a) square and (b) equilateral triangle shape ribs.
as inlet, outlet and test section to evaluate the results easily using an ASHRAE standard method [3,27]. The meshing domain work was conducted on ANSYS software which is commercially available for generating the meshing in software. The final form of geometry which is created in CREO 2.0 was imported in ANSYS meshing. The prerequisite amount of separations and the type of “bias” were allotted to each and every edge. With the intention of to attain regular shape of rectangular one is the mesh cells with the best orthogonal excellence, and the facing option of the mapped was initiated. At the end of the modelling study the required mesh was engendered by providing the prerequisite tools buttons (see figure 2). The generated mesh was very fine in size and it was accomplished adjacent to the walls with the purpose of solving the apprehensive governing differential equations precisely in the form of linear flow so as to the laminar sub-layers by the side of the meshed regions. It is observed that the size of the generated mesh amplified in the direction of the centre region, as shown in figure 2. The grid size found to be constant for the size of length in entry and leaving segments of the surfaces of the duct. Also the results observed to be was confirmed determined characteristic ratio of every grid at anyone was did not exceed 10. The mesh was created with acceptable values of orthogonal quality near to one and skewness factor near to less than 0.25, and are specified in table 2.

| Geometrical parameters          | Shape of the ribs |
|--------------------------------|-------------------|
|                                | Square            | Equilateral triangle |
| Height of duct (H) (mm)        | 20                | 20                  |
| Height of rib (e) (mm)         | 3                 | 3.89                |
| P/e ratio                      | 10                | 10                  |
| Length of test section (mm)    | 280               | 280                 |
| Length of outlet (mm)          | 115               | 115                 |
| Width of duct (mm)             | 100               | 100                 |
| length of inlet (mm)           | 245               | 245                 |

| Meshing Parameters             | Shape of the ribs |
|--------------------------------|-------------------|
|                                | Square            | Equilateral triangle |
| Nodes                          | 278558            | 239283              |
| Elements                       | 246300            | 210700              |
| Orthogonal quality             | 0.9837            | 0.9835              |
| Skewness                       | 0.1311            | 0.1153              |

After meshing, the solver settings are given to obtain the solution, the fluent code was given for solving the viscous incompressible flow through the available three-dimensional equations. The continuum was chosen as fluid and the properties of water were assigned to it. To run the simulation, the analogous form of simulation tool FLUENT concurrently can calculates the equations for flow patterns by various processors/computers. However, the running software can be able to repeatedly separate the partition of the grid into different areas to allocate the computational work between accessible amounts of processors.
The selection of boundary conditions are a crucial for attaining accurate results in a CFD problem, are given in table 3. The inlet and outlet sections are defined as velocity inlet and pressure outlet respectively with a velocity corresponding to Reynolds number $3 \times 10^3$ to $18 \times 10^3$ at the inlet and static pressure 0 atm at the outlet. After the testing of the ribs, the separated sections comes the heated surface. It is divided that, top face is the wall with the selection of properties of wooden block. The boundary conditions are chosen from previous works which has given satisfactory results on similar simulations [3]. The boundary conditions which are used for the current experimentation are specified in Table 3. The numerical model for fluid flow and heat transfer using an newly developed solar air heater with artificially roughened is should be followed the given expectations (a) fluid flow and heat transfer are in steady state (b) a completely developed flow of hydraulically and thermally (both should be in same steady state conditions), (c) duct wall thermal conductivity, roughness material and absorber plate all are self-governing of temperature (d) absorber plate, the duct wall and roughness material are standardised and isotropic. The remaining all other walls are set adiabatic and the ambient temperature of air is set at 300 K for all iterations. The numerical model was used to investigate the 2D fluid flow phenomenon and heat transfer rates of solar air heater of techniques. Furthermore, advanced software package tools of 2D CFD model used for simulation for analysing newly developed with an artificially roughened solar air heater through the viable CFD simulation tool in ANSYS FLUENT to resolve the conservation equations for the required parameters of momentum, mass and energy [28].

| S.no | Boundary       | Type            | Properties          |
|------|---------------|-----------------|---------------------|
| 1    | Inlet         | Velocity inlet  | 0.5 m/s             |
| 2    | Outlet        | Pressure outlet | Gauge pressure=0    |
| 3    | Absorber plate| Heat flux       | 1100 w/m²k          |
| 4    | Wall          | Insulated       | Insulated           |

### 3. Results and Discussions

The solar air heaters are designed usually swirl or vortex flow alternative to laminar flow to improve the mass and heat transfer equipment’s in solar products with the allowance of minimal loss of pressure. To use the artificial roughness, the shape of the winglets is categorised as triangular, rectangular and delta types are used to generate the vortex flow. Whereas, turbulent field synergy can achieve the flow patterns in suitable form of optimal conditions. In the model a combination of heat transfer added to the solid baffles. The designed solid baffles of square and equilateral triangle roughness are most efficient and commonly preferred heat transfer techniques. The CFD simulation analysis of 2-dimensional duct of solar air heater was conducted to analyse the influence of pitch of the roughness ration of (P/e), comparative height of the roughness and Reynolds number. Also, flow friction characteristics were used for the artificial roughness in the form of square and equilateral triangle rib roughness. The output of the results obtained from simulation are smooth duct with almost similar flow and thermal boundary conditions. It is also observed that the increase in Reynolds number the outlet temperature gradually decreasing in irrespective of the P/e ratios. The ratio of P/e values have significant effect on thermal behaviour and heat transfer flow to the solar heaters. The P/e ratios and values of the ratios are achieved similar condition when the minimum and maximum values are below 8 and above 13.33, respectively. The temperature distribution almost similar trend followed for these two conditions. Whereas, when the P/e ratio have optimum level the results are entirely different and achieved the maximum temperature distribution trend at P/e ratio of 10%. The results obtained at
Figure 3 The change in temperature with the relationship between P/e ratio and Reynolds number.

Figure 4 Temperature distribution of 2D square rib roughness (a) square and (b) equilateral triangle
Figure 5 The turbulent kinetic energy contour plot of Re = 10,000 and P/e = 10 at a relative roughness height of the ribs of (a) square and (b) equilateral triangle.

ratio of P/e is 10 and is unlikely the values of P/e are 8 and 13.3 and are always higher than the minimum and maximum values at any point of Reynolds number. However, the plot of the average values of non-dimensional Reynolds number of various standards of relative roughness pitch and also for relative roughness height at constant values of (ratio of P/e), has been illustrated in figure 3. The average Reynolds number rises with reverence to smooth duct by place in square rib on the bottom of the solar air heater duct of the absorber plate. It is observed that the escalation in Reynolds number resulting of reduction of laminar sub-layer thickness. Therefore, the creation of turbulence on the local wall owed to flow parting and rearranging between the absorber plate ribs for which enriches the heat transfer rate. The development of artificial roughness rib techniques aid to generate vortices and hence influence of flow mixing, which is subsidize to avoid or remove the generated heat commencing the absorber plate which has got heated. It is the fact that rise of Reynolds number resulted in increasing of turbulent dissipation rate and kinetic energy which primes to the enhancing of Nusselt number and turbulent intensity.

The simulated CFD results exhibits in contour graphs which is easy to understand the heat transfer phenomena and turbulent kinetic energy efficiently. These graphs are able to show the temperature, pressure and friction factor values where the maximum and minimum values distributed and its intensity clearly. The variation in intensity of the colour over the ribs roughness thickness volume. The corner of the square shaped rib roughness exhibits highest intensity of the temperature values and its intensity gradually decreasing towards underside of the ribs. The maximum volume fraction of temperature distribution is higher for equilateral triangle shaped type roughness ribs over the square type roughness ribs. The temperature distribution simulation graphs for altered values for height of relative roughness values are at a constant number Reynolds number 10,000 and the ratio of P/e is 10,
Figure 6 Variation of outlet temperature along with different rib roughness and Reynolds number of square and equilateral triangle

Table 4 The output of the simulation results of different roughness shaped used

| Parameters                  | Square rib | Equilateral triangle rib |
|-----------------------------|------------|--------------------------|
| Heat input (W/m²)           | 1100       | 1100                     |
| Inlet temperature (K)       | 300        | 300                      |
| Outlet temperature (K)      | 322.84     | 323.89                   |
| Increase in outlet temperature (%) | 7.61      | 7.96                     |
| Improvement in heat transfer coefficient | 3.21      | 3.35                     |

is illustrated in figure 4. The maximum value of temperature is obtained adjacent to the top side of the wall and reducing towards bottom side of the rib. However the temperature distribution of phenomena as of heat transfer can be implicit effectively by the CFD results of contour simulation results of the turbulent kinetic energy. Figure 5 exhibits the simulation results of simulation results of flow lines from the of turbulent kinetic energy for dissimilar roughness height of the values and Reynolds number of 10,000 at value of 10 ratio between relative roughness to pitch ratio (P/e). The simulation clearly illustrates the significant variation in the turbulent kinetic energy for the square and equilateral triangle shaped roughness heights. The highest intensity values are recorded at the just below the square shape and bottom corner of the square shaped roughness height (see figure 5a). In addition, it is gradually reduced to bottom of the roughness height. The highest value of 6.95 J/kg and minimum value is 2.11 J/kg of turbulent kinetic energy was recorded. Whereas, equilateral triangle shaped roughness heights shows the highest value of $7.47 \times 10^{-2}$ and minimum value of $3.26 \times 10^{-2}$ has been obtained. Unlike square shaped roughness height, triangle shaped contour plot (see figure 5b) illustrates the very less amount of
intensity factor of turbulent kinetic energy. It is worth to mention that the recorded values are not all dangers to operating conditions and surely the efficiency is much higher compared to the square shaped roughness height on the solar air heater ribs. Moreover, the highest values and intensity of turbulent kinetic energy created in mid-section of the triangle shaped roughness heights. The upper and lower parts to the mid-section are contained the similar view of the intensity. The appearance of this results led to more efficient results in the heat transfer point of view. However, in both the conditions found to be that the turbulent kinetic energy reductions with the increase in distance from the wall. As depicted in figure 3, with the increasing of value as Reynolds number resulted in roughness elements start to produce above the laminar sublayer. It is also noticed that as the Reynolds number increases, laminar sublayer gradually decreases. As perceived in figure 5b, the highest intensity of turbulent kinetic energy at mid-section, the heat removal also influenced by the local contribution (thickness) of the by vortices instigating from the roughness [3,29], and thus cause to improve in the heat transfer flow rate of artificially roughness surface when make the comparison with the original surface. It is also important to determine the velocity of the inlet flow and the outlet flow velocity with the purpose of understand the consequence of roughness shape and height. As the roughened duct length of the of solar heater considers, it shows to be determine to the flow velocity at the inlet value is less as compared to the outlet value of the duct. It is owing to the velocity flow acceleration in the form of stream-wise direction [3,15]. It is also observed that the instantaneous flow velocity contours of the equilateral triangular roughness ribs is regular and smooth and concentrated path followed, whereas, in square shaped roughness ribs velocity contours illustrates the irregular way, due to the wide path of the square shaped roughness and less concentration vertices can be expecting that it is the nature of the turbulence for square sectioned transverse ribs. Figure 6 illustrates the outlet temperature of the two types of artificial roughened ribs pertaining to Reynolds number. The outlet temperature at all the conditions lowers with the rising in Reynolds number. The equilateral triangle shaped roughened ribs quite higher than the square shaped ribs, it is because of the concentrated flow of velocity from the triangle groove. The friction factor values are changes between these two types of shaped roughens ribs. In both the conditions it is found that the increase Reynolds number the led to decrease in friction factor. It is owing to the lowering of viscous sublayer with further higher values in Reynolds number as explained the phenomena from the figure 3. The effect of artificial roughened solar heaters are clearly indicates the producing higher friction factor than the smooth surface of the solar heaters. The change in friction factor also affected by the height of the roughness for a constant number shows the relative roughness shape of the pitch size [30]. The friction factor values were observed at a height of the relative roughness is 0.045 at a dimensionless Reynolds number of 3010. The overall results shows the square and equilateral triangle shaped roughened ribs, and are given in table 4, which suggests that the triangle shaped artificial roughened ribs higher performance than the squared ribs.

4. Conclusions

The solar air heater roughened with equilateral triangle and transverse sections of square rib surface roughness over the absorber plate has been analysed by 2D view of the CFD analysis with a three different ratios of P/e and Reynolds number from 3000 to 20000. The contour plots of the temperature distribution, turbulent kinetic energy maps are used to analyse the consequence of solar air heaters with creation of artificially roughened surfaces. From the simulation results some inference are realised that the maximum ratio for friction factor determined 3.2 with respect to the artificially developed height of the roughness is 0.045 for the constant constrain of Reynolds number of 3010 designed for the use of conditions. Among the three P/e ratios, the value of 10 showed the optimum and efficient results for the square shaped roughened ribs. The ratio of roughness height and pitch has strong effect on the flow pattern and enactment of the solar air heater of artificially roughened surfaces. The outlet temperature of the two of the absorber plates decreased with accumulative of Reynolds number. The equilateral triangle transverse cross sectioned of the rib roughness of the absorber plate showed good outcomes over the square sectioned absorber plate.
References

[1] Gawande V B, Dhole A S, Zodpe D B and Chamoli S 2016 A review of CFD methodology used in literature for predicting thermo-hydraulic performance of a roughened solar air heater Renew. Sustainable. Energy. Rev. 54 550-605.

[2] Singh S, Singh B, Hans V S, Gill R S 2015 CFD (computational fluid dynamics) investigation on Nusselt number and friction factor of solar air heater duct roughened with non-uniform cross-section transverse rib Energy. 84 509-17.

[3] Yadav A S and Bhagoria J L 2013 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 Article ID 827131, 12 pages. doi:10.1155/2013/827131

[4] Yadav A S and Bhagoria J L 2013 Renewable energy sources-an application guide Int. J. Energy. Sci. 3 70-90.

[5] Duffie J A and Bekman W A 1980 Solar engineering of thermal processes 2nd ed. New York Wiley.

[6] Hans V S, Saini R P and Saini J S 2009 Performance of artificially roughened solar air heaters-a review Renew. Sustainable. Energy. Rev. 13 1854-69.

[7] Chandra P R, Alexander C R, and Han J C 2003 Heat transfer and friction behaviors in rectangular channels with varying number of ribbed walls Int. J. Heat Mass Transfer. 46 481-95.

[8] Muralimohan C H, Haribabu S, Reddy Y H, Muthupandi V and Sivaprasad K 2015 Joining of AISI 1040 steel to 6082-T6 aluminium alloy by friction welding J. Adv. Mech. Eng. Sci. 1(1) 57-64. http://dx.doi.org/10.18831/james.in/2015011006

[9] Muralimohan C H, Muthupandi V and Sivaprasad K 2014 The influence of aluminium intermediate layer in dissimilar friction welds Inter. J. Mater. Res. 105 350-57.

[10] Cheepu M, Muthupandi V, Srinivas B and Sivaprasad K 2018 Development of a friction welded bimetallic joints between titanium and 304 austenitic stainless steel Techno-Societal 2016, International Conference on Advanced Technologies for Societal Applications ICATSA 2016 ed Pawar P M, Ronge B P, Balasubramaniam R and Seshabhattar S (Springer, Cham) Chapter 73 709-17. https://doi.org/10.1007/978-3-319-53556-2_73

[11] Bhushan B and Singh R 2010 A review on methodology of artificial roughness used in duct of solar air heaters Energy. 35 202-12.

[12] Cheepu M, Ashfaq M and Muthupandi V 2017 A new approach for using interlayer and analysis of the friction welding of titanium to stainless steel Trans. Indian. Inst. Met. 70 2591-600. https://doi.org/10.1007/s12666-017-1114-x

[13] Muralimohan C H, Ashfaq M, Ashiri R, Muthupandi V and Sivaprasad K 2016 Analysis and characterization of the role of Ni interlayer in the friction welding of titanium and 304 austenitic stainless steel Metall. Mater. Trans. A. 47 347-59.

[14] Cheepu M M, Muthupandi V and Loganathan S 2012 Friction welding of titanium to 304 stainless steel with electroplated nickel interlayer Mater. Sci. Forum. 710 620-25.

[15] Kumar A, Saini R P and Saini J S 2012 Heat and fluid flow characteristics of roughened solar air heater ducts-a review Renew. Energy. 47 77-94.

[16] Muralimohan C H, Haribabu S, Reddy Y H, Muthupandi V and Sivaprasad K 2014 Evaluation of microstructures and mechanical properties of dissimilar materials by friction welding Procedia. Mater. Sci. 5 1107-13.

[17] Muralimohan C H, Muthupandi V and Sivaprasad K 2014 Properties of friction welding titanium-stainless steel joints with a nickel interlayer Procedia. Mater. Sci. 5 1120-29.

[18] Muralimohan C H and Muthupandi V 2013 Friction welding of type 304 stainless steel to CP titanium using nickel interlayer Adv. Mater. Res. 794 351-57.

[19] Cheepu M, Muthupandi V and Che W S 2018 Improving mechanical properties of dissimilar material friction welds Appl. Mech. Mater. 877 157-62. doi:10.4028/www.scientific.net/AMM.877.157
[20] Venkateswarulu D, Cheepu M, Krishnaja D and Muthukumaran S 2018 Influence of water cooling and post-weld ageing on mechanical and microstructural properties of the friction-stir welded 6061 aluminium alloy joints Appl. Mech. Mater. 877 163-76. doi:10.4028/www.scientific.net/AMM.877.163

[21] Cheepu M, Haribabu S, Ramachandraiah T, Srinivas B, Venkateswarulu D, Karna S, Alapati S and Che W S 2018 Fabrication and analysis of accumulative roll bonding process between magnesium and aluminum multi-layers Appl. Mech. Mater. 877 183-89. doi:10.4028/www.scientific.net/AMM.877.183

[22] Devireddy K, Devuri V, Cheepu M and Kumar B K 2018 Analysis of the influence of friction stir processing on gas tungsten arc welding of 2024 aluminum alloy weld zone Int. J. Mech. Prod. Eng. Res. Dev. 8(1) 243-52. DOI: 10.24247/ijmperdfeb201828

[23] Cheepu M, Venkateswarlu D, Mahapatra M M and Che W S 2017 Influence of heat treatment conditions of Al-Cu aluminum alloy on mechanical properties of the friction stir welded joints. Korean Welding and Joining Society, 11, 264-264. http://www.dbpia.co.kr/Journal/ArticleDetail/NODE07278590

[24] Karmare S V and Tikekar A N 2010 Analysis of fluid flow and heat transfer in a rib grit roughened surface solar air heater using CFD Sol. Energy. 84 409-17.

[25] Yadav A S and Bhagoria J L 2013 A CFD based heat transfer and fluid flow analysis of a conventional solar air heater J. Eng. Sci. Manage. Educ. 6 137-46.

[26] Yadav A S and Bhagoria J L 2013 A CFD based heat transfer and fluid flow analysis of a solar air heater provided with circular transverse wire rib roughness on the absorber plate Energy. 55 1127-42.

[27] ASHRAE Standard 93 Method of testing to determine the thermal performance of solar collectors American society of heating, refrigeration and air conditioning engineers Atlanta GA30329 2003.

[28] http://www.ansys.com/academic/free-student-products

[29] Webb R L and Eckert E R G 1972 Application of rough surfaces to heat exchanger design Int. J. Heat. Mass. Transfer. 15 1647-58.

[30] Chandra P R, Alexander C R and Han J C 2003 Heat transfer and friction behaviors in rectangular channels with varying number of ribbed walls Int. J. Heat. Mass. Transfer. 46 481-95.