Ashok Kumar Panchtilak, 2022, 10:1
ISSN (Online): 2348-4098
ISSN (Print): 2395-4752

A CFD Investigation of Flows through an Artificially Roughened Solar Air Heater

PG Scholar Ashok Kumar Panchtilak, Associate Prof. Dr. Irshad Ahmad Khan
Department of Mechanical Engineering,
Sagar Institute of Research & Technology,
Bhopal (M.P), India

Abstract- Solar air heaters, because of their simple in design, is cheap and most widely used collection devices of solar energy. The thermal efficiency of a solar air heater is significantly low because of the low value of the convective heat transfer coefficient between the absorber plate and the air, leading to high absorber plate temperature and high heat losses to the surroundings. This paper presents the study of heat transfer in a solar air heater by using Computational Fluid Dynamics (CFD). The effect of Reynolds number on Nusselt number is investigated. A commercial finite volume package ANSYS FLUENT 16 is used to analyze and visualize the nature of the flow across the duct of a solar air heater.

Keywords- Solar Air Heater, Heat transfer, Pressure Drop, CFD.

I. INTRODUCTION

Solar air heater is one of the basic equipment through which solar energy is converted into thermal energy. Solar air heaters, because of their simple in design, are cheap and most widely used collection devices of solar energy. The main application of solar air heaters is space heating, seasoning of timber, curing of industrial products and these can also be effectively used for curing/drying of concrete/clay building components.

A conventional solar air heater generally consists of an absorber plate, a rear plate, insulation below the rear plate, transparent cover on the exposed side, and the air flows between the absorbing plate and rear plate. A solar air heater is simple in design and requires little maintenance. However, the value of the heat transfer coefficient between the absorber plate and air is low and this results in a lower efficiency. For this reason, the surfaces are sometimes roughened in the air flow passage.

The concept of artificial roughness was first applied by Joule [1] to enhance heat transfer coefficients for in-tube condensation of steam and since then many experimental investigations were carried out on the application of artificial roughness in the areas of cooling of gas turbine, electronic equipments, nuclear reactors, and compact heat exchangers etc. Nunner [2] was the first who developed a flow model and likened this model to the temperature profile in smooth tube flow at increased Prandtl number.

The proposed flow model predicts that roughness reduces the thermal resistance of the turbulence dominated wall region without significantly affecting the viscous region. The argument was quantified by using the Prandtl analogy and replacing Pr by (f/fs)Pr. This model predicts that value of St/Sts decreases with increase in Prandtl number. The proposed flow model also predicts that St/Sts, is independent of the roughness type. A friction correlation for flow over sand-grain roughness was developed by Nikuradse [3].

Based on law of the wall similarity, Nikuradse presented the pressure drop results in terms of roughness function R and roughness Reynolds number e+.

Dipprey and Sabersky [4] developed a heat-momentum transfer analogy relation for flow in a sand-grain roughened tube and achieved excellent correlation of their data. The concept proposed by Dipprey and Sabersky was so common and it can be applied to any roughness for which law of the wall similarity holds. Prasad and Mullick [5] were the first who introduced the application of artificial roughness in the form of small diameter wire attached on the underside of absorber plate to improve the thermal performance of solar air heater for drying purposes.

After Prasad and Mullick's [5] work a number of experimental investigations of solar air heater involving roughness elements of different shapes, sizes and orientations with respect to flow direction have been carried out in order to obtain an optimum arrangement of roughness element geometry [6-7].

Chaube et al. [8] conducted a CFD based analysis of heat transfer and fluid flow characteristics of an artificially roughened solar air heater. Kumar and Saini [9] performed a CFD analysis of fluid flow and heat transfer characteristics of a solar air heaters having arc shaped artificial roughness on the absorber plate.

Karmare and Tikekar [10] carried out CFD investigation of fluid flow and heat transfer in a solar air heater duct with metal grit ribs as roughness elements on the absorber plate. Yadav and Bhagoria [11] conducted a numerical prediction to study only heat transfer behavior of a rectangular duct of a solar air heater having triangular rib roughness on the absorber plate.

Yadav and Bhagoria [12] presented the numerical prediction of fluid flow and heat transfer in a conventional solar air heater by CFD. A commercial finite volume package ANSYS FLUENT 12.1 was used to analyze the nature of the flow across the duct of a conventional solar air heater. Yadav and Bhagoria [13-14] presented a detailed literature survey about different CFD investigations on artificially roughened solar air heater. Several other types of turbulators elements were used extensively to improve the heat transfer characteristics [15-69].

The aim of present study is to analyze CFD investigation of ribbed rectangular channel on the flow and heat transfer.

The aim of our study is to improve the prediction of the flow in the solar air heater. A near-wall function for TKE will be implemented in Computational Fluid Dynamics code Fluent. Second order upwind and SIMPLE algorithm were used to discretize the governing equations. The FLUENT software solves the following mathematical equations which governs fluid flow, heat transfer and related phenomena for a given physical problem.

II. CFD MODELING AND ANALYSIS

Computational Fluid Dynamics (CFD) is the science of determining numerical solution of governing equation for the fluid flow whilst advancing the solution through space or time to obtain a numerical description of the complete flow field of interest.

The equation can represent steady or unsteady, Compressible or Incompressible, and in viscid or viscous flows, including non-ideal and reacting fluid behavior. The particular form chosen depends on intended application. The state of the art is characterized by the complexity of the geometry, the flow physics, and the computing time required obtaining a solution.

The 2-D computational domain used for CFD analysis having the height (H) of 20 mm and width (W) 100 mm as shown in Fig. 1. In the present analysis, a 2-dimensional computational domain of artificially roughened solar air heater has been adopted which is similar to computational domain of Yadav and Bhagoria [14].

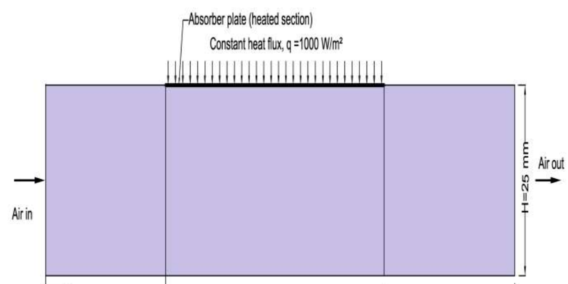

Fig 1. 2-D computational domain.

After defining the computational domain, non-uniform mesh is generated. In creating this mesh, it is desirable to have more cells near the plate because we want to resolve the turbulent boundary layer, which is very thin compared to the height of the flow field. After generating mesh, boundary conditions have been specified. We will first specify that the left edge is the duct inlet and right edge is the duct outlet. Top edge is top surface and bottom edges are inlet length, outlet length and solar plate.

All internal edges of rectangle 2D duct are defined as turbulator wall. Meshing of the domain is done using ANSYS ICEM CFD V16 software. Since low-Reynolds-number turbulence models are employed, the grids are generated so as to be very fine.

Present non-uniform quadrilateral mesh contained 161,568 quad cells with non-uniform quad grid of 0.22 mm cell size. This size is suitable to resolve the laminar sub-layer. For grid independence test, the number of cells is varied from 103,231 to 197,977 in five steps. It is found that after 161,568 cells, further increase in cells has less than 1% variation in Nusselt number and friction factor value which is taken as criterion for grid independence.

To select the turbulence model, the previous experimental study is simulated using different low Reynolds number models such as Standard k-ω model, Renormalization-group kε model, Realizable kε model and Shear stress transport k-ω model. The results of different models are compared with experimental results. The RNG k-ε model is selected on the basis of its closer results to the experimental results. The working fluid, air is assumed to be incompressible for the operating range of duct since variation is very less.

The mean inlet velocity of the flow was calculated using Reynolds number. Velocity boundary condition has been considered as inlet boundary condition and outflow at outlet. Second order upwind and SIMPLE algorithm were used to discretize the governing equations. The FLUENT software solves the following mathematical equations which governs fluid flow, heat transfer and related phenomena for a given physical problem.

III. RESULTS AND DISCUSSIONS

Results are presented in form of graphs, representing the average Nusselt number at different Reynolds numbers, and in form of temperature and velocity contours at particular sections for a fixed Reynolds number.

Fig. 2 shows the effect of Reynolds number on average Nusselt number for different values of relative roughness height (e/D) and fixed value of pitch. The average Nusselt number is observed to increase with increase of Reynolds number due to the increase in turbulence intensity caused by increase in turbulence kinetic energy and turbulence dissipation rate.

The roughened duct with relative roughness height of 0.06 provides the highest Nusselt number at a Reynolds number of 18000. The roughened duct with relative roughness height (e/d) of 0.015 provides the lowest Nusselt number at a Reynolds number of 3800. The maximum enhancement of average Nusselt number is found to be 2.78 times that of smooth duct for relative roughness height of 0.06.

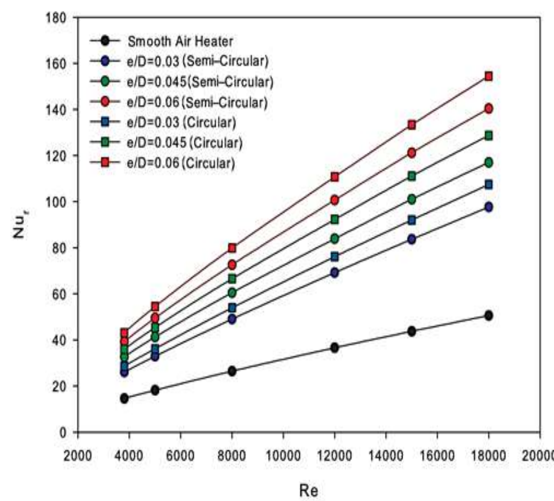

Fig 2. Variation in Nusselt number.

The heat transfer phenomenon can be observed and described by the contour plot of turbulence intensity. The intensities of turbulence are reduced at the flow field near the rib and wall and a high turbulence intensity region is found between the adjacent ribs close to the main flow which yields the strong influence of turbulence intensity on heat transfer enhancement.

Fig 3 shows that the friction factor decreases with the increasing values of the Reynolds number in all cases as expected because of the suppression of laminar sub-layer for fully developed turbulent flow in the duct.

The maximum and minimum value of friction factor occurs at relative roughness height (e/d) of 0.06 and 0.015 respectively for the range of parameters investigated. It is also observed that the maximum enhancement of average friction factor is found to be 4.24 times that of smooth duct for relative roughness height of 0.06 at a Reynolds number of 3800.

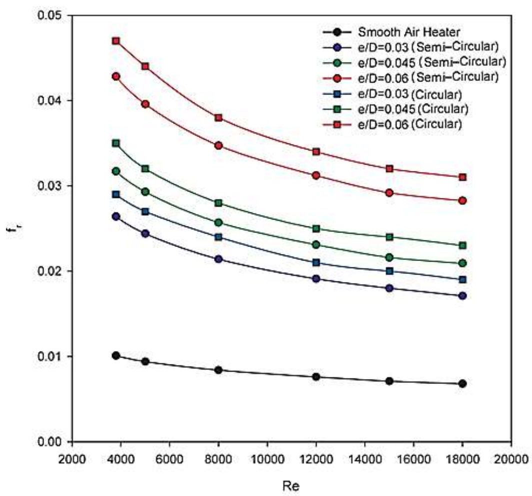

Fig 3. Variation in Friction factor.

The fluid friction phenomenon can be observed and described by the contour of pressure for circular rib. The contour plot of pressure is shown in Fig. 4. Similar pattern is obtained for contour plot of pressure for semi-circular rib. The flow friction phenomenon can be observed and described by the contour plot of pressure for semi-circular rib.

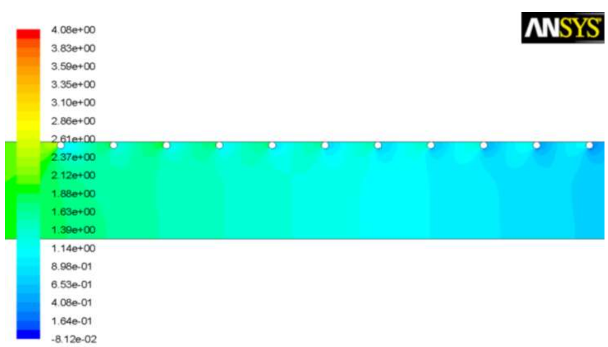

Fig 4. Contour plot.

In order to validate the present numerical model, the results are compared with available experimental results. Literature search in the area of artificially roughened solar air heater also reveal that the optimum value of relative roughness height generally lies between 0.03-0.047.

The comparison of optimum value of relative roughness height between present CFD simulation and available experimental/widely accepted numerical results. On comparison, it has been observed that the optimum value of relative roughness height for present CFD model is found to be 0.045 for circular and semi-circular sectioned rib. The optimum value of relative roughness height from present CFD investigation is found to fall in-between the accepted range i.e. 0.033 and 0.043. It can be seen that there is a good agreement between CFD and experimental/numerical results.

IV. CONCLUSION

The effect of relative roughness pitch and Reynolds number on the heat transfer coefficient and friction factor have been studied. CFD Investigation has been carried out in medium Reynolds number flow (Re = 3800–18,000).

Based on the CFD investigations of heat and fluid flow in a rectangular duct with protrusions as roughness element on one broad wall subjected to a uniform heat flux the following conclusions were drawn:

- Average Nusselt number increases with an increase of Reynolds number. The maximum value of average Nusselt number is found at relative roughness height of 0.06 at a higher Reynolds number, 18,000.
- Average friction factor decreases with an increase of Reynolds number. The maximum value of average friction factor is found at relative roughness height of 0.06 at a lower Reynolds number, 3800.
- The Renormalization-group (RNG) k-ε turbulence model predicted very close results to the experimental results, which yields confidence in the predictions done by CFD analysis in the present study. RNG k-ε turbulence model has been validated for smooth duct and grid independence test has also been conducted to check the variation with increasing number of cells.
- The discrepancy between available experimental data and present computational results is less than ±10%. It can therefore be concluded that the present computational results are reasonably satisfactory.

REFERENCES

[1] Joule JP. On the surface condensation of steam. Philosophical Transactions of the Royal Society of London 1861; 151: 133-160.

[2] Nunner W. Heat transfer and pressure drop in rough pipes. VDI-Forsch 1956; 22: 445-B, English trans, AERE Lib./Trans 1958; 786.

[3] Nikuradse J. Laws of flow in rough pipes. NACA Technical Memorandum 1950; 1292.

[4] Dippery DF, Sabersky RH. Heat and momentum transfer in smooth and rough tubes at various Prandtl number. Int J Heat Mass Transfer 1963; 6: 329–53.

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

[6] Hans VS, Saini RP, Saini JS. Performance of artificially roughened solar air heaters- A review. Renewable and Sustainable Energy Reviews 2009; 13: 1854–1869.

[7] Bhushan B, Singh R. A review on methodology of artificial roughness used in duct of solar air heaters. Energy 2010; 35: 202–212.

[8] Chaube, P.K. Sahoo, S.C. Solanki, Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater, Renewable Energy 31 (2006) 317-31.

[9] S. Kumar, R.P. Saini, CFD based performance analysis of a solar air heater duct provided with artificial roughness, Renewable Energy 34 (2009) 1285-91.

[10] S.V. Karmare, A.N. Tikekar, Analysis of fluid flow and heat transfer in a rib grit roughened surface solar air heater using CFD, Solar Energy 84 (2010) 409–17.

[11] A.S. Yadav, J.L. Bhagoria, A CFD analysis of a solar air heater having triangular rib roughness on the absorber plate, International Journal of ChemTech Research 5(2) (2013) 964-71.

[12] A.S. Yadav, J.L. Bhagoria, A CFD based heat transfer and fluid flow analysis of a conventional solar air heater, Journal of Engineering Science and Management Education 6(2) (2013) 137-146.

[13] A.S. Yadav, J.L. Bhagoria, Heat transfer and fluid flow analysis of solar air heater: a review of CFD approach, Renewable and Sustainable Energy Reviews 23 (2013) 60-79.

[14] A.S. Yadav, J.L. Bhagoria. 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, Elsevier, 55 (2013) 1127-1142. team. Philosophical Transactions of the Royal Society of London 1861; 151: 133-160.

[15] Hans VS, Saini RP, Saini JS. Performance of artificially roughened solar air heaters- A review. Renewable and Sustainable Energy Reviews 2009; 13: 1854–1869.

[16] Bhushan B, Singh R. A review on methodology of artificial roughness used in duct of solar air heaters. Energy 2010; 35: 202–212.

[17] Yadav, A. S., and Thapak, M. K. "Artificially roughened solar air heater: A comparative study," International Journal of Green Energy vol. 13, no. 2, 2016, pp. 143-172.

[18] Yadav Anil Singh, Thapak MK. Artificially roughened solar air heater: experimental investigations. Renewable and Sustainable Energy Reviews 2014; 36: 370-411.

[19] Yadav AS, Bhagoria JL. A CFD based 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 2014; 70: 1016–1039.

[20] Yadav AS, Bhagoria JL. Heat transfer and fluid flow analysis of solar air heater: a review of CFD approach. Renewable and Sustainable Energy Reviews 2013; 23: 60-79.

[21] Yadav AS, Bhagoria JL. 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 2013; 55: 1127-42.

[22] Yadav AS, Bhagoria JL. Modeling and simulation of turbulent flows through a solar air heater having square sectioned transverse rib roughness on the absorber plate. The Scientific World Journal 2013. DOI: 10.1155/2013/827131.

[23] Yadav AS, Bhagoria JL. A numerical investigation of turbulent flows through an artificially roughened solar air heater. Numerical Heat Transfer A 2014; 65: 679–698.

[24] Yadav AS, Bhagoria JL. Numerical investigation of flow through an artificially roughened solar air heater. International Journal of Ambient Energy 2013. DOI: 10.1080/01430750.2013.823107 (Article in press).

[25] Yadav AS, Bhagoria JL. Heat transfer and fluid flow analysis of an artificially roughened solar air heater: a CFD based investigation. Frontiers in Energy 2014; 8(2): 201-211.

[26] Yadav AS, Bhagoria JL. A CFD analysis of a solar air heater having triangular rib roughness on the absorber plate. International Journal of ChemTech Research 2013; 5(2): 964-71.

[27] Yadav AS, Bhagoria JL. A CFD based heat transfer and fluid flow analysis of a conventional solar air heater. Journal of Engineering Science and Management Education 2013; 6(2): 137-46.

[28] Yadav AS, Bhagoria JL. A numerical investigation of square sectioned transverse rib roughened solar air heater. International Journal of Thermal Sciences 2014; 79: 111-131.

[29] Ahn S. W. 2001. The effects of roughness types on friction factors and heat transfer in roughened rectangular duct. Int. J. Heat and Mass transfer. 28, 933-942.

[30] Chandra P.R., Alexander C.R.,Han J.C. 2003. Heat transfer and friction behaviors in rectangular channels with varying number of ribbed walls. Int. J. Heat and Mass transfer. 46, 481–495.

[31] Tanda Giovanni. 2004. Heat transfer in rectangular channels with transverse and V-shaped broken ribs. Int. J. Heat and Mass transfer. 47, 229–243.

[32] Tariq Andallib, Singh Kamlesh and Panigrahi P.K. 2002. Detailed measurement of heat transfer and flow characteristics in rectangular duct with rib turbulators mounted on the bottom surface. Engineering Turbulence Modelling and Experiments. 5, 445 - 454.

[33] Won S.Y., Ligrani P.M. 2004. Comparisons of flow structure and local Nusselt numbers in channels with parallel- and crossed-rib turbulators. 47, 1573–1586.

[34] Wang Lieke, Sunden Bengt. 2004.An experimental investigation of heat transfer and fluid flow in a rectangular duct with broken v-shaped ribs. Experimental Heat Transfer. 17,243–259.

[35] Liu Ye-Di, Diaz L. A., Suryanarayana N. V. 1984. Heat transfer enhancement in air heating fiat-plate solar collectors. Trans. ASME, J. of Solar Energy Engg.106, 358-363.

[36] Yadav, A. S. "Augmentation of heat transfer in double pipe heat exchanger using full & half-length twisted tape inserts," CSVTU Research Journal vol. 1, no. 1, 2008, pp. 67-73.

[37] Yadav, A. S. "Experimental investigation of heat transfer performance of double pipe U-bend heat exchanger using full length twisted tape," International Journal of Applied Engineering Research vol. 3, no. 3, 2008, pp. 399-407.

[38] Yadav, A. S. "Effect of half-length twisted-tape turbulators on heat transfer and pressure drop characteristics inside a double pipe u-bend heat exchanger," Jordan Journal of Mechanical and Industrial Engineering vol. 3, no. 1, 2009, pp. 17-22.

[39] Yadav, A. S., and Bhagoria, J. L. "An Economic Analysis of a Solar System," Corona Journal of Science and Technology vol. 2, no.1, 2013, pp.3-7.

[40] Yadav, A. S., and Bhagoria, J. L. "Renewable Energy Sources-An Application Guide: Energy for Future," International Journal of Energy Science vol. 3, no. 2, 2013, pp. 70-90.

[41] Bhaskar, B., Bhadoria, R. S., and Yadav, A. S. "Transportation system of coal distribution: a fuzzy logic approach using MATLAB," Corona Journal of Science and Technology vol. 2, no. 3, 2013, pp. 20-30.

[42] Thapak, M. K., and Yadav, A. S. "A comparative study of artificially roughened solar air heater," Corona Journal of Science and Technology vol. 3, no. 2, 2014, pp. 19-22.

[43] Thapak, M. K., and Yadav, A. S. "Analysis approaches of an artificially roughened solar air heater," Corona Journal of Science and Technology vol. 3, no. 2, 2014, pp. 23-27.

[44] Yadav, A. S. "CFD investigation of effect of relative roughness height on Nusselt number and friction factor in an artificially roughened solar air heater," Journal of the Chinese Institute of Engineers vol. 38, no. 4, 2015, pp. 494-502. doi: 10.1080/02533839.2014.998165

[45] Qureshi, T. A., and Yadav, A. S. "Heat transfer enhancement by swirl flow devices," International Journal of Current Engineering and Scientific Research vol. 3, no. 1, 2016, pp. 122-127.

[46] Khan, I. A., Yadav, A. S., and Shakya, A. K. "Prognosis and diagnosis of cracks of cantilever composite beam by vibration analysis and hybrid AI technique," International Journal of Advanced Technology in Engineering and Science vol. 4, no. 1, 2016, pp. 16-23.

[47] Yadav, A. S., Khan, I. A., and Bhaisare, A. K. "CFD Investigation of Circular and Square Sectioned Rib Fitted Solar Air Heater," International Journal of Advance Research in Science and Engineering (IJARSE) vol. 5, no. 01, 2016, pp. 386-393.

[48] Yadav, A. S., and Singh, S. "A CFD analysis of an artificially roughened solar air heater," RGI International Journal of Applied Science & Technology vol. 10 & 11, no. 01,02, 2016, pp. 1-6.

[49] Dwivedi, S., Yadav, A. S., and Badoniya, P. "Study of Thin-Walled Cone by Using of Finite Element Analysis in Deep Drawing," International Journal of Advanced Technology in Engineering and Science vol. 5, no. 5, 2017, pp. 587-591.

[50] Qureshi, T. A., Yadav, A. S., and Jain, A. "Recent alternative sources of energy- A brief review,"

RGI International Journal of Applied Science & Technology vol. 12 & 13, no. 01 & 02, 2017, pp. 70-71.

[51] Prasad, R., Yadav, A. S., Singh, N. K., and Johari, D. "Heat Transfer and Friction Characteristics of an Artificially Roughened Solar Air Heater," Advances in Fluid and Thermal Engineering, Lecture Notes in Mechanical Engineering. Springer, Singapore, 2019, pp. 613-626.

[52] Yadav, A. S., Singh, D. K., Soni, G., and Siddiqui, D. A. "Artificial Roughness and Its Significance on Heat Transfer of Solar Air Heater: An Assessment," International Journal of Scientific Research and Engineering Development vol. 3, no. 2, 2020, pp. 1134-1149.

[53] Yadav, A. S., and Sharma, S. K. "Numerical Simulation of Ribbed Solar Air Heater," Advances in Fluid and Thermal Engineering, Lecture Notes in Mechanical Engineering. Springer, Singapore, 2021, pp. 549-558.

[54] Yadav, A. S., Shrivastava, V., Ravi Kiran, T., and Dwivedi, M. K. "CFD-Based Correlation Development for Artificially Roughened Solar Air Heater," Recent Advances in Mechanical Engineering, Lecture Notes in Mechanical Engineering. Springer, Singapore, 2021, pp. 217-226.

[55] Shrivastava, V., Yadav, A. S., and Shrivastava, N. "Comparative Study of the Performance of Double-Pass and Single-Pass Solar Air Heater with Thermal Storage," Recent Advances in Mechanical Engineering, Lecture Notes in Mechanical Engineering. Springer, Singapore, 2021, pp. 227-237.

[56] Shrivastava, V., Yadav, A. S., and Shrivastava, N. "Thermal performance assessment of greenhouse solar dryer," Recent Trends in Thermal Engineering, Lecture Notes in Mechanical Engineering Springer, Singapore, 2022, pp. 75-82.

[57] Chouksey, V. K., Yadav, A. S., Raha, S., Shrivastava, V., and Shrivas, S. P. "A theoretical parametric analysis to optimize the bed depth of packed bed solar air collector," International Journal of Green Energy, 2021, pp. 1-11. doi: 10.1080/ 15435075.2021.1961263

[58] Yadav, A. S., Shrivastava, V., Dwivedi, M. K., and Shukla, O. P. "3-dimensional CFD simulation and correlation development for circular tube equipped with twisted tape," Materials Today: Proceedings, 2021. doi: 10.1016/j. matpr.2021 .02.549

[59] Yadav, A. S., Shrivastava, V., Sharma, A., and Dwivedi, M. K. "Numerical simulation and CFD-based correlations for artificially roughened solar air heater," Materials Today: Proceedings, 2021. doi: 10.1016/j.matpr.2021.02.759

[60] Sharma, N., Dev Gupta, R., Sharma, R. C., Dayal, S., and Yadav, A. S. "Graphene: An overview of its characteristics and applications," Materials Today: Proceedings, 2021. doi: 10.1016/j. mat pr.2021.03.086

[61] Yadav, A. S., Shrivastava, V., Sharma, A., Sharma, S. K., Dwivedi, M. K., and Shukla, O. P. "CFD simulation on thermo-hydraulic characteristics of a circular tube having twisted tape inserts," Materials Today: Proceedings, 2021. doi: 10.1016/j.matpr.2021.03.396

[62] Yadav, A. S., Shrivastava, V., Chouksey, V. K., Sharma, A., Sharma, S. K., and Dwivedi, M. K. "Enhanced solar thermal air heater: A numerical investigation," Materials Today: Proceedings, 2021. doi: 10.1016/j.matpr.2021.03.385

[63] Kumar, P., Darsigunta, A., Chandra Mouli, B., Sharma, V. K., Sharma, N., and Yadav, A. S. "Analysis of intake swirl in a compression ignition engine at different intake valve lifts," Materials Today: Proceedings, 2021. doi: 10.1016/j.matpr. 2021.03.663

[64] Modi, V. A., Kumar, P., Malik, R., Yadav, A. S., and Pandey, A. "Analysis of optimized turning parameters of Hastelloy C-276 using PVD coated carbide inserts in CNC lathe under dry condition," Materials Today: Proceedings, 2021. doi: 10.1016/j.matpr.2021.05.033

[65] S. Yadav, O. P. Shukla, and A. Sharma, "CFD analysis of heat transfer performance of ribbed solar air heater," Materials Today: Proceedings, vol. In press, 2022.

[66] S. Yadav, O. P. Shukla, and R. S. Bhadoria, "Recent advances in modeling and simulation techniques used in analysis of solar air heater having ribs," Materials Today: Proceedings, vol. In press, 2022, doi: 10.1016/j.matpr.2021.12.242.

[67] S. Yadav and A. Gattani, "Revisiting the influence of artificial roughness shapes on heat transfer enhancement," Materials Today: Proceedings, vol. In press, 2022, doi: 10.1016/j.matpr.20 21.12.254.

[68] S. Yadav, M. K. Dwivedi, A. Sharma, and V. K. Chouksey, "CFD based heat transfer correlation for ribbed solar air heater," Materials Today: Proceedings, vol. In press, 2022.

[69] R. Kumar, S. K. Verma, S. K. Mishra, A. Sharma, A. S. Yadav, and N. Sharma, "Performance Enhancement of SAH using Graphene/Cerium Oxide and Graphene -Black Paint Coating on Roughned Absorber Plate: A Comparative Study," Int. J. Vehicle Structures & Systems, vol. 14, no. 1, 2022, doi: 10.4273/ijvss.14.1.23.