INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X
Impact factor: 4.295
(Volume 4, Issue 3)
Available online at: www.ijariit.com

A review on CFD based flow and heat transfer analysis of various ribs in solar air heater duct by continuous ribs on absorber plate

Sitaram Sah
sah.sitaram319@gmail.com
Patel College of Science and Technology, Bhopal, Madhya Pradesh

Amit Kumar
amitavit689@gmail.com
Patel College of Science and Technology, Bhopal, Madhya Pradesh

ABSTRACT

In order to improve performances, in terms of higher thermodynamic efficiency and power output for heat exchanging devices like solar air heaters, heat exchangers and machines like gas turbines, the study of heat transfer mechanism and fluid flow play a vital role. Thermal assessment of heat exchanging devices, computational fluid dynamics (CFD) approach has advantages over the cost affecting practical experiments. To understand the complete phenomenon of heat transfer, heat flow, as well as fluid flow, must be considered in the numerical analysis. Hence, accurate numerical method turbulence model and equation solving approach are required to predict heat transfer behavior.

Keywords: *Solar air heater, turbulent flow, Nusselt number, ribs, Reynolds number, Pitch*

1. INTRODUCTION

The ribs applying to the wall of the flow passage of fluid break the laminar sub-layer and create local wall turbulence due to flow separation and reattachment of flow between consecutive ribs, which decreases the thermal resistance and highly augment the heat transfer. The enhancement of heat transfer by flow separation and reattachment, caused by ribs, is significantly higher than compared to the fin-effect, linked to the increased heat transfer area [6]. However, the use of artificial roughness results in higher friction losses leading to excessive power requirement for the air flow through the duct. Therefore, it is desirable that the turbulence must be created only in the region very close to the heat transfer surface, i.e. in the viscous sub-layer where the heat transfer takes place and the core flow should not be disturbed so as to avoid excessive friction losses. This can be achieved by keeping small the height of roughness elements in comparison with the duct dimensions. [7]

It has been found through experimental investigations that the enhancement in Nusselt number for air flow in a roughened channel with angled ribs is on the average higher than that roughened with transverse (90 deg) ribs of the same geometry. Secondary flows generated by the angled ribs are considered to be reason for these higher heat transfer coefficients [8]. [9] Studied both the thermal and fluid-dynamic behaviors of turbulent flows with two-dimensional ribs and three-dimensional blocks in the context of surface roughness effects. They solved the Reynolds-averaged Navier-Stokes equations, coupled with the k-ω turbulence model with near wall treatment, by a finite-volume method. They analyzed different three-dimensional configurations with ribs, both in line and staggered. For the in-line cases, they found that Nusselt number was about 20% lower than the results obtained from the 2-D simulations.

Augmentation of convective heat transfer of a rectangular duct with the help of baffles/ribs has been a common practice in the past few years. This concept is widely applied in enhancing the thermo-hydrodynamic efficiency of various industrial applications such as thermal power plants, heat exchangers, air conditioning components, refrigerators, chemical processing plants, automobile radiators and solar air heaters. The solar air heater is a device used to augment the temperature of air with the help of heat extracted from solar energy. These are cheap, have a simple design, require less maintenance and are eco-friendly. As a result, they have major applications in the seasoning of timber, drying of agricultural products, space heating, curing of clay/concrete building components and curing of industrial products.

The shape of a solar air heater of conventional application is that of rectangular duct encapsulating an absorber plate at the top, a rear plate, insulated wall under the rear plate, a glass cover over the sun-radiation exposed surface, and a passage between the bottom plate and absorber for air to flow in. The detailed constructional details of a solar air heater are shown in fig. 1.

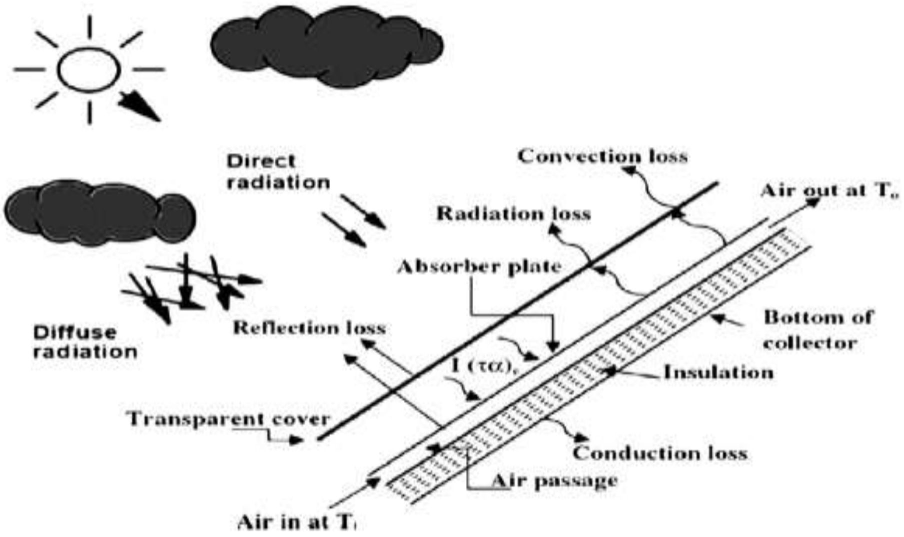

Fig. 1: Solar air heater constructional details

2. REVIEW OF PAST RESEARCH

Hari Raghavan. J and Rangu. P (2017) understood the thermal benefits of using a porous pin fin heatsink when compared to a conventional pin fin heatsink. The cross-section of the heatsink used in the analysis is square shaped. The Heatsink is to be analyzed numerically for natural convection assuming steady state condition. Finite volume method is considered for doing the thermal analysis using a Computational Fluid Dynamics tool called FloTHERM.

Arun Kumar T P and Vijayaraghu B (2017) studied a numerical simulation of heat transfer in the rectangular fin type of heat sink under various operating conditions. The methodology is validated by analyzing the flow through the rectangular channel with heat transfer. It is observed that the introduction ribs enhance the heat transfer capabilities. The effect of flow velocities on the Nusselt number at various rib length (0.2mm, 0.3mm, 0.4mm, and 0.5mm) as well as diagonal pitches(1.25mm, 1.5mm, 1.75mm, and 2mm) have been quantitatively analyzed. The analysis showed that rib length 0.5mm and diagonal pitch 1.25mm gives the highest enhancement in the heat transfer. The provisional of ribs also result in increases in the pressure drop through the passage. Hence it is necessary to optimize both heat transfer coefficient and pressure drop simultaneously.

Amit Garg et al (2017) researched on heat transfer and fluid flow characteristics in a channel in the presence of diamond-shaped baffles in the laminar flow regime. The computations are based on the finite volume method, the Navier Stokes equations along with the energy equation have been solved by using SIMPLE Technique. The unstructured triangular mesh is used for the computational domain. The fluid flow and heat transfer characteristics are presented for Reynolds numbers based on the hydraulic diameter of the channel ranging from 100 to 600. Effects of different baffle tip angles on heat transfer and pressure loss in the channel are studied and the results of the diamond baffle are also compared with those of the flat baffle. The velocity profiles were obtained for all the geometry considered and selected for different sections, namely, downstream and between the two baffles and the friction coefficients were obtained for different sections and for different Reynolds numbers. It is observed that apart from the rise of Reynolds number, the reduction of the baffle angle leads to an increase in the Nusselt number and friction factor.

K. Maliwan (2017) studied the flow and heat transfer characteristics in a rotating two-pass square channel with ribbed walls. In this study, the channel length-to-hydraulic diameter ratio of the rotating two-pass square channel (L/Dh), the rib height-to-hydraulic diameter ratio (e/Dh), rib angle of attack and the rib pitch-to-height (p/e) ratio are fixed at 11.33, 0.13, 60o and 10, respectively. The test fluid is air having the flow rate in terms of constant Reynolds number (Re) of 10,000. The rotation numbers (Ro) are varied from 0.1 to 0.4. The details of the local heat transfer distribution and the flow field of the rotating two-pass square channel are numerically studied by using commercial software ANSYS Fluent (ver.15.0).

Mayank Bhola (2017) presented the 2-D numerical simulation for heat transfer and friction factor characteristics of the ribbed turbulent channel, with bottom plate applied to constant flux thermal boundary condition, and the upper plate is insulated. The fluid domain is designed using ANSYS Workbench software and simulated using ANSYS Fluent software, based upon finite element method. Parameters like a Nusselt number, friction factor, and heat transfer coefficient have been obtained for the design, and contours of temperature and velocity are presented. The groove length/rib length (B) to channel height (H) ratios are taken as 0.5, 0.75, 1.0 for the geometry. The main objective of the work performed was to find the best B/H ratio under similar boundary conditions.

Navanath G. Ghodake et al (2016) focuses on the use of various Ribs to enhance heat transfer. The ribbed wall destabilized the flow. The separations and reattachments over the ribbed wall increase fluid mixing, create flow unsteadiness, interrupt the development of the thermal boundary layer and enhance the heat transfer. In this paper, CFD analysis of different shape ribs such as V-ribs, (simple and broken v-ribs), triangular and rectangular ribs placed in rectangular duct has been performed.

S. V. Kadbhane and D. D. Palande (2016) reviewed the parameter which affects convective heat transfer most for constrained dimensions of the heat sink. Also, this paper focuses on the assistance of mixed convection to natural convection to enhance heat transfer.

C. H. Liang et al. (2016) found the optimal designing parameters of a plate-fin heat sink under natural convection using the Particle Swarm Optimization (PSO) Algorithm. Minimization of entropy generation rate under given space restrictions is considered as objective functions. All relevant design parameters for plate-fin heat sinks are the fin height, fin number, fin thickness. The constraints of the variables are set according to the suggestion structure design. And these three variables influences on entropy generation are presented. In the present study, In order to prevent the size of the heat sink is too large, we use the penalty function method in this study. Then the code for the PSO is written in MATLAB. On this basis, the optimal size of the heat sink was obtained through the particle swarm algorithm for numerical simulation of this model: fin height is 44.8mm, a number of fins are 25, fin thickness is 0.6mm and base temperature is 342-6241 K.

A. Nagaraju et al (2016) analyzed 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 the STAR-CCM+ software. There is an appreciable increase in temperature at the outlet in solar air heaters using artificially roughened absorber plates. The V-shaped rib roughness gives a high rate of heat transfer.

Swajot Singh and Raji N. Mishra (2016) presented for a detailed investigation of the design Of solar air heater having different size of rib on the absorber plate by using the application of computational fluid dynamics (CFD).In this Solar air heater, an absorber plate is made of 'Aluminium' and Roughened with difference size ribs due which creates the turbulence in the flow of fluid (air) and increase the heat transfer from the absorber plate to the fluid. The commercial finite volume based CFD code ANSYS FLUENT 14.5 is used to simulate turbulent airflow through artificially roughened solar air heater. The results predicted by the present CFD investigation are much closer to experimental results. It can, therefore, be concluded that the present numerical results have demonstrated the validity of the proposed system. Thus it is possible to establish a validated model for the prediction of heat transfer and fluid flow phenomena in artificially roughened solar air heater. In order to predict the performance of the system, Nusselt number and friction factor correlations have been developed by using the data generated under CFD based investigation.

Santosh Kansal and Piyush Laad (2015) presented a best possible Heat Sink for efficient cooling of electronic devices. This paper deals with the comparative study of a heat sink having fins of various profiles namely rectangle, Trapezoidal, rectangle Interrupted, Square, circular inline and staggered, as heat sinks are the commonly used devices for enhancing heat transfer in electronic components. In this work, a new concept for cooling the electronic components using the Aluminium alloy heat sink is proposed. The selection of an optimal heat sink depends on a number of geometric parameters such as fin length, fin thickness, number of fins, base plate thickness, space between fins, fin shape or profile and material etc. Therefore for an optimal heat sink design, initial studies on the fluid flow and heat transfer characteristics of standard continuous heat sinks of different designs have been carried through CFD simulations. CFD analysis is conducted in order to establish the effect of geometrical fin parameters for natural convection heat transfer on different fin arrays.

Ravi Teja (2015) simulated the use of the laminar and k-ε model for predicting flow and heat transfer with measured flow field data in a stationary duct which sheds light on the detailed physics encountered in the fully developed flow region and the sharp 180° bend region. Among the major flow features predicted with accuracy is flow transition at the entrance of the duct, the distribution of mean and turbulent quantities in the developing, fully developed, and sharp 180° bend, the development of secondary flows in the duct cross-section and the sharp 180° bend, and heat transfer augmentation. Flow intensities in the sharp 180° bend are found to reach high values and local heat transfer comparisons show that the heat transfer augmentation shifts towards the wall and along the duct. Therefore, understanding of the unsteady heat transfer in sharp 180° bends is important.
Priyank Lohiya and Shree Krishna Choudhary (2015) studied forced convection of fully developed turbulent flow in a rectangular duct having ribs on the underside of the top wall. CFD solutions are obtained using commercial software ANSYS FLUENT v12.1. The working fluid in all cases is air.

Vivek Rao et al (2015) investigated the thermo-hydraulic performance of four ribs–roughened rectangular duct. 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 conditions of main rib configuration. For all cases, hydraulic diameter and angle of attack were kept constant to 33 mm and 600. Only relative roughness pitch and Reynolds number were varied from 8-12 and 8000-15000.

Ashok Singh Yadav et al (2015) performed a numerical investigation is conducted to analyze the 3-dimensional incompressible Navier–Stokes flow through the artificially roughened solar air heater for the Reynolds number ranges from 2000 to 20,000, the effect of geometrical parameters of the V-shaped perforated blocks on heat transfer and flow characteristics of rectangular duct, has been investigated. The governing equations are solved with a finite-volume-based numerical method. The commercial finite-volume based CFD code ANSYS FLUENT 14 is used to simulate turbulent airflow through artificially roughened solar air heater. The RNG k–ε turbulence model is used to solve the transport equations for turbulent flow energy and dissipation rate.

Arkan Al-take and Hasasn Ali Jurmut (2014) presented an experimental and numerical investigation of heat transfer characteristics of horizontal circular pipe500mm long using internal square ribs of 80mm width, 80mm height, n=7 in upper surface and 80mm width ,80mm height, n=6 in lower surface of pipe, p =100mm, with air as the working fluid. Reynolds number 31170 was taken. The steel pipe (ASM4120) was subjected to different constant surface temperatures. The experimental data obtained were compared with plain (without ribs) case. Based on the same coolant flow, the pipe with internal square ribs was found to possess the highest performance factors for turbulent flow. The results show a good agreement between theoretical and experimental by factor 4.4%. The heat transfer rates obtained from pipe upper surface and the lower surface is 213% over the smooth channel for a given Reynolds number.

Mostafa M. Awad (2013) compared the performance of convergent-divergent fins compared with those of other types of fins. To carry out this comparison, natural convection heat transfer in air from different type surface is investigated experimentally with consideration of the effects of radiant heat transfer. Plate-fins (Parallelogram fins), cylindrical solid/hollow pin fins and convergent-divergent fins are tested. From now, the plate fins will be termed as straight fins to distinguish from the plain plate which is the array base plate. The solid /hollow pin fins and convergent-divergent fins are arranged in staggered and inline arrangements. The experiments have been performed for different values of heat flux.

Raj Kumar (2013) presented the results of a CFD based analysis on heat transfer and friction characteristics to the air flow in the rectangular duct. The upper side of the heated plate is made rough with circular ribs of broken multi v-rib. The results of the broken multi v-rib the significant increase in heat transfer rate and friction loss over the smooth surface. The relative gap width (g/e) 1.0 rib provided the highest broken multi v-riband CFD model results were compared with broken multi v-rib, Realizable k-epsilon model results have been found to have a good agreement.

Anil Kumar et al (2013) presented the work performance of a solar air heater duct provided with artificial roughness in the form of thin circular wire in discrete angled rib geometries has been analyzed using Computational Fluid Dynamics (CFD). The effect of this geometry on heat transfer and friction factor and performance enhancement was investigated covering the range of roughness parameters, P/e=8, e/D=0.043, d/W=0.25, g/e=1.0, α=60° and working parameters (Reynolds number, Re from 2,000 to 20,000). Different turbulent models have been used for the analysis heat transfer and friction factor and their results are compared with the Dittus-Boelter Empirical relationship for a smooth surface. Renormalization k-epsilon model-based results have been found in good agreement and accordingly this model is used to predict heat transfer and friction factor in the duct.

Suman Saurav and V.N. Bartaria (2013) improve the thermal performance artificial roughness is provided on the underside of absorber plate due to which turbulence is created in the heat transfer zone and ultimately the performance of solar air heater improves considerably. 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 the Nusselt number was investigated to study the heat transfer, friction factor and flow characteristics in a solar air heater having triangular rib roughness on the absorber plate.

3. SUMMARY OF LITERATURE SURVEY

In this work, we reviewed the investigations carried out by numerous investigators to enhance the heat transfer and using obstacles of different shapes, sizes, and orientations to produce artificial roughness in flow ducts. Improving heat exchangers with the viewpoint of maximum convective heat transfer enhancement and reduced size, weight and cost is still taken into consideration. Using ribbed square channel is one of the most effective ways to increase the heat transfer in many engineering applications. Apart from appearance and geometric parameters of a roughened channel, working fluid plays a significant role on the heat transfer enhancement. Nanofluids with the definition of ultrafine particles suspended in a conventional base fluid with high thermal conductivity can considerably assist in this issue. Metallic, non-metallic and polymeric particles suspended in a base fluid with high thermo-physical properties can improve the thermos physical properties of the mixture and subsequently bring higher heat transfer enhancement compared to conventional working fluids. The influence of triangular ribs having an angle of attack of 300-600 was examined on flow factors and they found that the triangular rib with flow attack angle value of 300 had maximum heat transfer and minimum pressure reduction besides the microchannel.

The review reveals that convective heat transfer from rectangular plate fin array depends upon various factors such as geometrical parameters, operating parameters, fin material, fin array orientation, number of fins etc. But geometrical parameters affect most. As there are limitations to increase the size of fin array in order to increase surface area due to limitations on proposed length and width of fin array, optimal fin spacing was found a most important factor to enhance convective heat transfer. Instead of only going for natural convection or forced convection, mixed convection was also proposed for better enhancement in convective heat transfer as mixed convection enhanced natural convection.

4. REFERENCES

[1] Hari Raghavan. J, Rangu. P (2017), "A Study and Analysis on the Thermal Performance of a Pin Fin Heatsink for Natural Convection using CFD", International Journal of Engineering Research & Technology, 6 (5), PP: 883-888.

[2] Arun Kumar T P, Vijayaraghu B (2017), "Three Dimensional Numerical Analysis of Heat Transfer by Forced Convection through Heat Sink with Rectangular Ribs", International Journal of Modern Trends in Engineering and Research, 5 (1), PP: 238-243.

[3] Amit Garg, Sunil Dhingra, Gurjeet Singh (2017), "CFD Analysis of Laminar Heat Transfer in a Channel Provided with Baffles: Comparative Study between Two Models of Baffles: Diamond-Shaped Baffles of Different Angle and Rectangle", International Journal of Enhanced Research in Science Technology & Engineering, 3 (7), PP: 267-276.

[4] K. Maliwan (2017), "Numerical simulations on flow and heat transfer in ribbed two-pass square channels under rotational effects", Materials Science and Engineering, 5 (1), PP: 114-126.

[5] Mayank Bhola (2017), "Analysis of Heat Transfer of Ribbed Turbulent Channel using ANSYS", International Journal on Emerging Technologies, 8(1), PP: 236-242.
[6] Navanath .G.Ghodake, MRC. Rao, Dr. R. R. Arakerimath (2016), "Flow and Heat Transfer Analysis of Various Ribs for Forced Convection Heat Transfer", Journal of Emerging Technologies and Innovative Research", 3(7), PP: 42-47.
[7] S. V. Kadbhane, D. D. Palande (2016), "Review of Convective Heat Transfer from Plate Fins under Natural and Mixed Convection at Different Inclination Angle", International Research Journal of Engineering and Technology, 3 (2), PP: 467-473.
[8] C. H. Liang, S. Zeng, Z. X. Li (2016), "Optimal Design of Plate-Fin Heat Sink under Natural Convection Using a Particle Swarm Optimization Algorithm", International Journal of Heat and Technology", 34 (2), PP: 275-280.
[9] Nagaraju, B. Chandramohan Reddy (2016), "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, 5 (11), PP: 227-230.
[10] Swajot Singh, Raji N. Mishra (2016), "Evaluation of Convective Heat Transfer Coefficient at Different Altitudes in Atmospheric Regime Using Forced Convection", International Journal of Engineering Sciences & Research Technology, 5 (7), PP: 790-794.
[11] Santosh Kansal, Piyush Laad (2015), "Performance & Thermal Analysis of Heat Sink with Fins of Different Configuration using CFD", International Journal of Scientific & Engineering Research, 6 (6), PP: 1487-1495.
[12] Ravi Teja (2015), "Optimization of Heat Transfer through Rectangular Duct", International Research Journal of Engineering and Technology", 2 (4), PP: 1906-1910.
[13] Priyank Lohiya, Shree Krishna Choudhary (2015), "Numerical Study on Heat Transfer of Turbulent Duct Flow through Ribbed Duct", International Journal of Engineering Sciences & Research Technology, 4 (7), PP: 178-182.
[14] Vivek Rao, Dr. Ajay Gupta, Amit Kumar (2015), "CFD Based Heat Transfer Analysis of Solar Air Heater Duct Provided with Artificial Roughness", International Journal of Scientific & Engineering Research, 6 (5), PP: 28-35.
[15] Ashok Singh Yadav, Tarun Singh Samant, Lokesh Varshney (2015), "A CFD Based Analysis of Solar Air Heater Having V-Shaped Perforated Blocks on Absorber Plate", International Research Journal of Engineering and Technology", 2(2), PP: 822-829.
[16] Skullong S., Thianpong C. and Promvonge, P., 2015, Effects of rib size and arrangement on forced convective heat transfer in a solar air heater channel, Heat and Mass Transfer, pp. 1-11.
[17] Arkan Al-taie, Hasasn Ali Jurmut (2014), "Experimental and Numerical Investigation of Convective Heat Transfer in a Circular Tube with Internal square ribs", Journal of Babylon University/Engineering Sciences, 4 (22), PP: 910-918.
[18] Mostafa M. Awad (2013), "Assessment of Convergent-Divergent Fins Performance in Natural Convection", Journal of American Science, 9 (2), PP: 116-124.
[19] Raj Kumar (2013), "CFD based analysis of Heat Transfer and Friction Characteristics of Broken Multiple Rib Roughened Solar Air Heater Duct", International Journal of Mechanical and Production Engineering Research and Development, 3 (5), PP: 153-160.
[20] Anil Kumara, Muneesh Sethia, Khushmeet Kumara (2013), "Computational Fluid Dynamics Based Analysis of Angled Rib Roughened Solar Air Heater Duct", International Journal of Thermal Technologies, 3 (7), PP: 43-47.
[21] Suman Saurav, V.N. Bartaria (2013), "CFD Analysis of Heat Transfer through Artificially Roughened Solar Duct", International Journal of Engineering Trends and Technology, 4 (9), PP: 3937-3944.

