Available online: www.ijsrmes.com

International Journal for Scientific Research in Modern Engineering and Science

International Journal for Scientific Research in Modern Engineering and Science, 1(6): 36-42 (2020)

NUMERICAL SIMULATION OF SOLAR AIR HEATER WITH TRANSVERSE RIBS ON ABSORBER PLATE USING CFD

Dr. Rahul Mishra[1], Yogesh Kumar Sirmour[2]

[1]Assistant Professor, Department of Mechanical Engineering, Shri Shankaracharya Technical Campus, Bhilai (C.G.)
[2]Student, Department of Mechanical Engineering, Shri Shankaracharya Technical Campus, Bhilai (C.G).

Abstract

In today's scenario solar air heater plays a very important role in many heating and drying applications as it uses solar energy for heating purpose and dehumidification of air. It is an effective technique to use artificial roughness in the form of repeated ribs on a surface to augment the heat transfer rate. A numerical simulation has been carried out to investigate the heat transfer and fluid flow characteristics of fully developed turbulent flow in a rectangular duct having transverse quarter circular ribs as roughness element on absorber plate.

Commercially available ANSYS 14 @ was the CFD software employed to solve the concerned general differential equations numerically. This software numerically simulates using FINITE VOLUME METHOD. The navier stokes and energy equations have been solved in conjunction with low Reynolds number RNG k- turbulence model. Design of SAE consisted of three sections, test section of length L2 = 280 mm, entrance section of length L1= 245 mm and exit length of length L3 = 115 mm [13], width of duct is 100mm and height is 20 mm. Material of absorber plate is aluminum and fluid flowing through the duct is air[1].

Solar air heater of conventional applications is that of rectangular duct having 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 through passage is from 3800 to 18000[1].

Keywords: *Solar Air Heater; Finite Volume Method; Fin Cooling; Solar Panel.*

** Corresponding author*

1. Introduction

It is an effective technique to use artificial roughness in the form of repeated ribs on a surface to augment the heat transfer rate. Augmentation of convective heat transfer of a rectangular duct with the help of ribs is now common between engineers and scientists. This concept is widely applied in enhancing the thermo-hydraulic efficiency of various industrial applications such as solar air heaters, air conditioning components, refrigerators, chemical processing plants, automobile radiators and. 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 simple design, require less maintenance and are eco-friendly. As a result, they have major applications in 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 having 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.

Solar air heaters have higher thermal efficiency when the Reynolds number of air flow through the passage is from 3800 to 18000. In this range, the fluid flow is usually turbulent. Hence, all the research work pertaining to the design of an effective solar air heater involves turbulent flow. Conventional solar air heaters with all the internal walls being smooth usually have low efficiency. The solar air heater's internal surface can be artificially roughened by mounting certain ribs/obstacles of different shapes such as circular wires, rectangular ribs, triangular ribs, etc. periodically on the lower side of collector plate. This results in a considerable augmentation in the heat transfer rate, but at the same time leads to increase in friction factor thereby enhancing the pumping power requirements.

1.1 Introduction to Turbulent Flow

A laminar flow is characterized by regular and orderly motion of fluid elements. In this type of flow, viscous effects try to dampen out the disturbances of the fluid flow. Furthermore, in laminar flow, perturbations die out with fluid flow, but the reverse happens in case of turbulent flow, i.e. perturbations simply get amplified as the fluid flows. Turbulent flow occurs when the inertia effect dominates the viscous forces.

Turbulent flow regime can be generally divided into four different regions. The layer closest to wall is very thin and is popularly known as laminar sub-layer. Here, viscous forces are dominant and the fluid flow behavior is almost the same as that of laminar flow. The next layer is called buffer layer, in which there is still an effect of viscous forces dominance but the flow somewhat becomes a bit turbulent. The buffer layer is followed by inertial sub-layer, in which there is still some effect of turbulence, but not stronger. The turbulent (outer) layer is the core layer in which inertial forces become dominant over the viscous forces.

1.2 Introduction to Turbulence Modeling

Additionally, turbulent flows possess certain distinguishing features such as randomness of transport (fluctuation of properties) variable with respect to time and space. The momentum exchange between particles of a fluid when it exhibits turbulent flow is high accompanied by strong mixing. Furthermore, turbulent flows contain wide range of length and time scales. Moreover, turbulent flow parameters are highly sensitive to initial conditions. For these reasons, it is very difficult to capture the physics of turbulent flow accurately in a single continuum simulation. The complexity exaggerates as the Reynolds number increases further.

Since there is high level of fluctuation of turbulent flow transport quantities with respect to time and space, conventional Navier-Stokes equation cannot be applied to capture their behavior. The addressed complexities can be overcome by a method called turbulence modeling. The set of mean flow equations are enclosed by this computational process.

Geometry of solar air heater duct with roughness on absorber plate has been shown in figure 1.5 . It consisted of three sections, test section of length L2 = 280 mm, entrance section of length L1= 245 mm and exit length of length L3 = 115 mm [1].Aspect ratio is 5, width of duct is 100 mm and height is 20 mm. Material of absorber plate is aluminum and fluid flowing through the duct is air[13]. In fig 1.6 cross sectional view of "L" shaped and "quarter circular" transverse ribs have been shown with rib height (e) and pitch (P).

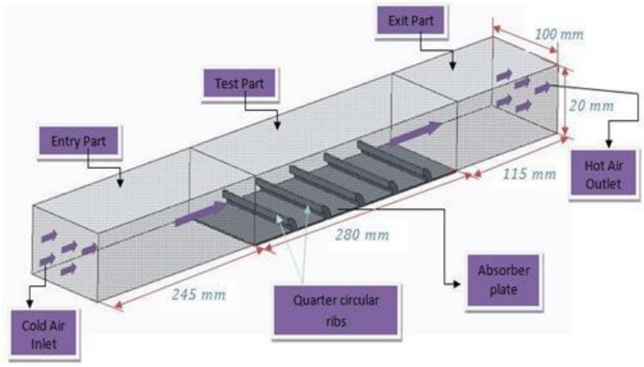

Fig 1:-D geometry of solar air heater duct with roughness on absorber plate

2. Literature Review

Gupta D, Solanki S C, Saini J S (1997) have analyzed roughened solar air heater with non-transverse ribs with 60 degree angle of attack with parameters relative roughness height (e/D_h) 0.023-0.05, Reynolds number and isolation (400-1300 W/m^2) and observed roughened solar air heaters are thermo-hydraulically more efficient if they operate in a given range of Reynolds numbers (say 3500 to 19,000 for a relative roughness height of 0.023 and isolation of 1000 W/m^3.

Chaube A, (2006) has carried out a numerical analysis of an artificially roughened solar air heater to study the roughness effect on friction factor and heat transfer. Commercially available FLUENT6.0 software was used to solve the continuity and momentum equations computationally. They varied the Reynolds number from 3000-20000 and the turbulent model SST-k-omega was selected on the basis of predictions of various different turbulence models with available experimental results in literature. The simulations were performed on nine different types of rib shapes. The authors observed that the numerical results corresponding to 2-D and 3-D model shad very less difference and hence the domain chosen was two- dimensional. They further stated that at regions exactly between two consecutive ribs, there was a large augmentation of heat transfer coefficient as a result of reattachment of flow with the absorber plate at these regions. At these locations, the turbulence intensity was the highest. The authors concluded that the highest heat transfer coefficient value was associated with chamfered ribs although rectangular rib of size 3*5 mm gave the best thermal performance.

Anil Singh Yadav, Bhagoria J L (2013) have done the performance analysis of small diameter circular rib on absorber plate of solar air heater and found that the maximum enhancement of average Nusselt number is found to be 2.31 times that of smooth duct for relative roughness pitch of 7.14 and for relative roughness height of 0.042. Results were validated by comparing with available experimental results and the discrepancy between available experimental data and computational results were less than 7.5%.

Anil Singh Yadav, Bhagoria J L (2014) have performed the numerical investigation on triangular rib solar air heater on CFD found that the maximum enhancement in the Nusselt number has been found to be 3.073 times over the smooth duct corresponds to relative roughness height (e/D) of 0.042 and relative roughness pitch (P/e) of 7.14 at Reynolds number (Re) of 15,000 in the range of parameters investigated.

Anil Singh Yadav, Bhagoria J L (2014) have done the performance analysis of transverse square ribs on absorber plate of solar air heater using CFD in ANSYS Fluent and found that the maximum enhancement of average Nusselt number is found to be 2.86 times that of smooth duct for relative roughness pitch of 7.14, relative roughness height of 0.042 and for Reynolds no. 15000.

Sukhmeet Singh, Bikramjit Singh, Hans V S, Gill R S, (2015) have performed a 3-dimensional CFD investigation to study the heat transfer and friction characteristics of solar air heater duct roughened with periodic transverse rib. The selected rib roughness has a new concept, it has non-uniform cross-section in the form of saw-tooth. And have compared with transverse ribs with uniform cross-section of circular, square and trapezoidal have also been investigated. The non-uniform cross- section saw tooth rib was found to result in higher Nusselt number than uniform cross-section ribs for Reynolds number above 7000 due to reduced low heat transfer area downstream of the rib caused by disruption inre-circulations. The maximum enhancement in Nusselt number for duct roughened with saw-tooth riband trapezoidal rib was 1.78 and 1.50 respectively.

Hyung Do et al. presented a resistance correlation for design tool of a natural convective heat with plate-fins for concentrating photovoltaic (CPV) module cooling is suggested. So, extensive experimental investigations are performed for different heat sink geometries and input power, as well as inclination angle. The thermo physical properties of water confirm that it as a good cooling medium. It transforms radiation to electrical energy and absorbs the heat from the panel. According to this, the panel is working in lower temperatures.

3. Objectives of The Work

By reviewing the past work, numerical study of solar air heater is an excellent method to understand in detail about how flow behaves under the presence of obstacles in solar air heaters. Parameters selected in present analysis are based on literature reviewed earlier.

- To investigate the variation in Nusselt number (Nu), friction factor (fr) and thermo-hydraulic performance factor (THPF) of solar air heater for transverse ribs with "quarter circular" shaped cross section as roughness geometry for various-
- Reynolds number (Re) = 3800-18000
- Relative roughness pitch (P/e) = 3.57-14.29 at constant relative roughness height (e/Dh) = 0.042

- Relative roughness height (e/Dh) = 0.03-0.054 at constant relative roughness pitch (P/e) = 7.14
- To compare the variation in Nusselt number (Nur), friction factor (fr) and thermo-hydraulic performance factor (THPF) of solar air heater at constant relative roughness height (e/Dh) = 0.042 for two different shape of roughness geometries-
  - ❖ Transverse ribs with reverse “L” shaped cross section
  - ❖ Transverse ribs with “quarter circular” shaped cross section for various-
  - ❖ Relative roughness pitch (P/e) = 7.14-14.29.
  - ❖ Reynolds number (Re) = 3800-18000.

4. Methodology

It can be advantageous to use CFD over traditional experimental-based analysis, 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 equiments are very inexpensive with CFD when compared to experiments.

There are a number of solution methods used in CFD codes. The fundamental basis of any CFD analysis is the Navier-Stokes equations, which govern any single-phase fluid flow. These equations are a set of coupled differential equations and these can be solved for a given flow problem by using calculus. But in practice, these equations are very difficult to solve analytically. However, these equations can be simplified by making approximations and simplifications. Recently, high speed computers have been used to solve these equations using a variety of techniques like finite difference, finite volume, finite element and spectral methods

4.1 Setup and flow specification

The generated mesh was then exported to FLUENT where the different flow and physical properties were specified. The energy option was switched on. Steady, pressure based solver has been selected with absolute velocity formulation. The flow equations are solved using RNG k–e turbulence model as it shows very good agreement with the empirical relations as shown in fig 4.4, analyzed by Vipin B Gawande et al. [13]. The working fluid is air and the absorber plate is made up of aluminum due to its higher absorptivity.

5. Result and Discussion

5.1 Grid Independence Test Results

Selection of the best grid size is very important to achieve accurate results. Hence grid independence tests were performed to predict appropriate grid dimensions. Grid was made successively finer to decrease the variation in Nusselt number for Reynolds number (Re) = 3800. As shown below in table 5.1.

S. No.-	Number of elements	Nusselt number	Absolute % difference
1	1,60,232	36.23	-
2	1,86,432	37.652	3.93
3	2,07,687	38.965	1.38
4	2,24,367	38.344	0.826
5	2,36,132	38.126	0.568

By grid independence test it can be seen that the increasing the number of elements beyond 224367 was not necessary as it has been found negligible percentage difference (< 1%) in Nusselt number and also to minimize the computational time required for the analysis.

5.2 Validation of CFD model

The CFD results are validated in the form of average Nusselt number and friction factor for smooth duct with the correlations proposed by Dittus-Boelter and Blasius. Comparison of CFD and correlation values of Nusselt number and friction factor is shown in Fig 5.1 and Fig 5.2. The CFD and correlations values are in good agreement to ensure the accuracy of the data being collected from the CFD results.

Nusselt number for smooth duct (Nu_s) is calculated by empirical correlation given by Dittus – Boelter,

$$Nu_s = 0.023\ Re^{0.8}\ Pr^{0.4}$$

Friction factor for smooth duct (f_s) can be calculated by correlation given by Blasius equation,

$$f_s = 0.0791\ Re^{-0.25}$$

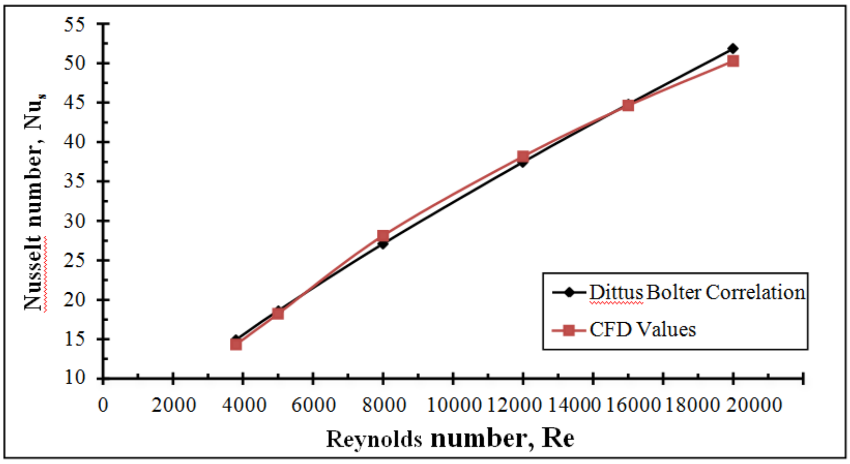

Fig 3: Comparison of Nusselt number for smooth duct between Dittus-Boelter correlation results and CFD results

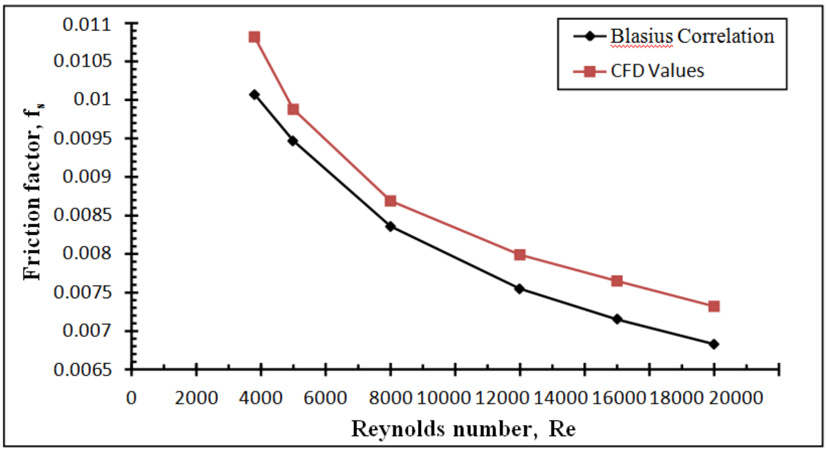

Fig 4: Comparison of friction factor for smooth duct between Blasius correlation and CFD results

References

[1] Gupta D, Solanki S C, Saini J S; "Heated fluid flow in rectangular solar air heater ducts having transverse rib roughness on absorber plates." Solar Energy Vol. 61, No. 1, pp. 33-42 (1997).

[2] Bhagoria J L, Saini J S, Solanki S C; "Heat transfer coefficient and friction factor correlations for rectangular solar air heater duct having transverse wedge shaped rib roughness on the absorber plate." Renewable Energy 25 (2002) 341–369.

[3] Chaube A, Sahoo P K, Solanki S C; "Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater." Renewable Energy 31 (2006) 317–331.

[4] Karmare S V, Tikekar A N; "Analysis of fluid flow and heat transfer in a rib grit roughened -surface solar air heater using CFD"; International Journal of Heat and Mass Transfer 50 (2007) 4342–4351.

[5] Yadav A S, Bhagoria J L; "A CFD (computational fluid dynamics) 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 (2013) 1127-1142.

[6] Yadav A S, Bhagoria J L; "A CFD based thermo-hydraulic performance analysis of an artificially roughened solar air heater having equilateral triangular sectioned rib roughness on the absorber plate"; International Journal of Heat and Mass Transfer 70 (2014) 1016–1039.

[7] ANSYS FLUENT 14.5. Documentation. Ansys, Inc; 2014.

[8] Yadav A S, Bhagoria J L; "A numerical investigation of square sectioned transverse rib roughened solar air heater" International Journal of Thermal Sciences 79 (2014) 111-131.

[9] Anil Kumar, Man-Hoe Kim; "Effect of roughness width ratios in discrete multi V- rib with staggered rib roughness on overall thermal performance of solar air channel" Solar Energy 119 (2015) 399–414.