CFD Investigation of Solar Air Heaters provided with an artificial roughness of a BOOT Shape in a rectangular duct.

[1]Kartik Goswami, MTech Student, Tula's Engineering College

[2]Mr.Ankit Jain, Assistan Professor, Tula's Engineering College

Article Info
Page Number: 3377-3385
Publication Issue:
Vol. 71 No. 4 (2022)

Article History
Article Received: 25 March 2022
Revised: 30 April 2022
Accepted: 15 June 2022
Publication: 19 August 2022

Abstract:-
This article is based upon the solar air heaters containing an artificial roughnesses of a boot shape represents the overall and complete study of the heat transfer taking place and the fluid flow processes using computational fluid dynamics (CFD). According to the used roughness in this very article we will be observing the total effect of the heat transfer and friction factor also. Now, this way we will come to the conclusion that how the way our project is quite beneficial from others. The boot shape which has been used in this project is having certain dimension which has been designed using ANSYS. We will first use the smooth-duct for the comparison and then further the variation will be done on the shape of our roughness and the parameters used in theroughness(boot) like angle of attack, pitch distance, height, etc. The Reynolds number will also be varied from 2000 to 18000 and the observation will be done that on which very Reynolds number the best result is being obtained.

INTRODUCTION

Solar air heater are the devices which are one of the very less complicated and compact type of device which when if compared to the solar water heaters. It is one of the very basic and simple type of device with the help of using it we obtain that the solar energy is easily converted into the thermal energy and hence we can also say that it acts as a transducer which is used to convert the one form of energy into the another one and hence the same is being done in this case too. The solar air heaters are generally consisting absorber plate, rear plate, a transparent cover on the external side which is being exposed, insulation provided below the rear plate and the air which has been considered as our working fluid is allowed to flow in the middle section of the absorber plate and rear plate. As earlier it is stated that in this very article the thermal performances are being measured. So as to measure all these parameters the very important values which has to be determined are the Nusselt number and the frictional factor. This very above two values Nusselt number and the frictional factor which has to be determined are being obtained from the various parameters which has been kept for the variations as stated in the abstract column. There are many various kinds of different shape of an artificial roughness which are being used or it can be said that the shapes which had been used earlier are such as U-Shapes, V-Shapes, W-Shapes, rectangular shapes, trapezoidal shapes, circular shapes, Y-Shapes, etc and many more are there. These all various shapes which had been used earlier had shown various heat transfer rates and thermal efficiencies respectively but according to the BOOT shape which has been considered for this article will show the best results from the others mentioned above.

The investigation represents the total and complete detailed study of the heat transfer processes and fluid flowing processes using an artificially roughened surface of a solar air heaters. The various effect shown due to the boot ribs on the heat transfer and fluid flowing processes have been investigated further. As stated earlier the Reynold number, relative pitch of roughness and relative height of roughness as well as angle of attack are chosen as a variable parameters. The mathematical simulation using CFD is done using ANSYS FLUENT 12.1 code.

One of the very interesting matter of fact is that solar air heaters doesn't show the proper heat transfer rate or thermal efficiency so as to neglect this very problem we added or used an artificial roughness by virtue of which the turbulence is created very easily using our BOOT shape roughness which finally results in a greater increase in heat transfer rate and thermal efficiency over the rectangular ducts. In other case the use of an artificial roughnesses results in a higher frictional losses but it has been tried to keep it very low by decreasing the friction factor which is done using an increase in a Reynolds number. Hence finally the condition for an optimum performance is being determined very easily in terms of a Nusselt number. The maximum value of a Nusselt number has been determined for the various ranges of parameters here investigated.

So, now let us proceed further and discuss what are the various mathematical expressions which had been used to determine the various parameters.

MATEMATICAL MODELLING

There are various mathematical relationships which could be used to determine the required conclusion but here in this case we need few very important mathematical relationships which have been used to determine our further investigation.

According to our objective of investigation which is being required to be found out we need to determine the maximum Nusselt Number:-

So,

According to Dittus – Boelter Equation:-

$$\mathbf{Nu = 0.023Re^{0.8}Pr^{0.4/0.3}(A)}$$

Where,

Nu = Nusselt number

Re = Reynolds number

Pr = Prandlt number

Here,

- Prandlt Numbers if raised to power 0.4 means it is concerned for hotness purpose
- Prandlt Numbers if raised to power 0.3 means it is concerned for coldness purpose

GOVERNING EQUATIONS USED IN RECTANGULAR DUCT

The given flow system is having three different kinds of flow condition which aredistinct in nature such as steady-laminar, unsteady-laminar and turbulent flow. And for all these above conditions the governing equations are developed. The various sections explained below mainly targets on the conservation equation which pertains laminar and turbulent flow conditions.

STANDARD K-ξ MODEL

This basically stands for two different kind of equations turbulence model. The k-ε model are having few strengths and weakness which are now known, various further improvements had been carried out for the model which results in its improved performance. The standard wall equation which was derived using the technique was too revolutionary at that period of time. The k-ε model used is one of the most important and very common turbulence model, in the case when a very high adverse pressure gradient occurs then it doesn't performs very well. As stated above that it is a 2-equation model which means that it includes two different another transport equation for representation of turbulent property of the flow. Finally it helps the two equation model to account for the historic effects like diffusion of turbulent energy and convection. "k" used for the purpose of denoting turbulence kinetic energy and "ε" used for denoting its rate of dissipation. The "k" used in the model transport equations aremainly derived from the exact equations, however the model transport equation used for ε is obtained by using the physical reasoning and rate of destruction of turbulence kinetic energy per unit time.

TURBULENT FLOW

In this case we are talking about the turbulent flow just because of our roughness which we had provided on our absorber plate that helps to create turbulence. There are further various flow parameters like velocity, temperature and pressure field of the flow are being determined by the help of solving governing equations:-

CONTINUITY EQUATION

The continuity equation is defined as the mathematical equation for the principle of applied mass conservation to an elemental or we can say that a specific control volume within a fluid provided under a certain motion. Mathematically it is given by:-

$$\frac{\partial \vec{u}}{\partial x} + \frac{\partial \vec{v}}{\partial y} + \frac{\partial \vec{W}}{\partial Z} = 0 \qquad \textbf{(B)}$$

Above expression represents the mass continuity equationswhich isused for the two dimensional kind of steady flows where it do not have any breaks in between.

MOMENTUM EQUATION

Momentum equation is an equation which is basically based on or we can say completely dependent on the laws of conservation of moment or else on the momentum principles of turbulent flow field, it ultimately results to that state were the net forces which are acting on a mass acceleration fluid is equals to the change in the momentum of flow per unit time in the direction.

X- Momentum equations:

$$\left(\vec{\mathbf{u}}\frac{\partial\vec{\mathbf{u}}}{\partial\mathbf{x}}+\vec{\mathbf{v}}\frac{\partial\vec{\mathbf{u}}}{\partial\mathbf{y}}+\vec{\mathbf{w}}\frac{\partial\vec{\mathbf{u}}}{\partial\mathbf{z}}\right)=-\frac{1}{\rho}\frac{\partial\mathbf{p}}{\partial\mathbf{x}}+\mathbf{v}\left(\frac{\partial^2\vec{\mathbf{u}}}{\partial\mathbf{x}^2}+\frac{\partial^2\vec{\mathbf{u}}}{\partial\mathbf{y}^2}+\frac{\partial^2\vec{\mathbf{u}}}{\partial\mathbf{z}^2}\right)\ \textbf{(C)}$$

Y- momentum equation:

$$\left(\vec{\mathbf{u}}\frac{\partial\vec{\mathbf{v}}}{\partial\mathbf{x}}+\vec{\mathbf{v}}\frac{\partial\vec{\mathbf{v}}}{\partial\mathbf{y}}+\vec{\mathbf{w}}\frac{\partial\vec{\mathbf{v}}}{\partial\mathbf{z}}\right)=-\frac{1}{\rho}\frac{\partial\mathbf{p}}{\partial\mathbf{y}}+\mathbf{v}\left(\frac{\partial^2\vec{\mathbf{v}}}{\partial\mathbf{x}^2}+\frac{\partial^2\vec{\mathbf{v}}}{\partial\mathbf{y}^2}+\frac{\partial^2\vec{\mathbf{v}}}{\partial\mathbf{z}^2}\right)\textbf{(D)}$$

Z – momentum equations:

$$\left(\vec{\mathbf{u}}\frac{\partial\vec{\mathbf{v}}}{\partial\mathbf{x}}+\vec{\mathbf{v}}\frac{\partial\vec{\mathbf{v}}}{\partial\mathbf{y}}+\vec{\mathbf{w}}\frac{\partial\vec{\mathbf{v}}}{\partial\mathbf{z}}\right)=-\frac{1}{\rho}\frac{\partial\mathbf{p}}{\partial\mathbf{z}}+\mathbf{v}\left(\frac{\partial^2\vec{\mathbf{v}}}{\partial\mathbf{x}^2}+\frac{\partial^2\vec{\mathbf{v}}}{\partial\mathbf{y}^2}+\frac{\partial^2\vec{\mathbf{v}}}{\partial\mathbf{z}^2}\right)\textbf{(E)}$$

Energy equation

For steady turbulent flow it is assumed that flow always remains steady and incompressible with the constant thermal conductivity without any compression work and also without any heat generations.

Mathematically for no viscous heating it is given by:-

$$\vec{\mathbf{u}}\frac{\partial\mathbf{t}}{\partial\mathbf{x}}+\vec{\mathbf{v}}\frac{\partial\mathbf{t}}{\partial\mathbf{y}}+\vec{\mathbf{w}}\frac{\partial\mathbf{t}}{\partial\mathbf{y}}=\boldsymbol{\alpha}\left(\frac{\partial^2\mathbf{t}}{\partial\mathbf{x}^2}+\frac{\partial^2\mathbf{t}}{\partial\mathbf{y}^2}+\frac{\partial^2\mathbf{t}}{\partial\mathbf{y}^2}\right)\qquad\textbf{(F)}$$

ASSUMPTIONS

While performing the numerical simulation various kinds of assumptions were considered which are as follows:-

(1) Flow was considered to be steady.
(2) In y direction the shear force was considered to be zero.
(3) The flow was considered to be incompressible.
(4) In y direction the variation of pressure was considered to be zero.
(5) Body force which takes place due to the presence of gravity is neglected.
(6) The axial heat conduction present in the fluid was negligible.
(7) At the section of inlet the flow was considered to be completely developed.

The property of air was considered to be constant at atmospheric pressure as well as temperature.

GEOMETRY OF BOOT SHAPE

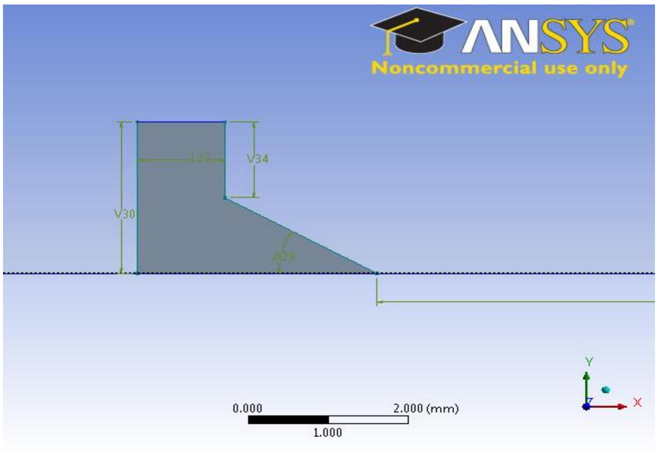

Fig.(1):- Geometry of boot shape

The Boot Shape can easily help to enhance or boost up the heat transfer successfully, but in the most cases this associates with the large pressure loss penalties. Recently, boot shape roughnesstechnique became an attractive method for maintaining the thermal effect because boot shape roughness enhances heat transfer with very low pressure and friction.As there is an immense increase in a Reynolds number which leads to increase in the heat transfer, it also helps to increase pumping power which is higher in comparison to heat transfer. It has also been noticed that thermo-hydraulic performance of roughened solar air heatersusually increases with the increase in Reynold number.

CFD SIMULATIONS AND RESULTS

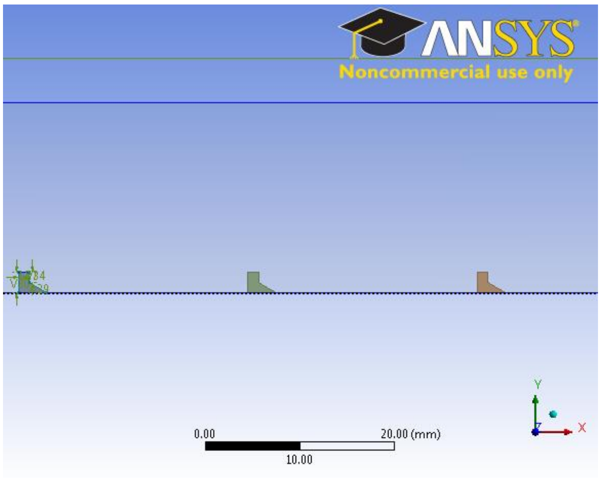

Fig.(2):- Simulation design of boot shape.

The figure 3 shows the comparison of Nusselt number in the case of boot shaped ribs between the ducts with different relative roughness slit-rib pitches p/e = 7, 8, 9, 10, 11, and 12 at height of the boot shape roughness e = 2.2 and angle of the boot shape roughness θ = 30˚ and smooth duct at the Reynolds number 2000 - 18000 In fig. (3)we have noticed that the Nusselt number was increased with respect to relative roughness pitch (p/e) increasing and maximum Nusselt

Nu = 157.432991 number was observed at p/e = 11 and e/Dh = 0.060, Reynold number Re = 18000.

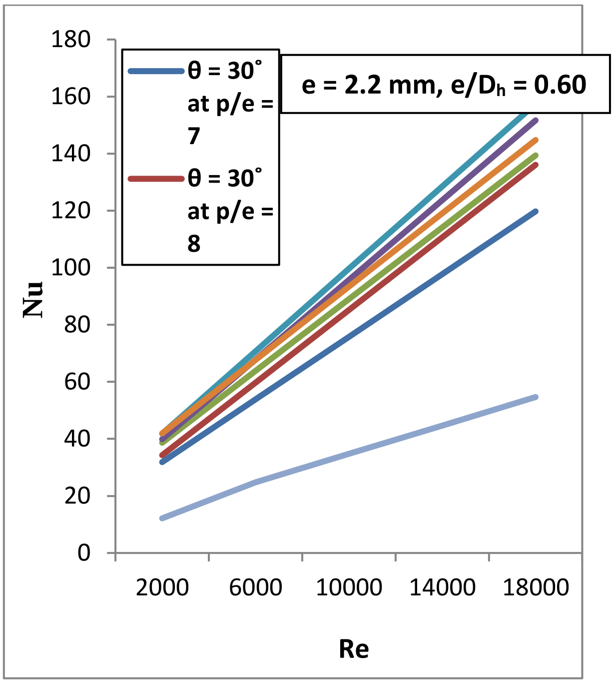

Fig.(3):-Shows the comparison of Nusselt numbers for roughness ducts with respect to the numerical investigation at θ = 30˚.

The figure 4 shows the comparison of Nusselt number in the case of boot shaped ribs between the ducts with different relative roughness slit-rib pitches p/e = 7 – 12 at θ = 40˚ and smooth ducts at the Reynold number 2000 - 18000 In the fig.4 we have noticed that the Nusselt number was increased with respect to relative roughness pitch (p/e) increasing and maximum Nusseltnumber Nu = 169.847681 was observed at p/e = 11 and e/Dh = 0.060, Reynold number Re = 18000.

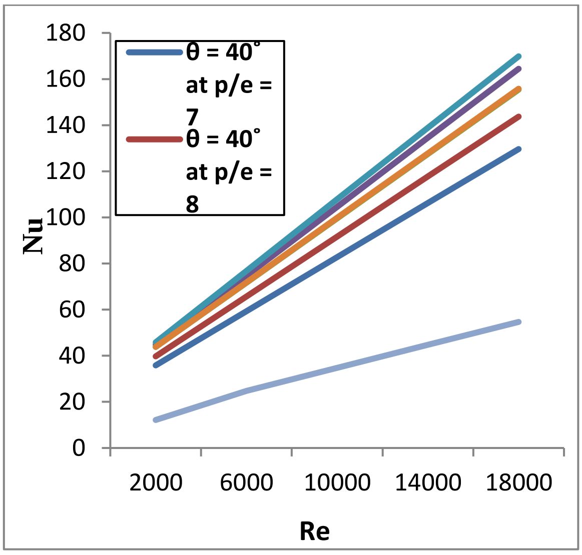

Fig.(4):- Comparison of Nusselt number for roughness duct with the respect to numerical investigation at θ = 40˚.

The figure 5 shows the comparison of Nusselt number in the case of boot shaped ribs between the duct with different relative roughness slit-ribs pitch p/e = 7, 8, 9, 10, 11, and 12at θ = 50˚and smooth duct at the Reynolds number 2000 - 18000 In fig. 5 we have been noticed that the Nusselt numbers were increased with relative roughness pitch (p/e) increasing and maximum Nusselt number Nu = 131.3815 was observed at p/e = 11 and e/Dh = 0.060, Reynold number 18000 and Nusselt number in smooth duct is lowest because no disruption in smooth duct.

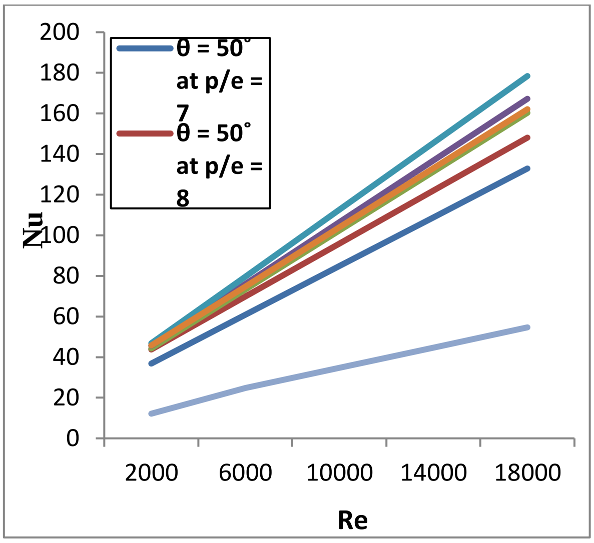

Fig.(5):- Comparison of Nusselt numbers for roughness duct with respect to the numerical investigationat θ = 50˚.

The given below figure 6 show the variation in the heat transfer rate with the variation of the relative pitch roughness p/e = 7, 8, 9, 10, 11, and 12 at θ = 60˚ andobtained maximum heat transfer Nu =148.660881 at P/e = 11.

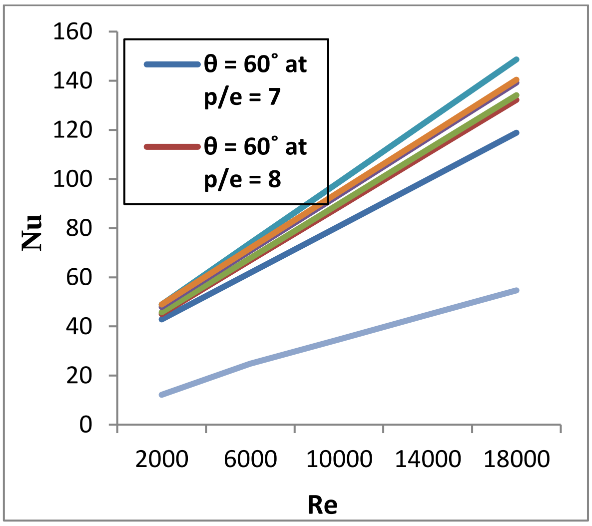

Fig.(6):-Shows the variations in the rate of heat transfer at θ = 60˚.

CONCLUSION

A tremendous research and investigation has been figured out from the last few decades using various numerical simulations which ultimately results to the proper enlightenment of heat transfer and fluid flow using our boot shape roughness. In this investigation,a numerical research has been conducted to studyheat transfer of boot shaped roughness ribduct of a solarair heater on the absorber plate. The effect of Reynolds number (2000, 6000, 10000, 14000, and 18000), relative roughness pitch (p/e = 7, 8, 9, 10, 11 and 12), Variation in the angle of attack on the roughness usedat rectangular duct and relative roughness height ratio (e/D_h= 0.060). The heat transfer taking place and frictional factor were studied very properly and are acknowledged with the help of Nusselt numbers and frictional factor as various plots shown above. In the numerical simulation the k-e model has been considered for the simulation purpose of turbulent flow in the rectangular duct.

The further very important and main conclusions are discussed below:-

1. According to the above result it is very clear that the k-ε model used provides us the results with very accurate acceptable engineering application for the proper analysis of fluid flow and the heat transfer patterns in the rectangular duct with addition of boot shaped ribs which results in increased thermal performance.
2. The increased numeric value of Reynolds number and increased slit-rib pitch to roughness height ratio results in the increase in the heat transfer and the decrease in friction factor.
3. The Nusselt number increases with the change in the boot shapeangle θ = 30°, 40°, 50° and 60˚ increases.
4. The highest Nusselt number **(Nu =169.847689)** is obtained at roughness height e/D=0.060, Reynolds number Re = 18000 and pitch ratio p/e = 11 and boot shape angle of attack θ = 40°, which has been discussed in plots.

NOMENCLATURE

Nu – Nusselt number
Re – Reynolds number
Pr – Prandlt number
θ – Angle of attack
p/e – Relative roughness pitch
e – Rib height
e/Dh – Relative roughness height

REFERRENCES

1. **01.Rajnesh Kumar, et al.2018 (Forward chamfered ribbed)** In this case the different values of the ratio of e/D was taken which helped to enhance the Nusselt numbers and the frictional factor was found to be very highfor the value of e/D at 0.043
2. **02.Sanjay K. Sharma, et al. 2017 (Thin ribs)** The Nusselt numbers were found to be very high and was 49.28 in case 1 and frictional factor was also found to be high upto the range of 2.88-7.18 in case 4.

3. **03.SompolSkullung, et al. 2017 (Pairs of trapezoidal-winglet groove)** The angle of attack was kept 45^0 and the heat transfer was higher and drop of pressure was increased upon the non-artificial channel.
4. **04.Khushmeet Kumar et al. 2017 (Arc Shaped Ribs set in 'S' shape)** This research was done on arc shape so there has been used a circular wire for investigation and heat transfer argument as well as friction factor relative to arc shape is significant to 0.6667.
5. **05.L. Varshney, et al. 2017 (Rectangular sectioned tapered rib)** There were 12 various tapered ribs at a taper of 1.6^0, 2.3^0 and 3.2^0 at the pitch of 10, 15, 20, and 25 mm were analysed at which maximum thermal enhancement was got at 1.6^0
6. **06. R.S. Gill et al. 2016 (Broken arc rib)** Here they placed a rib which was staggered between the broken ribs which resulted in a good friction factor and Nusselt number.
7. **07.Vipin B et al. 2016 (Reverse L Shape Rib)** Thermal performance is increased by the application of Reverse L shape and it occurred between 1.62 – 1.90
8. **08.Deep Singh Thakur et al. 2016 (Hyperbolic rib)** Using this hyperbolic rib the unwanted occurrence of small eddies at the corners are avoided and the heat transfer enhancement is increased.
9. **09.SompolSkullong, et al. 2016 (There geometry was STAGGERED – WINGLET PERFORATED TAPES)** The result was found such that when the perforated tapes with the winglet and staggered winglet resulted in a 1.2 times higher thermal enhancement factor than of non perforated tapes.
10. **10.Najah Ali, Al-Shamaniet al.2015 (Rib groove shapes)** In this case they concluded with the point of remarks that this type of grooves shows the very good heat transfer and best Nusselt number.
11. **11.H.T.Wang, et al.2015 (Textured asymmetric arc rib)** The parameters which were taken into consideration were p/e = 5 for compound rib including d/e = 0.06 and p/e = 6 for the asymmetric arc rib and triangular rib and this all progressed in the recital of the heat transfer.
12. **12.Mohammed O.A., et al. 2015 (Continuous rib turbulator)** The solid curved baffles which were placed nearby to the absorber plate that really resulted in the best performance valuation.
13. **13.Anil Kumar and Dongxu Jin, et al. 2014 (Multi v shape rib)** When multi v shape was compared to the smooth duct then the heat transfer enhancement was increased by 1.7-5.6
14. **14.SompolSkullong, et al. 2014 (Combined wavy-rib and groove turbulators)** The thermal performance was found to be 49-52% and the rib pitch was taken upto the channel height could easily apply groove alone so that it attains the maximum thermal performance.
15. **15.Anil Singh Yadav and Bhagoria J.L 2014 (Trasverse square rib)** This inner square section of ribs results into better thermal performance and the thermal hydraulic performance factor usually varied from 1.22 to 1.88