A NUMERICAL STUDY ON EFFECT OF RECTANGULAR SHAPED RIS ARRANGED IN DIFFERENT PATTERNS ON THERMAL PERFORMANCE OF A SOLAR AIR HEATER DUCT

Rajneesh Kumar Kushwaha[1], Akhilesh Sakya[2], Anas Ali[3]
M.Tech Research Scholar[1], Assistant Professor[2,3]
Mechanical Department
All Saints College of Technology Bhopal, India

ABTRACTS

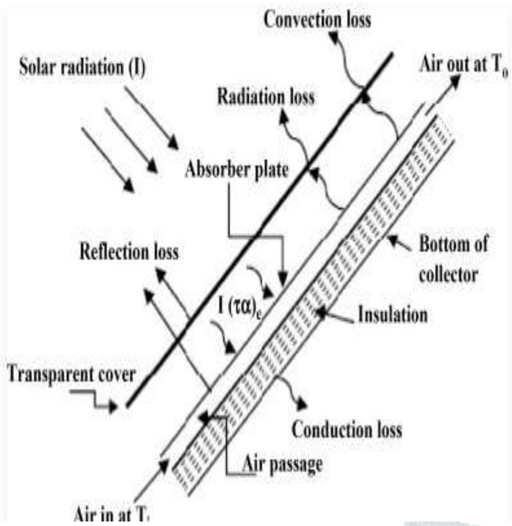
Solar air heater is a useful device that can be utilized to increase the temperature of air by extracting heat from solar energy. This thesis is concerned with a two-dimensional numerical study done to predict the influence of transverse rectangular cross-sectioned ribs on a solar air heater's convective heat transfer properties. Solar air heater is a rectangular duct consisting of an absorber plate on its top and heat falls only on the top of absorber plate. When Different shape and orientation of ribs/baffles are introduced just beneath the absorber plate, there is a considerable alteration in the thermal performance of air flowing through the rectangular duct. A comparison was taken between the results of thin (high aspect ratio) and square ribs arranged in three patterns, namely, single wall arrangement, staggered arrangement and in-line arrangement on two opposite walls. The variation between Nusselt number and Reynolds number range 4000-25000 was checked at a fixed rib pitch (p) and height (e) values. Computational fluid dynamics (CFD) simulations were performed using commercially available software ANSYS FLUENT v15.0. This result was compared with the existing (Previous) experimental ones while performing simulations under same conditions. Only Two methods were used to find the average Nusselt number in which one method extracted the local Nusselt number at many points and on averaging these, gave the average Nusselt number and the second method resembled the one used in the existing experimental work. The results was evident that as compared with smooth duct, the introduction of ribs led to a considerable augmentation in heat transfer. Good agreement was found between the existing numerical output and experimental results, Out of the three arrangements, the best thermal performance was given by thin inline ribs whose convective heat transfer coefficient was 1.93 times smooth duct's convective heat transfer coefficient.

Keyword- Solar, Energy, CFD, Convective, Heater, Reynolds, Nusselt Number

INTRODUCTION

Water and Energy both are the keys to modern life and provide the basis necessary for sustained economic development. Industrialized societies have become increasingly depend on fossil fuels for myriad uses. Mechanized agriculture, modern conveniences, and global population growth have only been made possible through the exploitation of inexpensive fossil fuels. Secure sustainable and future energy supplies will be the greatest challenge faced by all societies in this century. Because of growing world population and increasing modernization, global energy demand is projected to more than double during the first half of the twenty-first century and to more than triple by the end of the century. Current, the world's population is nearly 7 billion, and projections are for a global population approaching 10 billion by midcentury. In Future energy demands can only be met by introducing an increasing percentage of alternative fuels. Incremental development, in existing energy networks will be inadequate to meet this growing energy demand. Because of dwindling reserves and ever-growing concerns over the impact of burning carbon fuels on global climate change, fossil fuel sources cannot be exploited as in the past. Find sufficient supplies of clean and sustainable energy for the future is the global society's most daunting challenge for the twenty-first century. The future will be a mix of energy technologies with renewable sources of energy such as solar, wind, and biomass playing an increasingly important role in the new global energy economy. The main key question is: how long it will take for this sustainable energy changeover to occur? how much environmental, political, and economic damage is acceptable in the meantime? Solar energy is peerless in that it can easily provide electricity and purified water for these people today with minimal infrastructure requirements by using local energy resources that promote local economic development.

The Solar air heater is one of the basic equipment through which solar energy is change into thermal energy. Solar air heaters, due to their simple designing, are cheap and most widely used as a collection devices of solar energy. The main applications of the solar air heater are 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 traditionalor 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 rear plate and absorbing plate. A solar air heater is simple in shape or design and requires little maintenance. However, the value of the heat transfer coefficient between the absorber plate and air is low and this result in lower efficiency. For this reason, the surfaces are sometime roughened in the air flow passage.



Solar air heater

The use of artificial roughness on a surface is an effective technique to increase the rate of heat transfer to fluid flowing in a duct. Artificial roughness in the form of repeated ribs has been found to be a convenient method to increase the rate of heat transfer. Ribs of various shapes and orientations have been udes and the performance of such system has been investigated. Artificial roughness has been used to enhance the heat transfer coefficient by creating turbulence in the flow. It would also result in an increase in friction losses and hence greater pumping power requirements for air through the duct. In order to keep the friction losses at a low level, the turbulence must be created only in the region very close to the duct surface, i.e. in the laminar sub-layer.

The concept of artificial roughness was first used by Joule 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 equipment's nuclear reactors, and compact heat exchangers etc

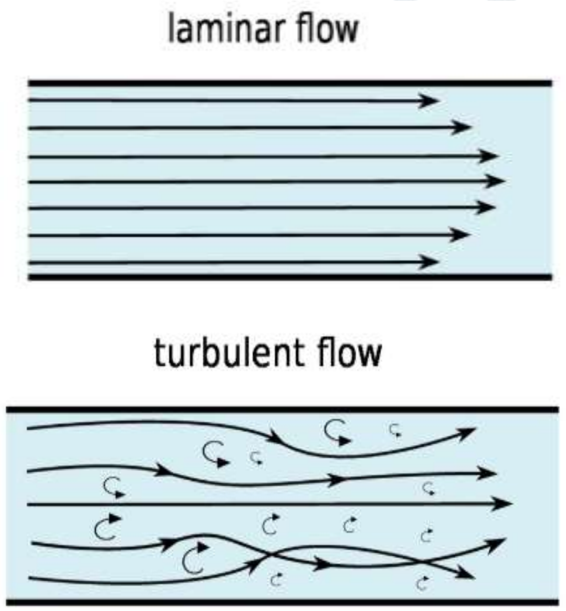
Convective heat transfer coefficient of turbulent flow and the friction factor are highly dependent on the surface roughness of the duct by which they passed. Hence, artificially roughened solar air heaters must be designed in such a way that their performance yields higher convective heat transfer rates from absorber plate to air low roughness to air flow. Research is being conducted in this field by many authors, whose work are generally involves performing experiments or carrying out numerical simulations with different types, sizes and patterns of ribs/ baffles and finding out the right parameters at which the heater gives optimal performance (minimum friction loss and maximum heat transfer). Some scientists, after performing research work on solar air heaters, develop a set of correlations for calculating Darcy's friction factor and Nusselt number in terms of operating and roughness parameters. The mechanism by which heat transfer, between air and roughened absorber plate, increases is breakage of laminar sub-layer. The introduction of ribs leads to local wall turbulence and 3 breakage of laminar sub-layers leading to periodic flow reattachment and separation. Vortices are formed near these baffles, which leads to a significant rise in Nusselt number.

Introduction to turbulent flow and Laminar flow

Turbulent flow is a flow in which the velocity components and other variables (e.g. pressure, density. if the fluid is compressible, temperature -

if the temperature is not uniform) at a point fluctuate with time in an apparently random fashion. In general, turbulent flow is time-dependent, rotational, and three dimensional. In turbulence flow velocity of fluid is high.

A laminar flow is characterized by regular and orderly motion of fluid elements. In this flow fluid is flow in straight flow with low velocity.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 flows occur when the inertial effects dominate the viscous forces.



Laminar and turbulence flow

Introduction to Turbulence modelling

Turbulent flows are given by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Since these fluctuations can be of high frequency and small scale, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the instantaneous (exact) governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve.

- **Spalart-Allmaras model**

- **k-ε models**

Standard k-ε model

Renormalization-group (RNG) k-ε model

Realizable k-ε model

- **k- *w* models**

Standard k-w model

Shear-stress transport (SST) k-w model

- **Reynolds stress model (RSM)**
- **Large eddy simulation (LES) model**

1. **RANS (Reynolds averaged Navier Stokes)/ RAS (Reynolds-average simulation)**: The statistical average form of Navier-Stokes equation is consider to model the turbulent flows. Some of the different types of RANS models are Renormalization group RNG-k-ε model, SST (Shear Stress Transport) k-ω model, Standard RNG-k-ε model, Standard SST-k-ω model and Realizable k- ε model.

2. **DNS (Direct Numerical Simulation):** Resolving all scales of turbulence, the Navier Stokes equation is numerically solved by this method.

3. **LES (Large Eddy Simulation):** LES method involves solving large turbulent eddies with the help of governing differential equations, whereas the sub-grid scales are modelled.

4. **DES (Detached eddy simulation):** The Combination of LES and RAS. The near-wall regions are treated with the help of RAS whereas the bulk flow regions are solved with the help of LES.

OBJECTIVE

The performance of solar air heaters are greatly altered by changing parameters such asduct's internal surface roughness and flow velocity of air. The average Nusselt number is strongly depend on these parameters. So this concept can be used in a positive way to increaseconvective heat transfer between air flowing inside the duct and the absorber plate, convective heat transfer. Numerical studyof solar air heaters using CFD software is an excellent tool to understand in detail how flowbehaves under the presence of obstacles in the solar air heaters. CFD results are more accurate ascompared to experimental results data. The most benefits of using CFD software are saving of time and lesser costs are required to complete the work. Hence, the aim of this work is to prove that CFD can be effectively used to design solar air heaters based on their thermal performance. In this study, a computational model was constructed to find out the solar air heater's thermal performance.

NUMERICAL SETUP

This work is concerned with carrying out two-dimensional simulations on an artificially roughness solar air heater, through which air of air flows. The internal surface of air heater was roughness with the help of transverse-square and thin (high aspect ratio) ribs. The ribs were arrange in different patterns namely one wall only, staggered and in-line on both lower and upper faces.

Computational fluid dynamics (CFD) provide a path of real predicting the behavior of a real fluid without having to perform any experiment and change in the problem setup are easily made. The CFD simulation is created in some step such as

Geometry modeling

Meshing consideration

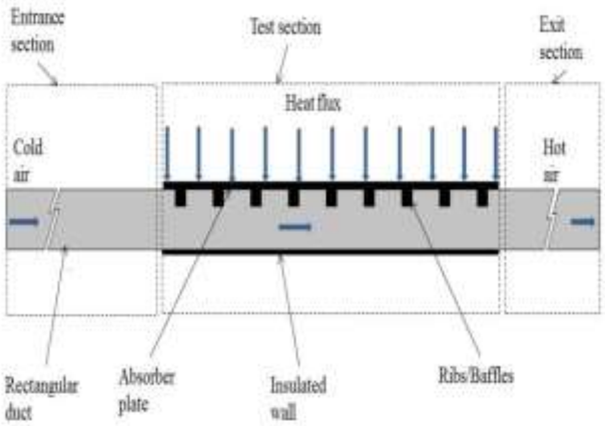
Boundary condition

Numerical calculation and Governing equation

Result and discussion

Geometry modeling

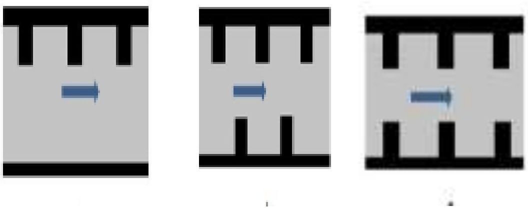
A rectangular section was considered. Rectangular section consist of three sections, test section of length L2,entrance section of length L1 and exit length of length L3. The domain on which numerical simulations were performed was two-dimensional. The rectangular duct was of length 2000 mm, width 300 mm and 30 mm with a test section length of 440 mm. Hence our domain test section length was 440 mm and its entrance and exit length dimensions were selected on the basis of ASHRAE recommendations, according to which an exit length more than 2.5√WH and entrance length more that 5√WH.



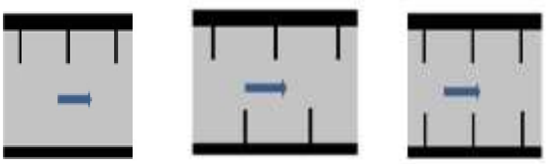
ANSYS Geometry of test section

Shows the geometry of the computational domain.

The different rib arrangements employed for simulation are indicated



Single square ribs (B) Staggered square ribs (C)In-line square ribs

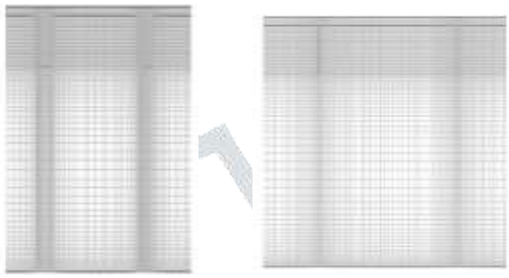


(D)

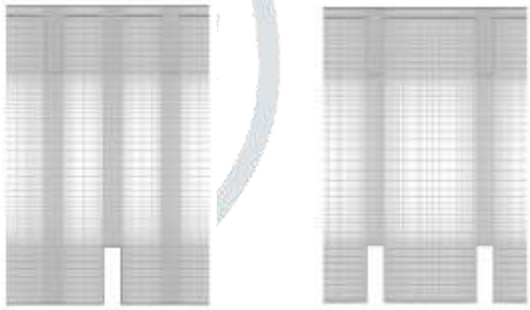
Single thin ribs (E) Staggered thin rib (F)In line thin ribs

Meshing consideration

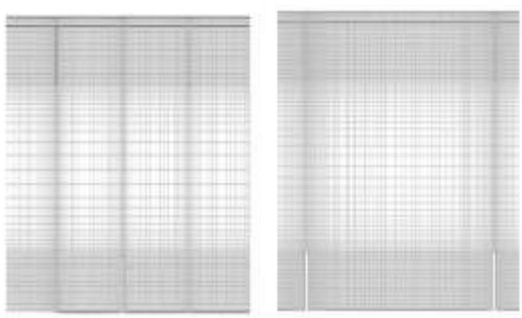
The result of numerical analyses depend upon the meshing of the model that is used. A too coarse mesh will give a higher error in the result and finer mesh size give low error in the final result. A fine mesh is require where the flow is changing rapidly, while a coarser mesh can be used where flow is uniform and less accuracy is needed . If size of mesh element is small enough so that numerical result is nearest to the experimental result



(A) Single square ribs (B) Single thin ribs



(C) Staggered square ribs (D) In-line square ribs



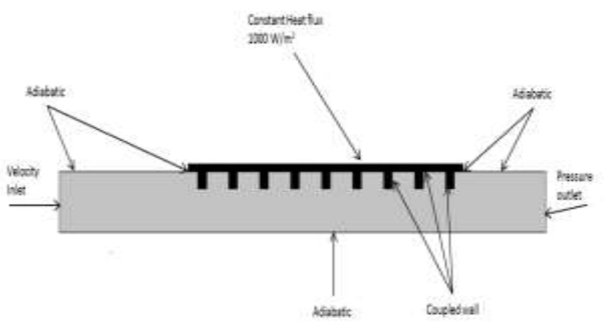
(E) Staggered thin ribs (F) In-line thin ribs

Boundary condition

Operating and Geometrical parameters used for CFD analysis

Test length of duct,	L_2 440 mm
Entrance length of duct	L_1 500 mm
Exit length of duct	L_3 240 mm
Duct height, H	30 mm
Duct width, W	300 mm
Duct hydraulic diameter, D_h	54.54 mm
Aspect ratio of duct,W/H	10
Constant heat flux	, q" 1000
W/m^2	
Range of Reynolds number	5000-23000

Different boundary conditions assigned to edges of computational domain



Different boundary conditions assigned to edges of computational domain

In this model we consider no-slip boundary conditions . Constant heat flux of 1000 W/m^2 was decided to be the boundary condition at the upper wall of the absorber plate. At the inlet, uniform velocity with an inlet temperature of 300 K and at the exit, invariable pressure (atmospheric pressure) boundary conditions were assigned. All the other edges were assigned as walls with insulated boundary conditions

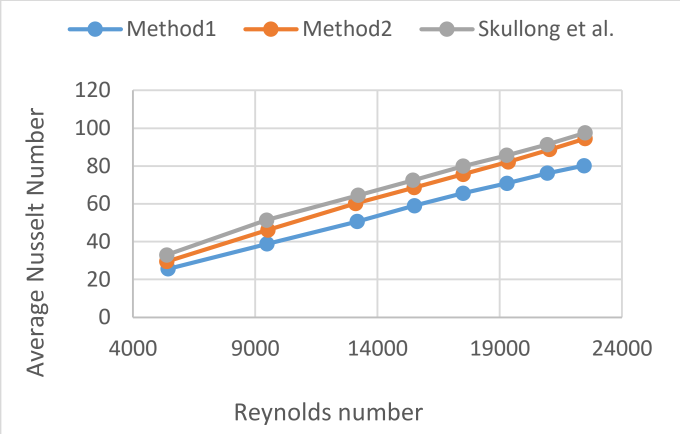
RESULT AND DISCUSSION

5.1Results and Discussions

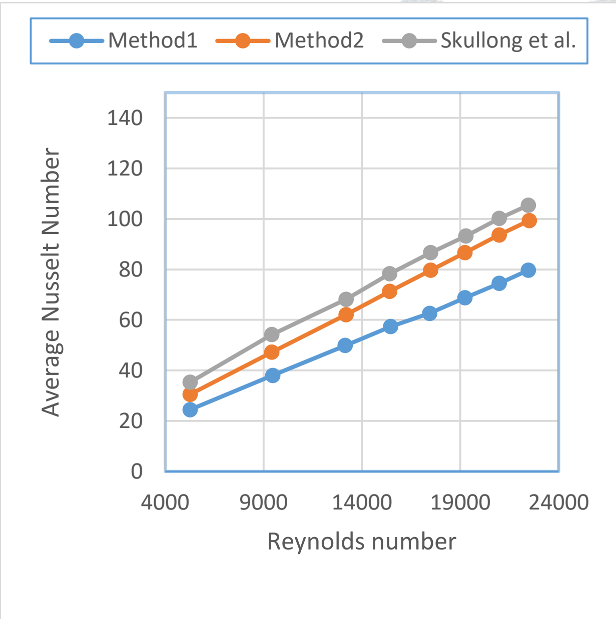
In this project, a computational model was constructed to measure the thermal performance of a solar air heater's. Solar air heater are consisted of baffles/ribs just below its absorber plate.

Simulation for different roughened ducts at different Reynolds number

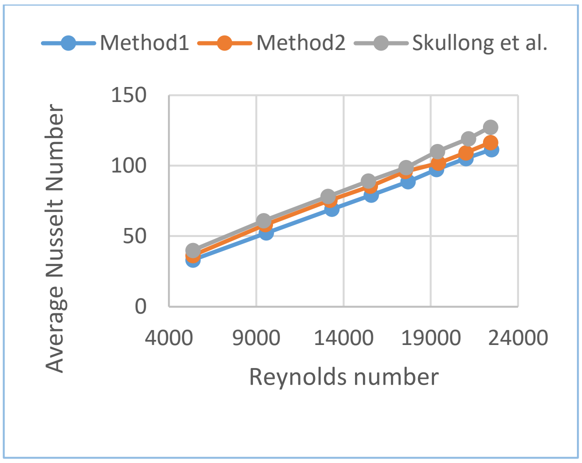
The average convective Nusselt number was measured using two methods. The first method (method-1) is given by equation 3.7 and takes the average of all 32 the local Nusselt numbers along the test section length. The second method is the most widely employed method of calculating Nusselt number in experimental works. The second method was used by Skullong et. clearly depicts the outcome of average Nusselt number alteration with Re for the geometries separately. As Re was raised, the average Nusselt number increased for all cases. The reason why this trend was observed was that as the Re was raised, the flow became more turbulent (more dominance of inertial effects over viscous effects) and hence the heat transfer rate increased. There was a decent agreement when numerical outputs were compared with the experimental ones existing in the literature. The Excellent matching between Nusselt numbers calculated using method 2 and the existing ones was observed for square ribs but good agreement was found in case of thin ribs.



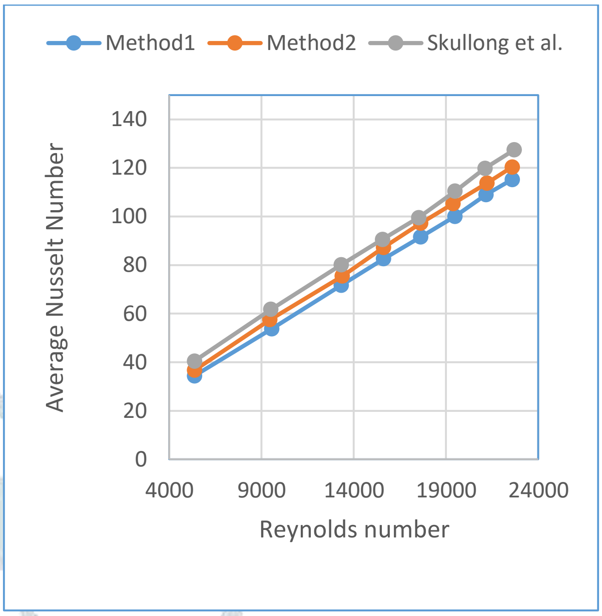
Results of numerical analysis at different Reynolds number for single square ribs



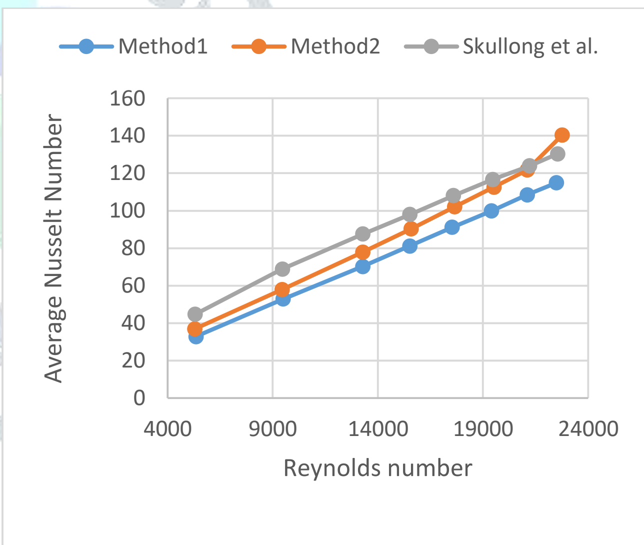
Results of numerical analysis at different Reynolds number for single thin ribs



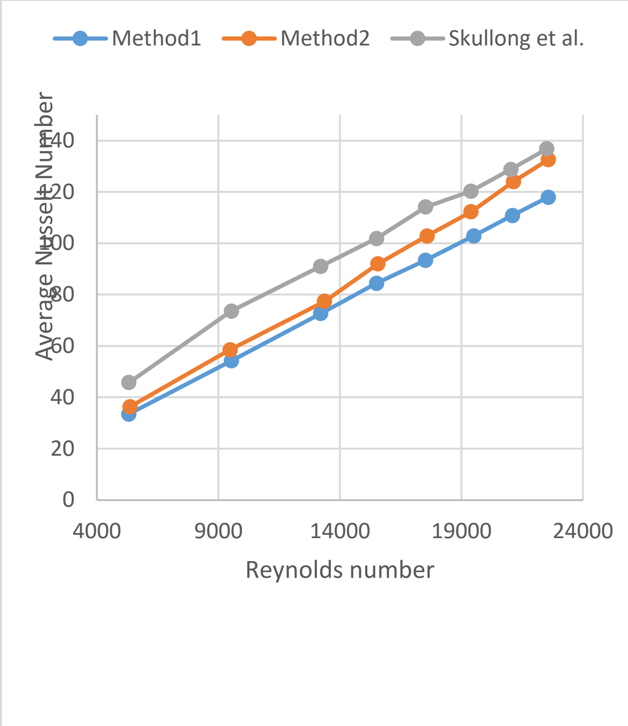
Results of numerical analysis at different Reynolds number for staggered square ribs



Results of numerical analysis at different Reynolds number for In-line square ribs



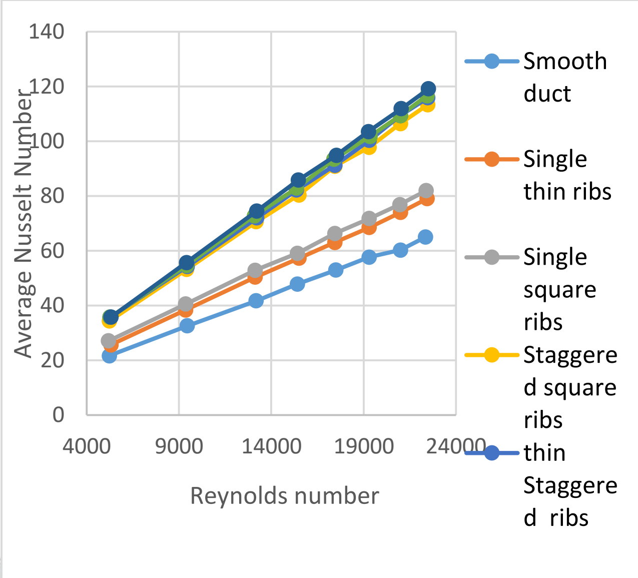
Results of numerical analysis at different Reynolds number for staggered thin ribs



Results of numerical analysis at different Reynolds number for Inline thin ribs

The Comparison of Nusselt number variation with Reynolds number for all the Geometries

Graphically outlines a comparison between Nusselt number and Reynolds number for all the geometries. It can be found from the graph that there was a considerable augmentation in Nusselt number for both thin and square ribs. Interestingly, thin ribs gave much better thermal performance than their square counterparts. In this simulation In-line gave the highest Nusselt number for the complete Reynolds number range.



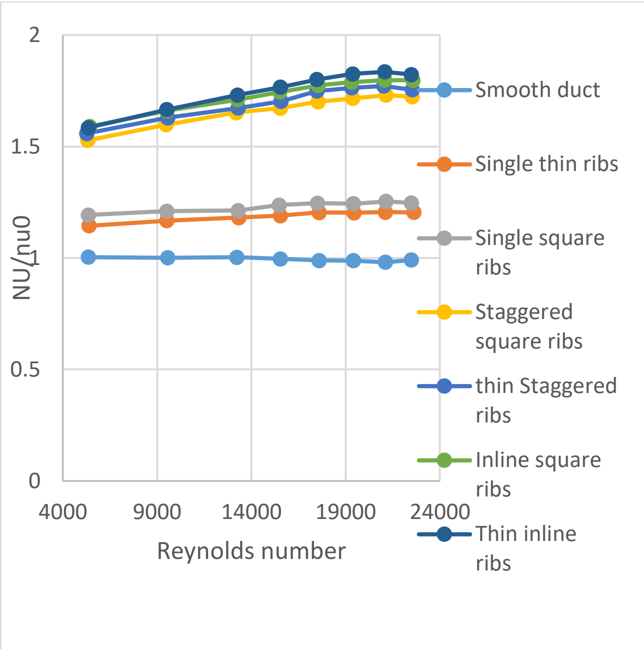
Variation of Nu with Re for all the cases

The ribbed configurations in the increasing order of Nusselt number are single thin ribs, The single square ribs, staggered square ribs, in-line square ribs, thin staggered ribs and thin inline ribs. The same pattern was observed in the experimental results.

Nusselt number enhancement (Nu/Nu0) versus Reynolds number for separate geometries

The Nusselt number increasing ratio gives information about the increment in Nusselt number of asolar air heater with ribs from that of a smooth one. Fig. gives a comparison Nusselt numberincreasing alteration with Re for all the arrangements. In-line thin ribs yielded the best Nusselt number enhancement ratio. The maximum Nu/Nu0 values with square ribs were 1.2, 1.72, 1.76 for single, staggered and in-line patterns respectively and those with thin ribs were 1.23, 1.79 and 1.82 respectively thereby clearly indicating that thin ribs have advantage over the square ones.

The highest value of Nu/Nu0 was observed for thin-in line ribs at Re = 21040. In the experimental work of Skullong et al. , the maximum Nu/Nu0 values with square ribs were 1.52, 1.82 and 1.87 1.74 for single, staggered and in-line patterns respectively and those with thin ribs were 1.65, 2.06 and 2.15 respectively.



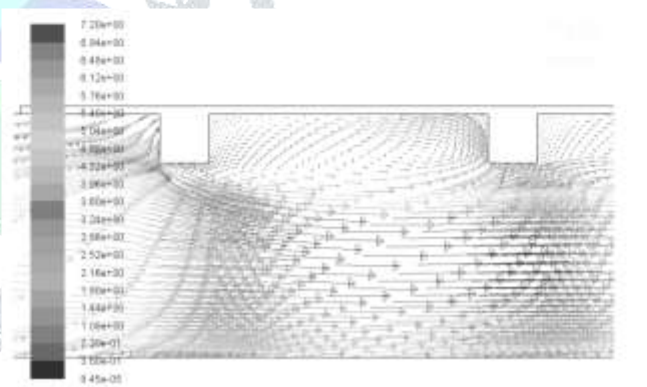
Variation of Nu/Nu0 with Re

Velocity characteristics

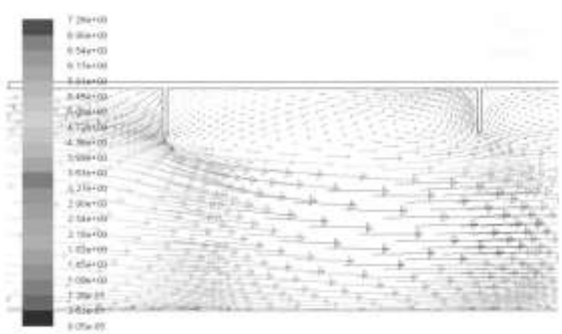
There is an extreme irregular in the contours of instantaneous velocity due to the effect of ribs. The eddy formation is clearly shown in the contours plot, in the vicinity of ribs. This fluid flow experiences sudden expansion leading to a separation region at the rib downstream. Separation is followed by reattachment of the fluid ahead the next rib. The periodic circulation of reattachment of flow leads to the formation of a small secondary vortex near the absorber plate and fluid junction.

The secondary vortices are more strongly created in case of thin ribs. Furthermore, it is observed that the velocity of the fluid is accelerated lengthwise as it passes through the test channel. Hence greater amount of disturbances are observed in the laminar sub layer thereby leading to a reduction in conductive transfer of heat and a significant augmentation in convective transfer of heat.

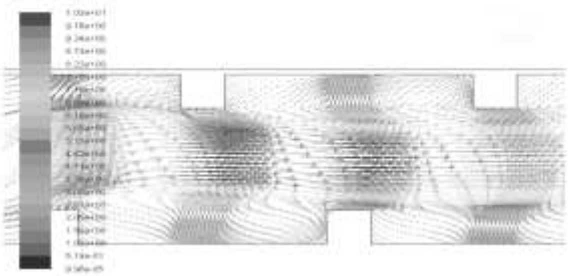
The formation of vortices and the stream line paths are exaggeratedly indicated in Fig. 4.8. The vortices are much stronger and bigger for in line configurations although there is more complexity in the velocity pattern in case of staggered rib patterns.



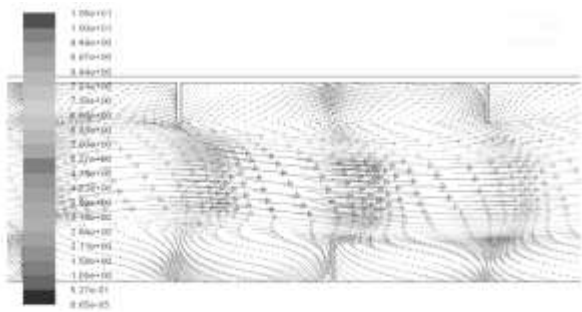
Velocity vector contours of turbulent flow for Single Square ribs



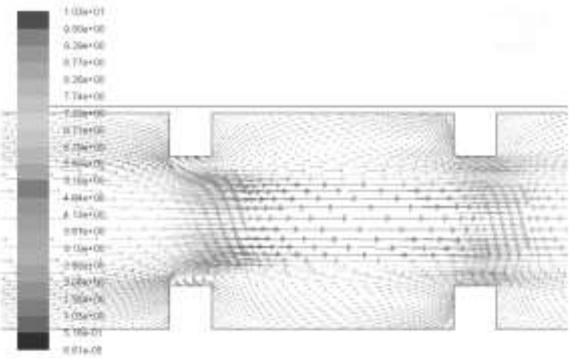
Velocity vector contours of turbulent flow for Single thin ribs



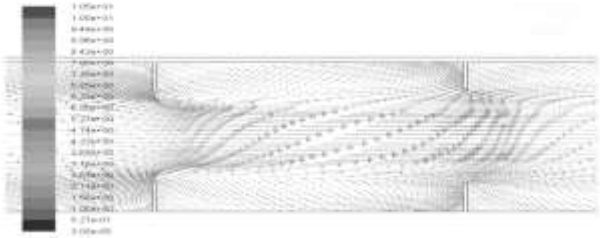
Velocity vector contours of turbulent flow for Staggered Square ribs



Velocity vector contours of turbulent flow for In-line square ribs



Velocity vector contours of turbulent flow for Staggered Thin ribs



Velocity vector contours of turbulent flow for In-line thin ribs

CONCLUSION AND FUTURE SCOPE

Rectangular ductwas made and numerical analysis was carried out on square and thin (high aspect ratio) ribshapes arranged in different fashion, namely single wall, staggered and In line ribs arrangement ontwo opposite walls including the absorber plate. In this project air was the working fluid and constant heat fluxwas applied only on the absorber plate’s top surface. The final result of numerical simulations drew the following conclusions

1.It was found that SST-k-omega can best predict thethermal performance of the solar air heater,

2.On comparing simulation results, pertaining to smooth duct’s average Nusselt number,for different turbulent models In this study,

3.Increase in Reynolds number leads to augmentation in Nusselt numberfor all the cases considered .

4.When baffles/ribs are placed just beneath the collector plate, there was a considerable alteration in the heat transfer coefficient of air.

5.From results it was found that that the thin ribs yielded better performance than the squared ones.Similar results were also observed by Skullong et al.in their experimental work.The staggered ribs gave lower Nusselt number than the in-line ones.

REFERENCE

1. 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

2. Kalogirou, S. A., 2013, Solar energy engineering: processes and systems. Academic Press.

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

4. Sukhatme K. and Sukhatme S. P., 1996, Solar energy: principles of thermal collection and storage, Tata McGraw-Hill Education.

5. Twidell, J. and Weir, A. D., 2006, Renewable energy resources, Taylor & Francis.

6. Cengel Y. A. and Cimbala J. M., 2006, Fluid mechanics (Vol. 1), Tata McGraw-Hill Education.

7. Prasad K. and Mullick S. C., 1983, Heat transfer characteristics of a solar air heater used for drying purposes, Applied Energy, 13(2): pp. 83-93.

8. Prasad B. N. and Saini J. S., 1988, Effect of artificial roughness on heat transfer and friction factor in a solar air heater, Solar Energy, 41(6): pp. 555-560.

9. Prasad B. N. and Saini J. S., 1991, Optimal thermo-hydraulic performance of artificially roughened solar air heaters, Solar Energy, 47(2): pp. 91-96.

10. Liou T. M. and Hwang J. J., 1993, Effect of ridge shapes on turbulent heat transfer and friction in a rectangular channel, International Journal of Heat and Mass Transfer, 36(4): pp.931-940.

11. Gupta D., Solanki S. C. and Saini J. S., 1993, Heat and fluid flow in rectangular solar air heater ducts having transverse rib roughness on absorber plates, Solar Energy, 51(1): pp. 31- 37.

12. Saini R. P. and Saini J. S., 1997, Heat transfer and friction factor correlations for artificially roughened ducts with expanded metal mesh as roughness element, International Journal of Heat and Mass Transfer, 40(4): pp. 973-986.

13. Karwa R., Solanki, S. C. and Saini, J. S., 1999, Heat transfer coefficient and friction factor correlations for the transitional flow regime in rib-roughened rectangular ducts, International Journal of Heat and Mass Transfer, 42(9): pp. 1597-1615.

14. Verma S. K. and Prasad B. N., 2000, Investigation for the optimal thermos-hydraulic performance of artificially roughened solar air heaters, Renewable Energy, 20(1): pp. 19-36.

15. Murata A. and Mochizuki S., 2001, Comparison between laminar and turbulent heat transfer in a stationary square duct with transverse or angled rib turbulators, International Journal of Heat and Mass Transfer, 44(6): pp. 1127-1141

16. Ahn S. W., 2001, The effects of roughness types on friction factors and heat transfer in

roughened rectangular duct, International Communications in Heat and Mass Transfer, 28(7): pp. 933-942

17. Momin A. M. E., Saini J. S. and Solanki S. C., 2002, Heat transfer and friction in solar air heater duct with V-shaped rib roughness on absorber plate, International Journal of Heat and Mass Transfer, 45(16): pp. 3383-3396

18. Chandra P. R., Alexander C. R. and Han J. C., 2003, Heat transfer and friction behaviors in rectangular channels with varying number of ribbed walls. International Journal of Heat and Mass Transfer, 46(3): pp. 481-495

19. Tanda G., 2004, Heat transfer in rectangular channels with transverse and V-shaped broken ribs, International Journal of Heat and Mass Transfer, 47(2): pp. 229-243.

20. Sahu M. M. and Bhagoria J. L., 2005, Augmentation of heat transfer coefficient by using 90 broken transverse ribs on absorber plate of solar air heater, Renewable Energy, 30(13): pp.2057-2073.

21. Chang S. W., Liou T. M. and Lu M. H., 2005, Heat transfer of rectangular narrow channel with two opposite scale-roughened walls, International Journal of Heat and Mass Transfer, 48(19): pp. 3921-3931

22. Bhagoria J. L., Saini J. S. and Solanki S. C., 2002, 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(3): pp. 341-369

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

24. Jaurker A. R., Saini J. S. and Gandhi B. K., 2006, Heat transfer and friction characteristics of rectangular solar air heater duct using rib-grooved artificial roughness, Solar Energy, 80(8): pp. 895-907

25. ang C., Guan Z., Zhao X. and Wang D., 2006, Numerical simulation study on transpired solar air collector, Renewable Energy Resources and a Greener Future, Vol. VIII-3-4

26. Karmare S. V. and Tikekar A. N., 2007, Heat transfer and friction factor correlation for artificially roughened duct with metal grit ribs, International Journal of Heat and Mass Transfer, 50(21): pp. 4342-4351