A review of CFD methodology used in literature for predicting heat transfer characteristics of a roughened solar air heater

[1]Amit Semalty, [2]Sumit Kumar, [3]Vijay Singh Bisht

[1]amitsemalty@gmail.com, [2] sumit.pce90@gmail.com, [3]vsinghbisht5@gmail.com
[1]M.tech (Department of thermal engineering)
[1]Faculty of Technology(UTU Dehradun), Uttarakhand, India

Abstract: A solar air heater is experimentally explored by many researchers to study the effect of operating and geometrical parameters on heat transfer and fluid flow characteristics. The accessibility of highly advanced computer hardware and progress in numerical methods, motivate the future researchers to carry out the simulations of solar air heater using various roughness geometries with different ranges of operating and geometrical parameters. CFD is emerging as a suitable and efficient tool to detect the optimum configuration of rib for its maximum thermo hydraulic performance before carrying out the actual experimentation. This article discusses the different approaches used to experimentally investigate the thermal performance of solar air heater. The computational approach i.e. CFD methodology is discussed using the reference of commercial CFD software ANSYS FLUENT. The main objective of the article is to present a detailed review of the literature that deals with the application of CFD in the design of solar air heater. This paper presents a concise summary of each reviewed article in the form of, type of computational domain (2D/3D), turbulence model used, type of CFD commercial software used, range of operating and geometrical parameters and best results obtained from CFD. The article also tabulated the optimum values of relative roughness pitch (P/e) and relative roughness height (e/D) for maximum heat transfer enhancement in roughened solar air heater duct.

Index Terms **- CFD, ANSYS FLUENT, Artificial roughness, SAHs.**

I. INTRODUCTION

A solar air heater is a thermal system which is used to convert solar energy into thermal energy. There is a wide variety of application of the solar air heater which involves drying of agricultural and marine products, space heating and heating of buildings to maintain comfort in the winter season. Solar fish dryer is a very efficient tool for small fisherman groups. It has been very well noted that the performance efficiency of solar air heater is very low because of the low convective heat transfer coefficient between absorber plate and flowing working fluid (air).The reason for the same can be cited that the presence of a laminar viscous sub layer is the cause for the convective heat transfer coefficient. The resistance to heat transfer arises due to this laminar viscous sub layer that is eliminated by providing artificial roughness on the underside of the absorber plate. Many researchers investigated the effect of various roughness geometries on heat transfer and friction characteristics in the solar air heater. The researchers have also developed mathematical algorithms as an analytical tool to simulate solar thermal systems and to optimize the thermal performance of solar air heater. Artificial roughness on the absorber plate towards the side of the flow is an effective method to improve the performance of an air heater. Investigators are trying to focus the attention on finding out the best suitable roughness geometry of the absorber plate which could give best performance for mass flow rate used in SAHs. A lot of studies have been reported in the literature on artificially roughened surfaces for heat transfer improvement but most of the studies were carried out with two opposite or all the four walls roughened. An early study on the effect of roughness on friction factor and velocity distribution was performed by Nikuradse[2], who conducted a series of experiments with pipes roughened by sand grains and since then many experimental investigations were carried out on the application of artificial roughness in the area of gas turbine airfoil cooling system, gas cooled nuclear reactors, cooling of electronic equipment, shipping machineries, combustion chamber liners, missiles, re-entry vehicles, ship hulls and piping networks etc. Literature survey indicates that most of the studies before year 2000 in this field were conducted experimentally. Prasad and Saini [4] used small-diameter protrusion wires and concluded that geometrically similar roughness (for a given P/e and e/D) produce the same effect on heat transfer and friction factor. Gupta et al.[5] conducted experimental study on transverse wire as roughness element to determine optimum design and operating conditions. Karwa et al.[6] used chamfered rib roughness to experimentally investigate and optimize chamfer angle, Sawhney et al.[7] investigate staggered and inline wavy delta winglets to get optimum configuration on the base of thermo-hydraulic performance. In experimental investigation possibility to make perfect geometry and to analyze flow phenomena is quite difficult as this will require testing of number of plates and involve considerable amount of time and money. CFD investigation overcomes this difficulty. CFD simulations are relatively inexpensive and time taken in computation is relatively less. Time requirement is likely to decrease further with the use of high speed computers. For this reason researchers are moving towards CFD investigations on SAHs.

NOMENCLATURE

Aa - Gross collector area
Ac - Frontal area, mm2
Dh - Equivalent or hydraulic diameter of duct, mm
g - Groove position/width of gap, mm

H - Depth of duct, mm
I - Intensity of solar radiation, W/m2
L - Length of test section of duct or long way length of mesh, mm
P - Pitch, mm
qu- Useful heat flux, W/m2
Qu- Useful heat gain, W
Tam - Mean air temperature, K
Ti - Fluid inlet temperature, K
Tf - Fluid outlet temperature, K
Tpm - Mean plate temperature, K
W - Width of duct, mm
w - Width of rib, mm
ΔP - Pressure drop, Pa
SAHs- Solar air heaters

DIMENTIONLESS PARAMETER:
e/D - Relative roughness height
e/H - Rib to channel height ratio
Fo - Collector efficiency factor
Fr - Collector heat-removal factor
fr - Friction factor for rough surface
g/e - Relative gap width
g/p - Relative groove position
Gd/Lv - Relative gap distance
L/e - Relative log way length of mesh
Nu - Nusselt number
Nur - Nusselt number for rough channel
Nus - Nusselt number for smooth channel
P/e - Relative roughness pitch
Pr - Prandtl number
St - Stanton number
W/H - Duct aspect ratio
W/w - Relative roughness width

GREEK SYMBOL:
Φ - Wedge angle/chamfer angle, degree
α - Angle of attack , deg.
δ - Transition sub-layer thickness, mm
ε - Dissipation rate,m^2/s^3
η_{th} - Thermal efficiency
μ -Dynamic viscosity, Ns/m^2
ρ - Density of air, kg/m3
ω - Specific dissipation rate,1/s

II. METHODS FOR SOLAR AIR HEATER ANALYSIS:

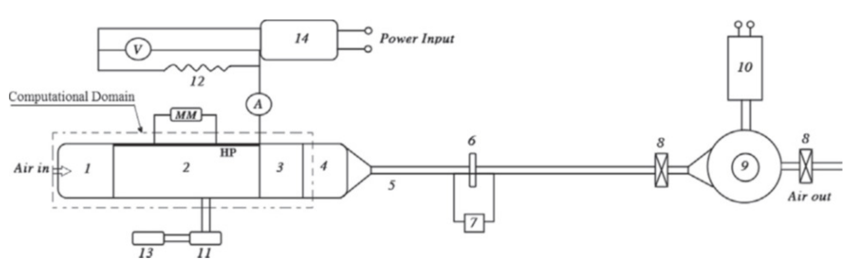
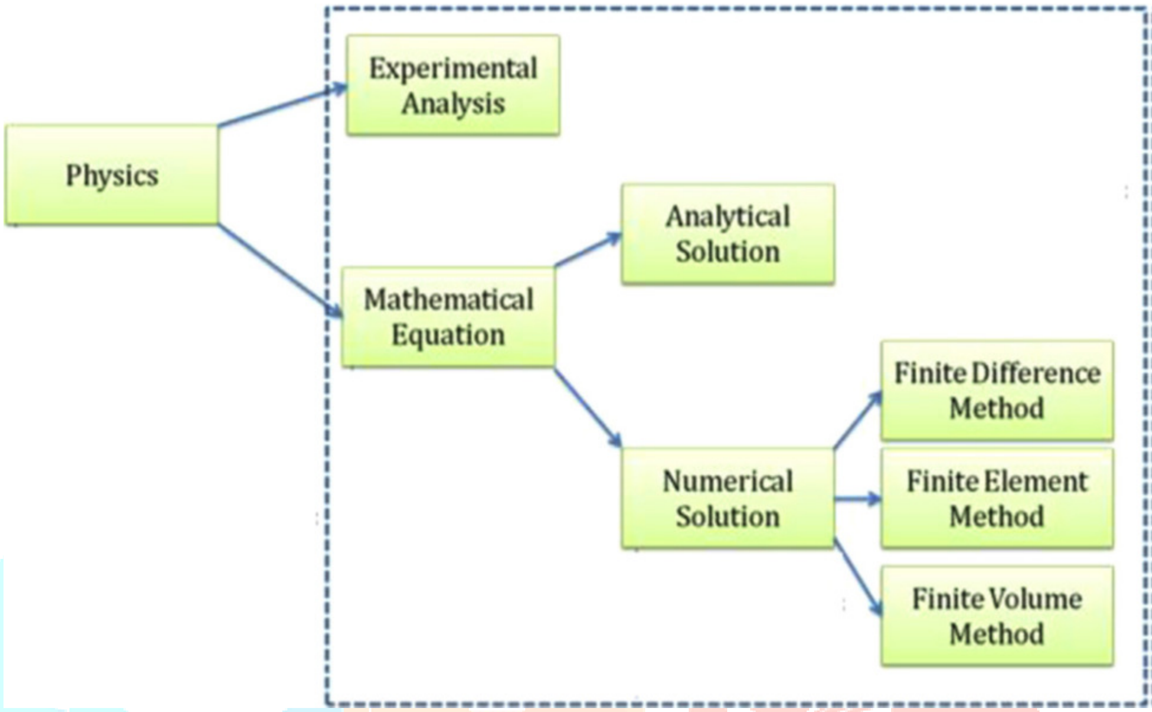


Fig.1.Experimental setup of solar air heater
1 Entry Section 8 Control Valves
2 Test Section 9 Centrifugal Blower (Suction type)
3 Exit Section 10 Electric Motor
4 Plenum 11 Selector Switch
5 G.I Pipe 12 Variable Transformer
6 Orifice Plate 13 Milli-Voltmeter

7 U-Tube Manometer 14 Power Source
A-Ammeter V-Voltmeter
MM Micro-Manometer HP Heater Plate



2.1. Experimental approach

In this approach, a prototype of a solar air heater is manufactured and experiments are performed on this prototype. The Proto type may be of the same dimensions as that of the actual solar air heater or a scale model (reduced or enlarged).The operating parameters are used for the evaluation of thermal performance of a solar air heater. During experimentation, the Researcher should take care of factors like time required for the study, cost involved in the manufacturing of prototype, availability of experimental facility and measurement devices. Human error, measurement error, atmospheric condition while conducting experiments affect the accuracy of the data collected from the experimental setup. Many investigators have carried out experimental analysis with various roughness geometries using a different range of operating and geometrical parameters and attempted to predict accurate values of heat transfer coefficient and friction factor for a roughness geometry considered for investigation.

2.2. Analytical/mathematical approach

The physics of fluid and heat flow in a solar air heater is Expressed in the form of mathematical equations (differential or integral) and are solved using analytical approach. The mathematical equations correlate different properties of the flow like velocity, temperature, density, viscosity etc. The ease of solving these mathematical equations depends upon the complexity of the equations, boundary conditions and complexity of geometry. The mathematical equations for general fluid and heat flow governing equations are second order non-linear partial differential equations and it is difficult to get the closed form solution of these Partial differential equations. To simplify these complex mathematical equations, the researcher has to put many assumptions for the problem under consideration like flow is incompressible, no slip boundary, negligible heat losses, inviscid flow etc. These Assumptions remove some terms of the governing equations and Simplified equations can now be solved. An analytical approach was used by Ammari [9] to develop a mathematical model for predicting the thermal performance of a solar air heater with slats. An Exact analytical solution for fully developed convective heat Transfer in rectangular ducts under a constant heat flux was derived by Shahmardan et al.

2.3. Computational approach:

Computational fluid dynamics (CFD) is the science of determining a numerical solution to the governing equations of fluid flow whilst advancing the solution through space or time. This Approach (CFD) is used in various applications like aerospace, automotive, biomedical, chemical processing, HVAC, hydraulics, marine, oil and gas, power generation and sports. The use of computational approach(CFD)mainly taken into consideration when physics is well captured in a set of governing equations, specific numerical methods are available for governing equations and when computational resources are available for solving numerical algorithm. The prediction of physics of fluid and heat

Flow of solar air heater in these numerical methods is carried out using the following steps:

1. Mathematical Modeling. The equations governing, the fluid flow is derived from mathematical statements of the three fundamental physical principles viz. mass is conserved (continuity equation), Newton's second law(momentum equation)and energy is conserved (energy equation).These governing equations together are also called as Navier–Stokes equations. Governing equations can be written in terms of Cartesian(x, y, z), cylindrical (r, θ, z) or spherical(r, θ, ϕ) coordinates in space to get better resolution of the geometry.

2. Discretization of PDE's. Flow equations are coupled non-linear Partial differential equations (PDE) as changes in the flow variables are represented by partial derivatives. In computational approach, terms in the PDE's are converted into a Computer analogue, which is then solved using the selected numerical method. This process is called "Numerical discretization" and is achieved using the finite difference method, finite element method or finite volume method. The salient features of these methods are:

a. Finite Difference Method (FDM)
- Governing equations are in integral form
- Computational domain is divided into number of grid nodes
- One algebraic equation per grid node
- Linear algebraic equation system
- Applied to structured grids

b. Finite Element Method (FEM)
- Computational domain is divided into finite number of elements
- Governing equations are solved for each element.
- Overall solution of the domain is obtained by combining the Solution at each element.

c. Finite Volume Method (FVM). This method was developed by Patankar [11].
- Governing equations are in integral form
- Computational domain is sub-divided into number of control volumes
- Governing equations are applied for each control volume
- Computational node locates at the centroid of each control volume.
- This method is applied for all types of grids including complex geometries.

Various CFD software tools use above discretization methods for solving governing equations. ANSYS FLUENT is a commonly used CFD code by previous researchers for carrying out analysis of a solar air heater. The CFD code ANSYS FLUENT carried out the analysis in three main stages as(i)pre-processing(ii)solver and (iii) post-processing. These three stages are briefly explained by considering the computational domain of solar air heater analyzed by yadav et al. [12] with circular transverse wire rib roughness on the absorber plate, in the following subsections.

2.3.1.Pre- processing:

This is the first step of the CFD simulation process. In this step initially we have to think of our modeling goals like the initial Assumptions (steady, unsteady, inviscid, laminar, turbulent etc.) Applied to a given problem to simplify it, the degree of accuracy needed, computational time etc. After gathering this information, the analysis includes following main steps:

i) Identify the domain of interest. Out of the total assembly of a solar air heater only a rectangular duct with entry, test and exit section, absorber plate and heater plate is selected for computational domain. This is done to simplify the physical model.

ii) Creating a solid model of the domain. In the literature, this selected computational domain is approximated as 2D or a 3D problem. To save computational time, analysis was carried out with 2-D approximation (for continuous roughness geometries like chamfered rib, rectangular, square, circular, semicircular, triangular etc). 3-D approximations may be utilized for complex roughness geometries like V-shaped, Z-shaped, arc shaped roughness etc. The solid model of the domain is generated in CAD software, removing unnecessary features (fillets, nuts, bolts, etc.) which may complicate meshing or grid generation process.

iii) Mesh generation. Meshing or grid generation is the most vital part of the CFD analysis. Meshing means dividing the flow domain into a number of sub domain which has a form of geometric primitives like hexahedron and tetrahedron in 3-Dand triangles and quadrilaterals in the 2-D physical domain. The sub domains are called as element or cells and collection of all elements or cells is called a mesh or grid. The presence of good meshing has a significant effect on the rate of convergence, solution accuracy and computation time. During meshing, appropriate boundary conditions at cells are specified. In an analysis of a solar air heater, many researchers used very fine non-uniform quadrilateral mesh to resolve the laminar sub-layer.

2.3.2. Solver:

After identifying the physics of the problem, governing algebraic equations is discretized using FDM, FEM or FVM and solved within each sub domains which are created during meshing in the solver. The commercial software available as a solver are ANSYS FLUENT, ANSYS CFX, OpenFOAM,GASPCFL3D, TYPHON, OVERFLOW etc. The CFD analysis of a solar air heater is carried out using ANSYS-FLUENT or ANSYS CFX tool by the researchers previously. The steps used by yadav et al.[12] in the analysis of a roughened solar air heater using ANSYS-FLUENT CFD tool are as follows–

1) Define material properties. In a solar air heater analysis, the incoming air is assigned a material property as fluid (air) and Absorber plate as solid (aluminum).

2) Selection of appropriate physical model. For selecting appropriate model for CFD analysis of roughened solar air heater, Yadav et al. [12] have carried out validation of various turbulence models such as Standard k- (model, Renormalization-group k- (model, realizable k- (model, Standard k- (model and Shear Stress Transport (SST) k- (model by comparing the Nusselt number predicted using these turbulence models with Dittus–Boelter [13] and blasius correlations [14] for smooth duct. The variation of the nusselt number with Reynolds number of different turbulence models is shown in Fig.2. They observed that the results obtained by Renormalization-group(RNG) k-ε model shows 72.58% absolute percentage deviation in predicted values and the values calculated from dittus–boelter correlations. The results generated from Realizable k- (model and Standard k- (model are under predict, whereas that obtained from Standard k- (model and SST k-model are over predicted. Since the results obtained by renormalization-group(RNG) k- (turbulence model are in good agreement with the Dittus–Boelter and Blasius correlation results, this model was selected for the CFD analysis of a solar air heater.

3) Prescribe operating parameters. The operating parameters used in the CFD analysis of a roughened solar air heater include uniform heat flux, Reynolds number(Re), Prandtl number(Pr),relative roughness pitch(P/e), relative roughness height(e/D) and a duct aspect ratio(W/H).The variable parameters such as relative roughness pitch(P/e), relative roughness height(e/D) and a duct aspect ratio(W/H) selected as per need of the analysis.

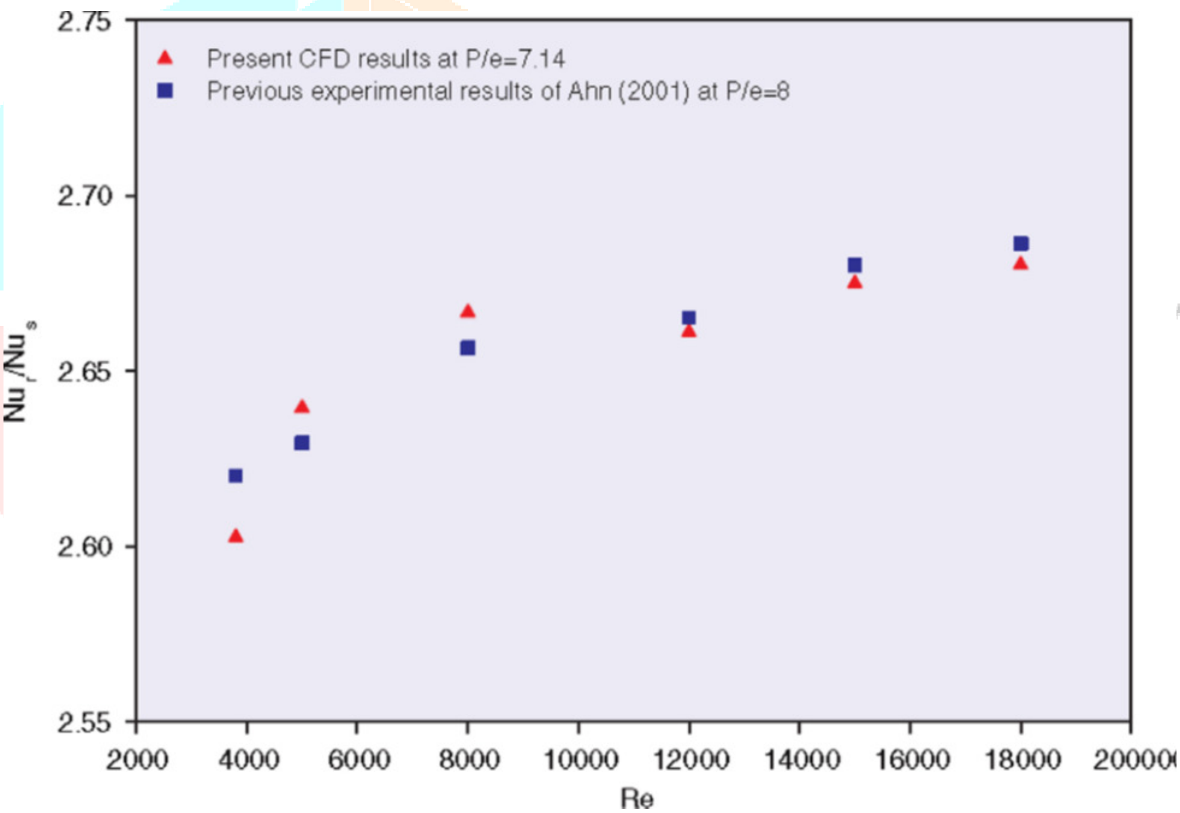


Fig.2. Comparison of CFD analysis with experimental results

4) Prescribe boundary conditions at all boundary zones: Physical conditions are required on the boundaries of the flow domain for analysis. The surface boundaries associated with a solar air heater are inlet section, outlet section, walls of solar air heater duct and heater plate. At the inlet of solar air heater, uniform velocity was assumed and a fixed pressure (1.013-105 Pa) boundary condition was applied at the exit section. The top wall of the test section is allotted with a constant heat flux boundary condition. The average intensity (heat flux) of solar radiation is assumed as1000W/m2 and was used in many previous CFD analysis of solar air heater. The adiabatic boundary condition was implemented over the remaining walls of the solar air heater duct. At the start of the analysis, the incoming air was maintained at 300K. Another property called turbulence intensity (I) is also applied at inlet, calculated using a relation, I=0.16(Re$^{)-1/8}$ reported in ref.

5) Prescribe solution method. Governing equations were solved using a segregated pressure based solver with a finite volume based technique in FLUENT solver. The Semi Implicit Method for Pressure-Linked Equations (SIMPLE) developed by Patankar [11] was used to couple velocity and pressure.

Table.1. Difference between Experimental and Computational approach:

Experimental	Computational
Actual physical system can be analyzed in the real world.	Physical system is simplified from actual physical system.
Experiments can be performed at limited number of points and time instants.	Here simulations are performed with high resolution in space and time.
The cost associated with experimental setup is generally higher.	Except for initial investment for commercial software, no investment is required.
Experiments can be performed for limited range of problems and operating conditions.	The approach can be extended for any range of problem virtually and for realistic operating conditions.
The modification in experimental setup is not possible once manufactured.	A large number of variation in domain can be possible here.
For data collection, large numbers of measuring instruments are required.	Various tools are available for calculations of flow properties.
Experiments are not limited by the complexity of the problem.	The computational approach is limited by availability of mathematical model especially for complex physical systems.
Experimental approach is slow, sequential and single purpose.	Computational approach is fast, parallel and multi-purpose.

2.3.3. Post-processing:

Post-processing is used to examine the desired flow properties. Post processing consists of visualization too land numerical reporting tool. Visualization tools extract properties in the form of contour plots, streamline plots, vector plots, line and shaded contour plots, 2D and 3D surface plots etc. and help to locate area of separation and shocks. Numerical reporting tools are used to calculate quantitative results in terms of finite values of velocity, pressure, temperature, heat transfer coefficient, surface and integrated quantities, flux balances etc. In CFD analysis of a roughened solar air heater using square sectioned transverse rib, carried out by Yadav et al. [17], the contour plots of velocity, pressure, turbulence kinetic energy and turbulence intensity are provided as post processing results as shown. In post processing, to analyze the thermal and hydraulic performance of a solar air heater following parameters are calculated, thermal performance of roughened solar air heater duct is measured in terms of the Nussselt number using, which uses heat transfer coefficient evaluated from CFD results.

2.3.4. Validation of CFD model: CFD analysis is validated by comparing the results of CFD analysis with available experimental data to make sure the results are in good agreement with reality.

III. Applications of CFD in various aspects of solar air heaters:

A lot of experimental and theoretical studies have been reported to evaluate performance of solar air heater. Kumar et al. [19] experimentally investigated heat transfer and friction characteristics of solar air heater by using discrete W-shaped roughness on one broad wall of solar air heater. The maximum enhancement of Nusselt number and friction factor as a result of providing artificial roughness was found to be 2.16 and 2.75 times that of smooth duct. Mittal et.al. [20] presented a comparison of effective efficiency of solar air heaters having different types of geometry of roughness elements on the absorber plate. The effective efficiency was computed by using the correlations for heat transfer and friction factor developed by various investigators within the investigated range of operating and system parameters. Prasad and Saini [21] developed various design curves for artificially roughened solar air heater that gave the optimal thermo hydraulic performance. Prasad and Saini [22] investigated the effect of relative roughness pitch (P/e) and relative roughness height (e/D) on the heat transfer coefficient and friction factor for fully developed turbulent flow in a solar air heater duct with small diameter protrusion wires on the absorber plate. It was found that for a given relative roughness pitch, both the Nusselt number and friction factor increased with increasing relative roughness height and for a given relative roughness height both the Nusselt number and friction factor decreased with increasing relative roughness pitch, but not indirect proportion. Aharwal et al. [23] carried out an experimental investigation of heat transfer and friction factor characteristics of a rectangular duct roughened with repeated square cross-section split rib with a gap, on one broad wall arranged at an inclination with respect to the flow direction. The maximum enhancement in Nusselt number and friction factor was observed to be 2.59 and 2.87 times of that of the smooth duct respectively. The thermo-hydraulic performance parameter was found to be maximum for the relative gap width of 1.0 and the relative gap position of 0.25. Muluwork [24] carried out an experimental analysis of a solar air heater having V-shaped staggered discrete ribs on the absorber plate and reported that maximum heat transfer enhancement occurred at an angle of attack of 601. Prasad and Mullick [25] utilized artificial roughness in a solar air heater duct in the form of small diameter wires to increase the heat transfer coefficient for relative roughness height and relative roughness pitch of 0.019 and12.7, respectively. Gupta [26] investigated the effect of relative roughness height, angle of attack and reynolds number on heat transfer and friction factor in rectangular duct having circular wire ribs on the absorber plate. The maximum enhancement of Nusselt number and friction factor as a result of providing artificial roughness was found to be1.8 and 2.7 times that of smooth duct. Verma and Prasad [27] investigated the effect of geometrical parameters of circular wire ribs on heat transfer and friction factor. It was observed that the value of heat transfer enhancement factor varies from 1.25 to 2.08 within the range of parameters investigated. The value of optimal thermo-hydraulic performance was found to be about 71% corresponding to roughness Reynolds number of 24.Karwa [28] carried out a comparative experimental study of augmented heat transfer and friction in a rectangular duct of a solar air heater with rectangular cross-section ribs arranged in V-continuous and V-discrete pattern. Vijay Bisht et.al [29] in their review paper concluded that V shape and protrusion roughness enhance performance of solar air heater.

IV. Discussion about turbulence models:

The literature survey reveals that the solar air heaters are thermo-hydraulically more efficient if system operate sat Reynolds numbers ranges from 3000 to19000. Reynolds Number inside the rectangular duct of solar air heater shows that the flow is turbulent. One of the great challenges in the design of a solar air heater using CFD approach is the selection of appropriate turbulence model. A turbulence model is a computational procedure to close the system of mean flow equations. Turbulence models allow the calculation of the mean flow without first calculating the full time-dependent flow field. Modern CFD programs offer a large range of methods and models to simulate turbulence. A number of software based on CFD codes have been developed, few of the mare: CFX, FLUENT, PHOENICS, FLOVENT and STAR-CD. Each software is usually supported by supplementary software for different applications, such as domain model preparation, mesh generation etc. Under the present study, commercial CFD code ANSYS FLUENTv12.1 is used. There are several turbulence model included in the commercial CFD software ANSYS FLUENT v12.1. The following turbulence models, available in ANSYS FLUENT v12.1, are tested: Standard k–e turbulence model, Realizable k–e turbulence model, Renormalization-group (RNG) k–e turbulence model, Standard k–o turbulence model, and Shear Stress Transport (SST) k–o turbulence model.

V. Selection of best turbulence model:

To achieve the accurate prediction of heat transfer and friction factor in a solar air heater, the predictive ability of five different turbulence models including : The Standard k–e turbulence model, the Realizable k–e turbulence model , the Renormalization-group (RNG) k–e turbulence model, the Standard k–o turbulence model, and the Shear Stress Transport(SST) k–o turbulence model, are investigated and compared with the available experimental data. The average Nusselt number is observed to increase with increase in Reynolds number for all turbulence models. It is observed that the results obtained by Renormalization-group (RNG) k–e model are in good agreement with the Dittus–Boelter

empirical correlation results. The average absolute percentage deviations between the values predicted by Standard k–w model and Dittus-Boelter empirical correlation results is found to be 3.58% for Nusselt number. Prediction by Standard k–w model shows more deviation with dittus–boelter empirical correlation results. Five different turbulence models, for steady state conditions, available in FLUENT, are tested and the RNG k–e model is proven as the most appropriate. The computed average Nusselt number and friction factor are in very good agreement with the corresponding experimental values.

VI. Conclusions:

On the basis of the review of the literature and CFD investigation of solar air heater, the conclusion can be summarized as follows:

1-The influences of the five different turbulence models such as: Standard k–e turbulence model, the Realizable k–e turbulence model, the Renormalization-group(RNG) k–e turbulence model, the Standard k–w turbulence model, and the Shear Stress Transport(SST) k–w turbulence model on the quality of the obtained results are tested. It appears from the performed calculations that the Renormalization-group k–e model yields the best results for two dimensional flows through a conventional solar air heater done by CFD analysis in the design of a solar air heater.

2-CFD- simulation results are found to be in good agreement with experimental results and with the standard theoretical approaches. Although there are some small discrepancies due to some experimental imperfectness matters, we still have a good confidence in the CFD simulation program that can be used in the future for more complex solar air heater problem.

3-In recent years CFD has been applied in the design of solar air heater. The quality of the solutions obtained from CFD simulations are largely within the acceptable range proving that CFD is an effective tool for predicting the behavior and performance of a solar air heater.

4-One of the great challenges in the design of a solar air heater using CFD approach is the selection of appropriate turbulence model. The decision about a suitable turbulence model chosen in a CFD computation is not easy. In summary, the purpose of this article is to illustrate the modern use of CFD in design of solar air heater and to use this to indicate the wide open future of CFD design. No matter how mature the techniques of CFD may become, the array of future and challenging applications of CFD is limitless. There is tremendous scope for future study of solar air heater with CFD approach. The information presented here will be beneficial for beginners in this area of research. Authors hope that this article has opened the horizons of CFD analysis of solar air heater to researchers.

REFERENCES:

[1] Foster R, Ghassemi M, Cota A. Solar energy : renewable energy and the environment. NewYork: CRC Press Taylor & Francis group;2010.
[2] Li X. Green energy: basic concepts and fundamentals.1sted.. NewYork: Springer; 2011.
[3] MaczulakA.Renewableenergy:sourcesandmethods.1sted..NewYork: info base Publishing;2011.
[4] Kalt schmitt M, Streicher W, Wiese A. Renewable energy: technology, economics and environment.1sted.. New York: Springer; 2007.
[5] Quaschning V. Understanding renewable energy systems .3rded..London: Earth scan; 2005.
[6] Twidell J, WeirT.Renewableenergy:sources.2nded..NewYork: Taylor& Francis; 2006.
[7] Sukhatme S P, Nayak J P.Solar energy.3rded..NewDelhi: Tata McGraw Hill; 2011.
[8] Date A W. Introduction to computational fluid dynamics.1sted..New York: Cambridge University Press; 2005.
[9] Chung TJ, editor.CambridgeUK:CambridgeUniversityPress;2002.
[10] Cebeci T, Kafyeke F, Shao JP, Laurendeau E. Computational fluid dynamics for engineers. 1sted..NewYork: Springer; 2005.
[11] Ferziger J H, Peric M.Computational method for fluid dynamics.3rded.. New York:Springer;2002.
[12] AndresonJrJD.Computational fluid dynamics—the basics with applications.1st ed..New York: Mcgraw-Hill;1995.
[13] Blazek J.Computational fluid dynamics—principles and applications.1sted..Oxford, UK:Elsevier;2001.
[14] Garg HP,Prakash J.Solar energy fundamentals and applications.1sted.. New Delhi:TataMcGraw-Hill;2000.
[15] Duffie J A, Beckman W A.Solar engineering of thermal processes.2nded..New York:Wiley;1980.
[16] PatankarSV.Numericalheattransferandfluidflow.1sted..USA:Hemi-
sphere PublishingCorporation;1980.
[17]Tannehill JC, Andeson DA, Pletcher RH. Computational fluid mechanics and heat transfer.2nded..London: Taylor &Francis;1997.
[18] Versteeg H K, Malalasekera W.An introduction to computational fluid dynamics. 2nded..England: Pearson Education Limited;2007.
[19] Kumar A, Bhagoria J L, Sarviya R M. Heat transfer and friction correlations for artificially roughened solar air heater duct with discrete W-shaped ribs. Energy Conversion and Management2009;50:2106–17.
[20] Mittal M K, Varun R P, Singal S K. Effective efficiency of solar air heaters having different types of roughness elements on absorber plate. Energy 2007;32:739–45.
[21]Prasad B N, Saini J S. Optimal thermo hydraulic performance of artificially roughened solar air heaters. SolarEnergy1991;47(2):91–6.
[22] Prasad B N, Saini J S. Effect of artificial roughness on heat transfer and friction factor in a solar air heater. Solar Energy1988;41(6):555–60.
[23] Aharwal KR, Gandhi BK, SainiJS. Experimental investigation on heat-transfer enhancement to a gap in an inclined continuous rib arrangement in a rectangular duct of solar air heater. Renew Energy2008;33:585–96.
[24]Muluwork KB.Investigations on fluid flow and heat transfer in roughened absorber solar heaters. PhD thesis. IIT Roorkee, India;2000.
[25]PrasadK, Mullick SC.Heat transfer characteristics of a solar air heater used for dryingpurposes.AppliedEnergy1983;13:83–93.
[26] Gupta D,SolankiSC,SainiJS.Heatandfluidflowinrectangularsolarair
heaterductshavingtransverseribroughnessonabsorberplates.Solar
Energy 1993;51(1):31–7.
[27]Verma SK, Prasad BN.Investigation for the optimal thermo hydraulic performance of artificially roughened solar air heaters.RenewalEnergy2000;20(1):19–36.
[28] KarwaR. Experimental studies of augmented heat transfer and friction in asymmetrically heated rectangular ducts with ribs on heated wall in transverse, inclined, v-continuous and v discrete pattern. International communications in Heat and MassTransfer2003;30(2):241–50.
[29] Vijay Singh Bisht, Anil Kumar Patil, Anirudh Gupta, Review and performance evaluation of roughened solar air heaters, Renewable and Sustainable Energy Reviews,Volume 81, Part 1, 2018