SHIVALIK COLLEGE OF ENGINEERING DEHRADUN

Department of Mechanical Engineering

"SIMULATION OF SOLAR AIR HEATER"

A Report Submitted For Major Project

In
BACHELOR OF TECHNOLOGY

In
Mechanical Engineering

By

ASHISH DHASMANA (130410101004)
SANTOSH JOSHI (640410104054)
SUMIT PAL (130410104065)

Under the Supervision of
A.P KULDEEP RAWAT
Mechanical Engineering Department

Shivalik College of Engineering, Dehradun (Uttarakhand)

May, 2017

CERTIFICATE

This is to certified that **ASHISH DHASMANA, SANTOSH JOSHI, SUMIT PAL** has carried out the project work presented in this report entitled **"SIMULATION OF SOLAR AIR HEATER"** for the award of **Bachelor of Technology** from Shivalik College of Engineering, Dehradun affiliated to the Uttarakhand Technical University, Dehradun under my supervision. The report embodies results of original work, and studies are carried out by the students himself and the contents of the report do not form the basis for the award of any other degree to the candidate or to anybody else from this or any other University/Institution.

(MR. KULDEEP RAWAT)
Assistant Professor
Shivalik College of Engineering,
Dehradun Uttarakhand

Date: 12 May 2017

Head
Department of Mechanical Engg.
SCE, Dehradun UK

12/5/17

ABSTRACT

Given that the future of our planet is intricately entwined with the future choices of energy, effective exploitation of non-conventional energy sources is becoming increasingly essential for modern world as fossil fuels are hazardous to environment and cannot sustain supply for long time as they are not renewable. Moreover, demand of energy is increasing rapidly. In this scenario, solar energy is being seen as potential viable resource for ever increasing hunger of the energy for the development of nation and by and large globe. In this seminar work, effort has been made to demonstrate this reality with proof of statistics from reliable sources. Furthermore numerous new designs of Solar Air Heater are emerging in various aspects, in different number of roughness, in different cost. Extensive review of research done in this field in recent past is covered with their design characteristics and their suitability for specific conditions and applications with respect to their merits and demerits.

ACKNOWLEDGEMENTS

One cannot hope to acknowledge fully ones depth in affairs of learning. Nevertheless we wish to convey my appreciation to certain teachers, colleagues and collaborators in this Endeavour.

It is great pleasure to express my profound sense of gratitude and reverence to my project supervisor MR. **Kuldeep Rawat, Assistant. Professor, Department of Mechanical Engineering, Shivalik College of Engineering, Dehradun, Uttarakhand.** He was always a source of encouragement and inspiration, and constantly guided me for accomplishment of this task with meticulous care. The work would have been very difficult without his deep insight and knowledge about the subject matter with a combination, with the appropriate methodology of research. We owe to him the most and consider myself fortune, to have had the opportunity to accomplish the work under his guidance.

We are extremely grateful and highly obliged to **Dr. Kuldeep Panwar, Head of Mechanical Engineering Department, Shivalik College of Engineering, Dehradun** with whom We have a special bond of kinship. His sincere, affable attitude and helping nature will always remain a pleasant part of my memory.

My tender sentiments are for my friends who directly or indirectly involved in various stages of the present research work. I am a person among the millions who have a thinking that their parents are beyond comparison in the whole world. They have strongly influenced my life since they remained as a guiding star, whenever, I lost the direction.

ASHISH DHASMANA **SANTOSH JOSHI** **SUMIT PAL**

TABLE OF CONTENTS

CHAPTER 1

INTRODUCTION

.1 INTRODUCTION

he present work is a numerical study on heat transfer and friction characteristics in solar air heater luct with obstacles the motivation for this study is come from the fact that the formation of eddies nfluence the performance of the solar air heater. Numerical simulation is conducted using k-ε ırbulence model different arrangement of obstacles are investigated, this work leads to the nprovement of heat transfer and thermo hydraulic performance of SAHs. Angle of attack (α) and bstacles arrangement significantly affect the Nussel number and friction factor. Solar air heater ecome more popular and used in various application and also reduction in fuel consumption and osts of installation and operation. The CFD results for the base model are validated against xperimental results and are found to have good arrangement.

nprovement of thermal performance of a solar air heater can be obtained by enhancing the rate f heat transfer. In comparison to conventional type. Number of studies has been report ed in the terature on the performance of matrix air heater with different material packing and elaborated arious parameters such as design; heat transfer enhancement. The dwindling in nature of fossil eserves and the increasing demand for energy in various forms, urged the search for alter- native esources. The renewable energy resources received pro- minent attention in recent years, as it is roven to be a viable alternative of conventional energy sources. Solar energy which is available or free of cost in almost all part of the world is a non- polluting reservoir of fuel. Solar air heater s basically a flat plate collector used for generating hot air for applications such as crop drying, ndustrial process heating, space heating, timber seasoning etc.,

The energy and exergy analysis of solar air heater having absorber plate corrugated in the shape of trapezoid along with sensible heat storage material is carried out. Gravel is used as a sensible heat storage material and placed below the absorber plate. Thermal performance analysis of solar air heater with and without sensible heat storage materials is carried out and compared with flat plate solar air heater. The parameters such as solar radiation intensity, temperatures at various

positions in solar air heater, and daily thermal and exergy efficiency are calculated. The average daily thermal efficiency of flat plate solar air heater
and that having trapezoidal corrugation are 8.5% and 12.2% respectively, while trapezoidal corrugated absorber with sensible heat storage material shows thermal efficiency of 36.6%.The maximum exergy efficiency obtained having trapezoidal corrugated absorber with sensible heat storage material is 12.56 %.
Given the geographical location, India has enormous scope to generate solar energy. This accounts to the fact that India is a tropical country and hence receives solar radiation throughout the year, which may be as much as 3,000 hours of sunshine equivalent to more than 5,000 trillion kWh. Approximately, 4-7 kWh of solar radiation per square meters is received by all parts of India, that being equivalent to 2,300-3,200 sunshine hours per year.

The efficiency of corrugated collectors are high and turbulence enhances the heat transfer as a result efficiency is more. Esen [4] carried out the experimental study for a double flow solar air heater placed different obstacles on the absorber plate and compared with a solar air heater without obstacles on the absorber plate. The authors measured inlet and outlet temperatures and absorber plate temperatures and found that exergy efficiency of flat plate collector is less compared to collector with obstacles which is mainly due to irreversibility's are more in flat plate collector. The obstacle creates a good turbulence and reduces the dead zones so efficiency of collector with obstacles is more compared to the collector without obstacles. Many authors [5-8] carried out their analysis with different shapes of obstacles placed on the absorber plate and they have studied the energy and exergy efficiencies. Many solar air heaters are reported in the literature and the disadvantage of these collectors that, they can't store solar energy after sunshine hours. For continuous space heating the energy required for longer hours this can be done.
A model of solar air heater with selective absorber coating with longitudinal fins at the bottom of absorber has been prepared to produce hot gases by consuming solar energy in day time. It is kept in position in such a way that air will flow due to forced convection and the entire setup placed at an angle of 15□ to 17□ inclination as per the considered global position (20.2961° N, 85.8245° E). Air flow rate is increased by applying a DC suction fan at outlet which sucks the air to flow over extended surfaces which are present at bottom inlet channel. Special selective coating applied over the entire Aluminum absorber plate to increase the heat absorbing capacity. Average ambient,

inlet and plate temperatures are recorded daily along with solar flux. It is found that the instantaneous efficiency and air outlet temperature increased up to 69% and 94°C respectively. Theoretical analysis has been made as well for the present set up.

The difference between the theoretical and experimental values lies within 1.23-9.75%. Solar air heaters (SAH) have vast applications both in industrial and agricultural sectors. So, huge research is going on to enhance the performance of solar air heaters by putting number of effort in various enhancement techniques like increasing the heat transfer rate, proper tracking geometry and better heat absorbing medium of high Absorptivity and putting various thermal energy storage mediums. Dong Ho et al. studied wire mesh packed solar air heaters where due to high turbulence intensity and enlarged heat transfer area, the heat transfer efficiency is enhanced. The technical feasibility of the recycle-effect on heat transfer rate of air through extended surface up to small height has been confirmed by different researchers worldwide. Application of the concept of double-pass in the design of a double-pass solar air heater with fins attached has been mentioned along with technical and economical feasibility which provides better performance Its proven that, by using pack bed SAH with various thermal storage devices can increase the performance from 45% to 53. Nowzari et al. involved in optimization and experimentation of SAH where they found that double pass solar air heater with quarter perforated cover with 3cm hole to hole distance with 0.032kg/s mass flow provide best result in its class. Computational Fluid Dynamics (CFD) based approach has been implemented extensively in design and performance analysis of various solar air heaters.

From literatures, we found that most of researchers analyzed the performance of SAH having air entry at top channel and released at the bottom. Present approach is an exercise to obtain the performances of different kind of SAH with reversed air flow. In this present setup, air initially flows between heated fins in the lower channel generating rise in turbulence which helps in absorbing heat capacity. During the flow of air in top channel carrying hot air, further heat transfer occurs to the incoming air from absorber plate of top channel. The purposes of present experimental approach is to check the overall output performances over existing ones as the flow direction is just reversed in comparison with the conventional.

Many studies have studied the influence of various rib design parameters on fluid flow and heat transfer in solar air heaters. The most common design for solar air heaters is regularly spaced transverse continuous ribs, which have been shown to provide considerable heat transfer enhancement over a smooth channel. The transverse rib results in formation of two vortices just downstream and upstream, which produces separated flow. Subsequent reattachment of the boundary layer provides significant heat transfer enhancement. Prasad and Saini showed that the maximum increase in Nusselt number for a solar air heater roughened with transverse wire ribs was 2.38 times, compared to a smooth case. The heat transfer coefficient will reach a local peak at the reattachment point and decrease after this point, due to boundary layer redevelopment, until encountering the successive rib. Larger rib pitch produces a larger area of low heat transfer coefficient downstream of the reattachment point, too small rib pitch will mean the separated flow is unable to reattach before arriving at the next rib. Hence, for a fixed rib height, there is an optimum rib pitch for maximum heat transfer enhancement. Prasad and Saini showed the optimum rib pitch is approximately 8–10 times the rib height. The vortex generated by a transverse rib is essentially stagnant since the fluid in the vortex always remains within the rib vicinity, which increases vortex fluid temperature and the wall temperature near the vortex, and hence lowers the local heat transfer coefficient in that region. However, by angling the rib, the fluid in the vortex can leave the rib vicinity. For inclined ribs, fluid flows into the inter-rib region from the rib leading end and exits near the rib trailing end, subsequently joining the main stream to form a spanwise rotating secondary flow. This secondary flow increases the heat transfer coefficient at the rib leading end region producing a large span wise variation of local heat transfer coefficient. Hence, besides rib pitch and height, the rib attack angle is another important parameter influencing heat transfer enhancement. by incorporating thermal energy storage materials. In the present work, the authors carried out experiments with and without sensible heat storage material (Gravels) with the solar air heater.

1.2 Literature Survey

Performance analysis of double pass solar air heater with bottom extended surface by rudra nandan pramanika

A model of solar air heater with selective absorber coating with longitudinal fins at the bottom of absorber has been prepared to produce hot gases by consuming solar energy in day time. It is kept in position in such a way that air will flow due to forced convection and the entire setup placed at an angle of 15to 17inclination as per the considered global position (20.2961° N, 85.8245° E). Air flow rate is increased by applying a DC suction fan at outlet which sucks the air to flow over extended surfaces which are present at bottom inlet channel. Special selective coating applied over the entire Aluminum absorber plate to increase the heat absorbing capacity. Average ambient, inlet and plate temperatures are recorded daily along with solar flux. It is found that the instantaneous efficiency and air outlet temperature increased up to 69% and 94□C respectively. Theoretical analysis has been made as well for the present set up. The difference between the theoretical and experimental values lies within 1.23-9.75%.

Numerical study on thermal hydraulic performance improvement in solar air heater duct with semi ellipse shaped obstacles by tabish alam

This paper presents numerical study on heat transfer and friction characteristics in rectangular solar air heater duct with semi elliptical shape obstacles. 3-D simulations have been conducted using Renormalization-group k-ε turbulence model. Obstacles are placed on the absorber plate in V-down shape at different angle of attack (a), ranging from 30_ to 90_. Two different arrangements of obstacles namely; inline and staggered arrangements have been investigated. Four different values of Reynolds number, ranging from 6000 to 18,000 have been considered to determine the values of Nusselt number and friction factor. Angle of attack (a) and obstacles arrangement significantly affect the Nusselt number and friction factor. In staggered arrangement, maximum enhancement in Nusselt number and friction factor have been observed 2.05 and 6.93, respectively, at an angle of attack (a) of 75_ and corresponding enhancement in case of inline arrangement are found as 1.73 and 6.12, respectively. The maximum enhancement in Nusselt number at 75_ angle of attack is due to combined effect of high turbulence and lateral movement of air flow. Thermo hydraulic performance has also been determined which shows staggered arrangement are more superior than inline arrangement for all values of angle of attack (a) investigated in present study.

A critical review on artificial roughness provided in rectangular solar air heater duct by tabish alam.

Applications of artificial roughness on the underside of absorber plate in solar air heater duct have been widely used to improve heat transfer with moderate increase of friction factor. The design of the roughness shape and arrangement is most important to optimize the roughened surfaces. The roughness parameters and ribs arrangement are responsible to alter the flow structure and heat transfer mechanisms are mainly governed by flow structure. The critical reviews on various artificial roughness elements available in literature have been conducted and the effects of the roughness patterns are discussed. The Nusselt number and friction factor correlations for various roughness elements have been summarized. A comparison study of thermo hydraulic performance of different roughness elements has also been reported to understand the results of applications of artificial roughness.

Heat transfer and friction factor correlations for a solar air heater duct roughened artificially with broken arc ribs by v.s. Hans,

In present experimental investigation, solar air heater duct with aspect ratio 12 roughened with broken arc rib has been investigated. The broken arc was formed by creating symmetrical gap in continuous arc. To investigate the influence of roughness parameters of broken arc rib on Nusselt number as well as on friction factor, thirty seven broken arc rib roughened plates having relative roughness pitch (P/e), relative gap width (g/e), relative gap position (d/w), relative roughness height (e/Dh) and arc angle (α) varying from 4-12, 0.5-2.5, 0.2-0.8, 0.022-0.043 and 15°-75° respectively, have been investigated for Reynolds number range of 2000-16000. Keeping similar flow conditions, results of broken arc rib roughened duct have been compared with smooth and continuous arc rib roughened ducts. The maximum increase in Nusselt number and friction factor over that of continuous arc rib roughened duct was 1.19 and 1.14 times respectively. The corresponding values over that of smooth duct were 2.63 and 2.44 times respectively. Experimental results of heat transfer and friction in flow have also been correlated in terms of flow and roughness geometry parameters.

Solar air heaters: design configurations, improvement methods and Applications – a detailed review by *a.e. kabeela*

This article aims to present a review of the literature dealing with improvement methods, design configurations and applications of different types of solar air heaters (SAHs). Different investigations have been made on SAHs either experimental or theoretical in order to improve their performance. Different modifications on absorber plate have been made. One of the important improving theories is using fins to increase the heat transfer area. Various types of fins achieved enhancement in the performance such as longitudinal fins, corrugated fins and fins attached with baffles. The recycling process also enhanced the performance of SAHs. Also it is found that to improve both the heat transfer and thermo-hydraulic performance of SAH, artificially roughened absorbers are used. Using artificial roughness, the results showed good enhancement of both Nusselt number and friction coefficient for a wide range of Reynolds number. Also in the view of using heat storage materials, especially phase change materials (PCMs), various types of packing bed and selective coated absorbers give good improvement of the efficiency of SAHs. As a result of these improvements, SAHs became more popular in more applications, with a good reduction in fuel consumption and costs of installation and operation.

Experimental investigation on performance of a double pass artificial roughened solar air heater duct having roughness elements of the combination of discrete multi V shaped and staggered ribs by Ravi Kant Ravi

Double pass solar air heater (DPSAH) provided with roughness on each side of the absorbing surface is considered as a significant and interesting design advancement that has been used to enhance the performance of the collector. In this paper an experimental analysis has been conducted to study the effect of roughness parameters on thermo hydraulic performance of double pass duct having discrete multi V shaped and staggered rib. The study has involved the values of Reynolds number (Re) from 2000 to 20000 and relative staggered rib size (r/e) from 1 to 2.5. Other parameters like relative staggered rib position (P0/P) of 0.2, angle of attack (a) of 60, relative gap distance (Gd/Lv) of 0.70, relative pitch ratio (p/ e) of 10, relative roughness height (e/D) of 0.043 and relative gap width (g/e) of 1.0 are kept constant. Based on the study, heat transfer and pressure drop in single and double pass mode have been estimated at range of ribs and performance parameters and results are compared with smooth ducts under same operating conditions. It has been found that the roughness geometry used on each side of the plate in double pass mode enhances both frictional losses as well as heat dissipation rate.

Matrix solar air heaters – A review by K. Rajarajeswari

Improvement of thermal performance of a solar air heater can be obtained by enhancing the rate of heat transfer. Thermal efficiency of matrix solar air heater is higher in comparison to conventional type. Number of studies has been reported in the literature on the performance of matrix air heater with different material packing and elaborated various parameters such as design; heat transfer enhancement, flow phenomenon and pressure drop in the duct. Majority of the studies revealed an increase in the thermal efficiency for matrix collect or as compared to conventional plane solar air heater. This review paper presents an extensive study of the research carried out on matrix solar air heater. Based on literaturereview, it is concluded that the solar air heaters performed well when packed with porous medium and this is due to the geometrical parameters of porous material. In addition double pass porous bed solar air heaters performed better than single pass.Various types of matrixmaterials used in the literature and correlations developed for heat transfer and friction factor by different researchers have been presented. Much attention has been devoted in this paper to portrait the development of various types of solar air heaters over the years.The merits and demerits of different models evolved by many researchers have been critically analyzed.

1.3 OBJECTIVES

The main objective of this study is linked to the idea of being able to model the temperature fluctuations that occur in the air heater. These temperatures are initiated by wall temperatures as well as the heat flux. Therefore a basic understanding of the fluid dynamics as well as material conditions in the form of heat transfer is essential in this work. The main objective of report is to describe and analyze the knowledge about operational flexibility in air heater, based on mathematical / numerical methods. More specifically, the report aims to:

- Staggered multiple V-shaped ribs were proposed for heat transfer enhancement.
- Characteristics of the flow structure and heat transfer were derived and analyzed.
- Staggered V-shaped ribs performed better than the corresponding inline arrangement.
- The optimum stagger distance for maximum heat transfer enhancement was derived.

1.4 SCOPES

- Computational Fluid Dynamics (CFD) is used to compute the flow characteristic of the rifled tube.
- Only single phase flow is used to determine the flow characteristic in the smooth tube and rifled tube.
- Water is used as working fluid.
- Smooth tube and rifled tube are assumed to be placed horizontally in this research.

CHAPTER 2

GOVERNING EQUATIONS AND NUMERICAL SIMULATION

;OVERNING EQUATIONS AND NUMERICAL SIMULATION

.1 GOVERNING EQUATION

he behaviour of the flow is generally governed by the fundamental principles of the classical ıechanics expressing the conservation of mass and momentum. Here the considered steady, ıcompressible, turbulent flow is modeled by the momentum and continuity equations. The ɔntinuity and the momentum equations are as follows.

.1.1 CONTINUITY EQUATION

'ontinuity Equation also called conservation of mass. Consider fluid moves from point 1 to point . The overall mass balance is Input – output = accumulation. Assuming that there is no storage ıe Mass input = mass output. However, as long as the flow is steady (time-invariant), within this ıbe, since, mass cannot be created or destroyed then the above equation. According to continuity quation, the amount of fluid entering in certain volume leaves that volume or remains there and ccording to momentum equation tells about the balance of the momentum. The momentum quations are sometimes also referred as Navier-Stokes (NS) equation. They are most commonly ısed mathematical equations to describe flow. The simulation is done based on the NS equations ınd then K-Epsilon model. Continuity equation can be expressed as:

$$\frac{\delta(\rho\bar{u})}{\delta x} + \frac{1}{r}\frac{\delta(\rho r\bar{v})}{\delta r} = 0 \qquad (3.1)$$

2.1.2 MOMENTUM EQUATION

Axial component (z-component)

$$\rho \overline{v}\left[\frac{\delta \overline{u}}{\delta r} + \overline{u}\frac{\delta \overline{u}}{\delta x}\right] = \frac{\delta \overline{p}}{\delta x} + \frac{\delta}{\delta x}\left(\mu_{eff}\frac{\delta \overline{u}}{\delta x}\right) + \frac{1}{r}\frac{\delta}{\delta r}\left(\mu_{eff}\frac{\delta \overline{u}}{\delta r}\right) + \frac{\delta}{\delta x}\left(\mu_{eff}\frac{\delta \overline{u}}{\delta x}\right) + \frac{1}{r}\frac{\delta}{\delta r}\left(\mu_{eff}\frac{\delta \overline{u}}{\delta r}\right)$$

(3.2)

Radial component (r-component)

$$\rho\left[\overline{v}\frac{\delta \overline{r}}{\delta r} + \overline{u}\frac{\delta \overline{v}}{\delta x}\right] = -\frac{\delta p}{\delta z} + \frac{\delta}{\delta x}\left(\mu_{eff}\frac{\delta \overline{v}}{\delta x}\right) + \frac{1}{r}\frac{\delta}{\delta r}\left(r\mu_{eff}\frac{\delta \overline{r}}{\delta r}\right) + \frac{\delta}{\delta x}\left(\mu_{eff}\frac{\delta \overline{u}}{\delta r}\right) + \frac{1}{r}\frac{\delta}{\delta r}\left(r\mu_{eff}\frac{\delta \overline{v}}{\delta r}\right) - 2\mu_{eff}\frac{\overline{v}}{r^{2}} + \rho\frac{\overline{w}^{2}}{r}$$

(3.3)

Tangential Component (Θ- component)

$$\rho\left[\overline{v}\frac{\delta \phi}{\delta r} + \overline{u}\frac{\delta \phi}{\delta x}\right] = \frac{\delta}{\delta z}\left[\mu_{eff}\frac{\delta \phi}{\delta x}\right] + \frac{1}{r}\frac{\delta}{\delta r}\left[r\mu_{eff}\frac{\delta \phi}{\delta r}\right] - \frac{2}{r}\frac{\delta}{\delta r}\left[\mu_{eff}\phi\right]$$

(3.4)

Here $\overline{u}$, $\overline{v}$ and $\overline{w}$ are the mean velocity components along z, r and θ directions respectively and the variable $\phi = r\overline{w}$.

The total effective viscosity of the flow is given by,

$$\mu_{eff} = \mu_{l} + \mu_{t}$$

(3.5)

Here μ_l and μ_t stand for molecular or laminar viscosity and eddy or turbulent viscosity respectively. The molecular or the laminar viscosity is the fluid property and the eddy viscosity or the turbulent viscosity is the flow property. By using dimensional analysis, the eddy viscosity μ_t can be expressed as,

$$\mu_t = \rho V_t l \tag{3.6}$$

ere V_t , is the turbulent velocity scale and l is the turbulent length scale. It was postulated by andtl and Kolmogorov and later adopted in the standard k-ε model that

$$l = \frac{K^{3/2}}{\varepsilon} \tag{3.7}$$

$$V_t \sim \sqrt{k} \tag{3.8}$$

From the equation (3.6) the eddy viscosity is obtained and it is given by

$$\mu_t = \frac{\rho c \mu k^2}{\varepsilon} \tag{3.9}$$

Themodeling constant, c_μ in the eddy viscosity formulation, as shown in equation (3.10), empirically tuned for the simple shear layer. Meanwhile, there is no mechanism in the k-ε model nich can either amplify the turbulent intensity or eddy viscosity in the presence of concave or nvex curvature. Therefore, the expression for eddy viscosity in the standard k-ε model is nsidered to be inadequate to account for the streamline curvature effect. It is evident that odifications to the standard k-ε model are necessary to include the curvature effects. Therefore e constant c_μ is considered. The constant, c_μ is given by

$$C_\mu = \frac{-k_1 k_2}{\left[1+8k_1{}^2 \frac{k^2}{\varepsilon^2}\left(\frac{\delta U_S}{\delta n}+\frac{U_S}{R_C}\right)\frac{U_S}{R_C}\right]} \tag{3.10}$$

ι the equation (3.10), $U_s = \sqrt{\bar{u}^2 + \bar{v}^2}$ and R_c is the radius of curvature of the streamline oncerned (Ψ constant).

2.1.3 THE TURBULENT MODELING

2.1.3.1 KAPPA-EPSILON MODEL

The K-epsilon model is most commonly used to describe the behavior of turbulent flows. It was proposed by A.N Kolmogrov in 1942, then modified by Harlow and Nakayama and produced K-Epsilon model for turbulence. The Transport Equations for K-Epsilon model are for k, Realizable k-epsilon model and RNG k-epsilon model are some other variants of K-epsilon model. K-epsilon model has solution in some special cases. K-epsilon model is only useful in regions with turbulent, high Reynolds number flow.

K – Equation

$$\rho[\bar{u}\frac{\partial k}{\partial x} + \bar{v}\frac{\partial k}{\partial r}] = \frac{\partial}{\partial x}[(\mu_l + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x}] + \frac{1}{r}\frac{\partial}{\partial r}[r(\mu_l + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial r}] + \rho g - \rho\varepsilon \qquad (3.11)$$

Where, G is the production term and is given by

$$G = \mu_t[2\{(\frac{\partial \bar{v}}{\partial r})^2 + (\frac{\partial \bar{u}}{\partial x})^2 + (\frac{\bar{v}}{r})^2\} + (\frac{\partial \bar{u}}{\partial r} + \frac{\partial \bar{v}}{\partial x})^2] \qquad (3.12)$$

The production term represents the transfer of kinetic energy from the mean flow to the turbulent motion through the interaction between the turbulent fluctuations and the mean flow velocity gradients.

ε - Equation

$$\rho[\bar{u}\frac{\partial \varepsilon}{\partial x} + \bar{v}\frac{\partial \varepsilon}{\partial r}] = \frac{\partial}{\partial x}[(\mu_l + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial x}] + \frac{1}{r}\frac{\partial}{\partial r}(r\mu_l + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial r}] + C_{S1}G\frac{\varepsilon}{k} - C_{s2}\frac{\varepsilon^2}{k} \qquad (3.13)$$

Here C_{s1}, C_{s2}, σ_k and σ_ε are the empirical turbulent constant. The values are considered according to the Launder, *et. al.* (1974).

2.2 PERFORMANCE PARAMETERS

This section describes how heat transfer and pressure drop are characterized. Included are dimensionless groups, equations for heat transfer and efficiency calculations, and equations for making pressure drop calculations. Following the descriptions of the performance parameters, the values as read from the graphs in the research done by Wang, et. al. (1996) for friction factor *f* and Colburn *j*-factor vs. Reynolds number.

2.2.1 DIMENSIONLESS GROUPS

Accurate characterisation of the flow friction and heat transfer is very important in rating and sizing heat exchangers. Dimensional groups are used for this characterisation: heat transfer defined with the Colburn factor *j* and pressure drop defined by friction factor *f*. Below is a summary of the dimensionless groups used in this project, the equation to calculate it, and a brief description.

2.2.1.1 REYNOLDS NUMBER

The Reynold's number *Re* represents the ratio of flow inertial forces to viscous forces. The Reynold's number characteristic dimension for this study is the tube collar diameter D_c.

$$\mathrm{Re} = \frac{\rho.U_i.D_h}{\mu} \tag{3.14}$$

2.2.1.2 FRICTION FACTOR f

The Fanning friction factor is the ratio of wall shear stress to the flow kinetic energy. It is related to pressure drop tube heat exchangers as:

$$f = \frac{\Delta p.\dfrac{D_h}{L_t}}{\dfrac{1}{2}.\rho.U_i^2} \tag{3.15}$$

Where L_t is length of duct,(m); ρ is air density (Kg/m^3); Δp is pressure drop (Pa).

2.2.1.3 COLBURN j-FACTOR

The Colburn *j*-factor is the ratio of convection heat transfer (per unit duct surface area) to the amount virtually transferable (per unit of cross-sectional flow area).

$$j = \frac{Nu}{Re_{D_c} \cdot Pr^{1/3}} \quad (3.16)$$

2.2.1.4 NUSSELT NUMBER Nu

The Nusselt number is the ratio of convective conductance h to molecular thermal conductance k/D_h.

$$Nu = \frac{h}{k / D_h} \quad (3.17)$$

The Nusselt number is based on the hydraulic diameter D_h. There are different calculations for this available in the literature. The hydraulic diameter in this study is the ratio of the 4 times the minimum free flow air-side area to the wetted perimeter (ratio of air-side surface area to heat exchanger length), and is given by the following expression (Fornasieri and Mattarolo, 1991).

$$D_h = \frac{4(F_p - t)(P_t - D_c)P_l}{2(P_l P_t - \pi D_c^2 / 4) + \pi D_c (F_p - t)} \quad (3.18)$$

2.2.1.5 PRANDTL NUMBER Pr

The Prandtl number Pr is the ratio of a fluid's momentum diffusivity to thermal diffusivity.

$$\mathrm{Pr} = \frac{\nu}{\alpha} = \frac{\mu C_p}{k} \quad (3.19)$$

2.2.1.6 PRESSURE DROP

The pressure drop determines the amount of pumping power needed to run a heat exchanger. It is therefore important to characterize the pressure drop for design. This section describes how the pressure drop relates to the pumping power, followed by a description of what causes the pressure drop and finally the pressure drop equations for tube-and-fin heat exchangers are presented as:

$$\Delta p = \frac{L}{D}\frac{\rho V^2}{2} \tag{3.20}$$

Pumping power P is often seen as an important design constraint because the pressure drop in a heat exchanger (along with associated pressure drops in the inlet/outlet headers, nozzles, ducts, etc.) is proportional to the amount of fluid pumping power needed for the heat exchanger to function, as given by the following expression:

Pumping power $P = \frac{\dot{m}\Delta p}{\rho}$ (3.21)

The overall pressure drop consists of two parts: (1) the pressure drop in the heat exchanger core, and (2) the pressure drop from associated devices the fluid flows through before and after the heat exchanger core (i.e. inlet/outlet manifolds, nozzles, valves, fittings, ducts, etc.)

CHAPTER 3

MATERIALS AND METHODS

3. MATERIALS AND METHODS

3.1 NUMERICAL SOLUTION PROCERSS

Due to the advances in computational hardware and available numerical methods, CFD is a powerful tool for the prediction of the fluid motion in various situations, thus, enabling a proper design. CFD is a sophisticated way to analyze not only for fluid flow behavior but also the processes of heat and mass transfer. Advances in physical models, numerical analysis and computational power enable simulation of the heat transfer characteristics in three-dimensional circumstances. A three dimensional approximation of a turbulent flow is chosen to explore since the three-dimensional approach is considerably greater than two dimensional and moreover, a turbulent flow is fundamentally three-dimensional. Owing to extremely long computation times, detailed studies on the fluid flow through pipes in three-dimensional flow are very uncommon. Hence, the simulation of the three-dimensional flow field under complex geometrical conditions is seemingly intricate and challenging task.

The available computational fluid dynamics software package FLUENT is used to determine the related problems. FLUENT uses a finite volume method and requires from the user to supply the grid system, physical properties and the boundary conditions. When planning to simulate a problem, basic computation model considerations such as boundary conditions, the size of computational domain, grid topology, two dimensions or three-dimension model, are necessary. For example, appropriate choice of the grid type can save the set up time and computational expense. Moreover, a careful consideration for the selection of physical models and determination of the solution procedure will produce more efficient results. Dependent on the problem, the geometry can be created and meshed with a careful consideration on the size of the computational domain, and shape, density and smoothness of cells. Once a grid has been fed into FLUENT, check the grids and executes the solution after setting models, boundary conditions, and material properties. FLUENT provides the function for post processing the results and if necessary refined the grids is available and solve again as the above procedure. As described in the objective, the purpose of this study is to investigate numerically the effect of mal distribution in the tubular heat exchanger.

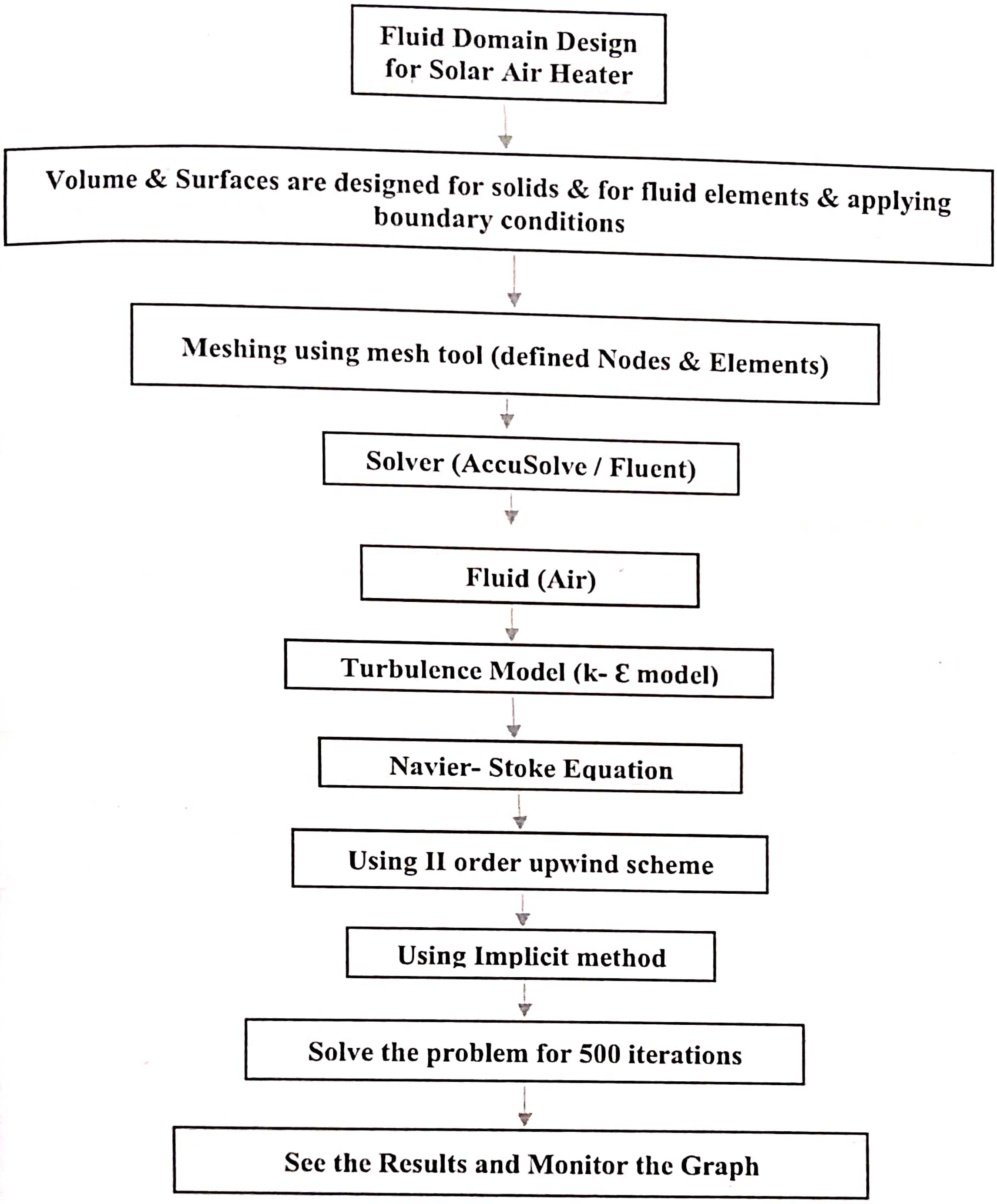
The whole analysis is carried out with the help of software "ANSYS Fluent 14.5". ANSYS Fluent 14.5 is computational fluid dynamics (CFD) software package to stimulate fluid flow problems. The two dimensional computational domain modelled using hex mesh for 2-D models. The complete domain of 2-D tube in all cases have fix element size. Grid independence test was performed to check the validity of the quality of the mesh on the solution. Further refinement did not change the result by more than 0.9% which is taken as the appropriate mesh quality for computation.

3.1.1 METHODOLOGY

The methodology of the present study can be divided into four stages of process flow which are geometry modeling, pre-processing, processing and post-processing. Various steps in adopted methods are:

- Mathematical modeling of the system considered in present study.
- Developed the model in SOLIDWORKS.
- Validation of present work with previous research.
- Calculation of heat transfer parameters.
- Run program to obtain the plots with different boiler tube parameters.
- Plotting & analysis of obtained plots.
- Optimization of the system.

FLOW CHART SHOWING THE APPROACH FOR SOLVING THE PROBLEM



3.2 SIMULATION OF FLOW THROUGH BOILER TUBE

For the CFD analysis of solar air heater first the fluid domain was designed using Solid works. The boundary conditions applied to the channel, the assumptions made, the equations used, the results obtained after calculation and then the results were analysed.

3.2.1.1 GEOMETRICAL DESCRIPTION

Geometry of solar air heater was designed using solid works 2013. To see the fluid behavior inside the channel. Corrugated shape was given to see the changes in heater. These geometrical models of plates are used for studying the effects of the variation of Reynolds number on the performance of heater. The geometrical parameter of the heater are shown in figure 4.1.

Table 3.1 The dimension of air heater geometry parameters

Heater type	Solar air heater having corrugated shape inside the channel
Length	300 mm
Fluid	Air
Width	50 mm
Height	20 mm

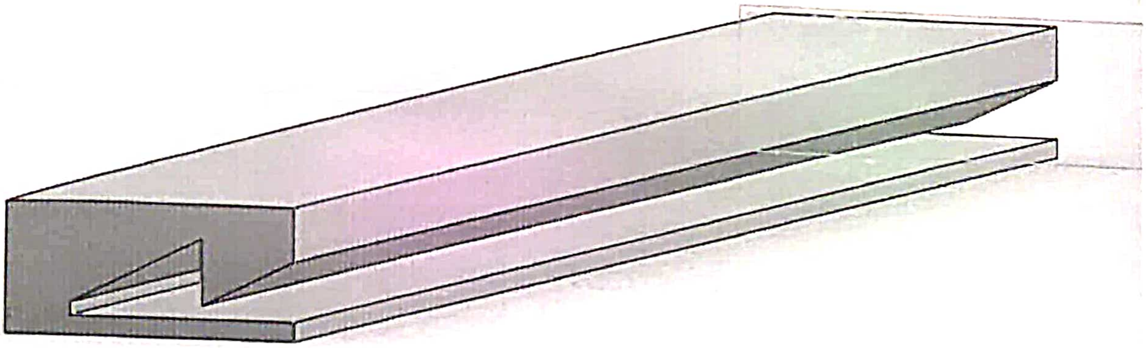


Figure 3.1 (a) Solar air heater model

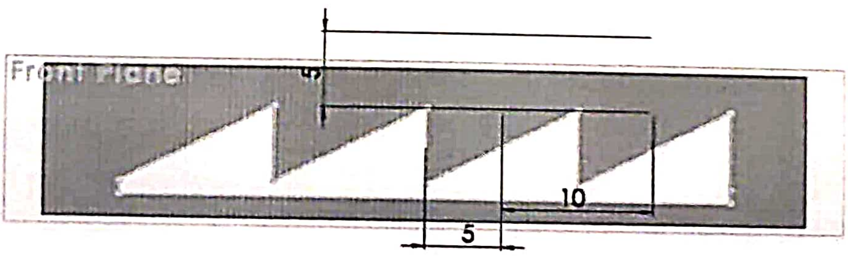


Figure 3.1 (b) Model showing various dimensions

3.2.1.2 MESH

A non-uniform mesh in both horizontal and vertical direction directions proved to be sufficient to model the system. The meshing size is comparatively small near the boundaries so a good estimate of the gradients can be obtained. Mesh sensitivity analysis was performed to obtain the best and suitable mesh elements. Following mesh details were used to solve the fluid domain.

Nodes	239067
Elements	123586
Element Size	5.e-004 m
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	5.e-004 m

3.2.2 ANANLYSIS AND SOLUTION

The governing equations were solved using the commercial CFD package FLUENT with the following simplifying assumptions:

The following settings were used for the model solution:

Table : Simulation settings

Function	Specification
Solver	Default: segregated (solver), implicit (formulation), 3D (space), absolute(velocity formulation), cell based (gradient orientation),

	superficial velocity
Viscous	Laminar: laminar Turbulent: k-ε, realizable, non-equilibrium wall function. Every other setting is default
Material	Steel. Default: ρ =8030Kg/m^3, C_p=502.48 J/Kg.K, k=16.27 W/Mk
Boundary Conditions: Fluid Inlet Outlet Wall	Air. Every other thing is default. Mass flow rate (Mass flow rate specification method), Mass-flow rate =0.034Kg/s (one of the values used), Total temperature = 300K, Operating pressure = 101325 Pa, Direction vector (Direction specification method), Absolute (Reference frame), Z- component of flow direction = 1, Y-component of flow direction = 0, Intensity and hydraulic diameter (Turbulence specification method), Turbulence intensity = 5%. Default: pressure-outlet Thermal: Temperature (Thermal conditions), Temperature = 1000K, Wall thickness = 0m, Heat generation rate 0 w/m^3 , Material name : Steel, Momentum: Stationary wall (Wall motion), No slip (shear condition), Roughness height = 0m, Roughness constant = 0.5

Solution controls	COUPLED (Pressure-velocity coupling), Under-Relaxation factors: Pressure= 0.75, Density = 1, Body forces = 1, Momentum = 0.75, Turbulence kinetic energy = 0.8, Turbulence dissipation rate = 0.8, Turbulent viscosity = 1, Energy = 1. Discretization: Standard (Pressure), Second order upwind (Momentum, Turbulent kinetic energy, Turbulent dissipation rate, and Energy)
Residual monitors	Print and Plot (Options), Default: Convergence criterion; Continuity = .001, X-velocity = 0.001, Y-velocity = 0.001, Energy = 1e-06, k=0.001, H=0.001. Scale (Normalization)
Solution initialization	Default: Relative to cell zone (Reference frame), Initial values: Gauge pressure = 0Pa, X-velocity = 0m/s, Y-velocity = 0m/s, Turbulent kinetic energy 1.39 m^2/s^2, Turbulent dissipation rate=1.73 m^2/s^3, Temperature = 300K
Run calculation	Number of iterations = 1000

CHAPTER 4

RESULTS AND DISCUSSIONS

RESULTS AND DISCUSSIONS

The mathematical model and methodology for solving the problem used in present work is liscussed in previous chapter. This mathematical model is studied, solved and validated using a ANSYS (FLUENT) programme in the present work. Present work also includes parametric study of solar air heater geometry and obtaining heat transfer characteristics and different parameters of air heater.

Simulations were carried out with constant mass flow rate for the geometry considered here for olar air heater. The post processor was carried out in Fluent, and during the process, the following bservations were encountered

As discussed in above chapter, 300 mm length and width of 50 mm solar air heater geometry are aken for the simulation in present work. The other parameters have to be calculated in order to ptimize the complete system. Following sections describes the parametric study and the ptimization of the system.

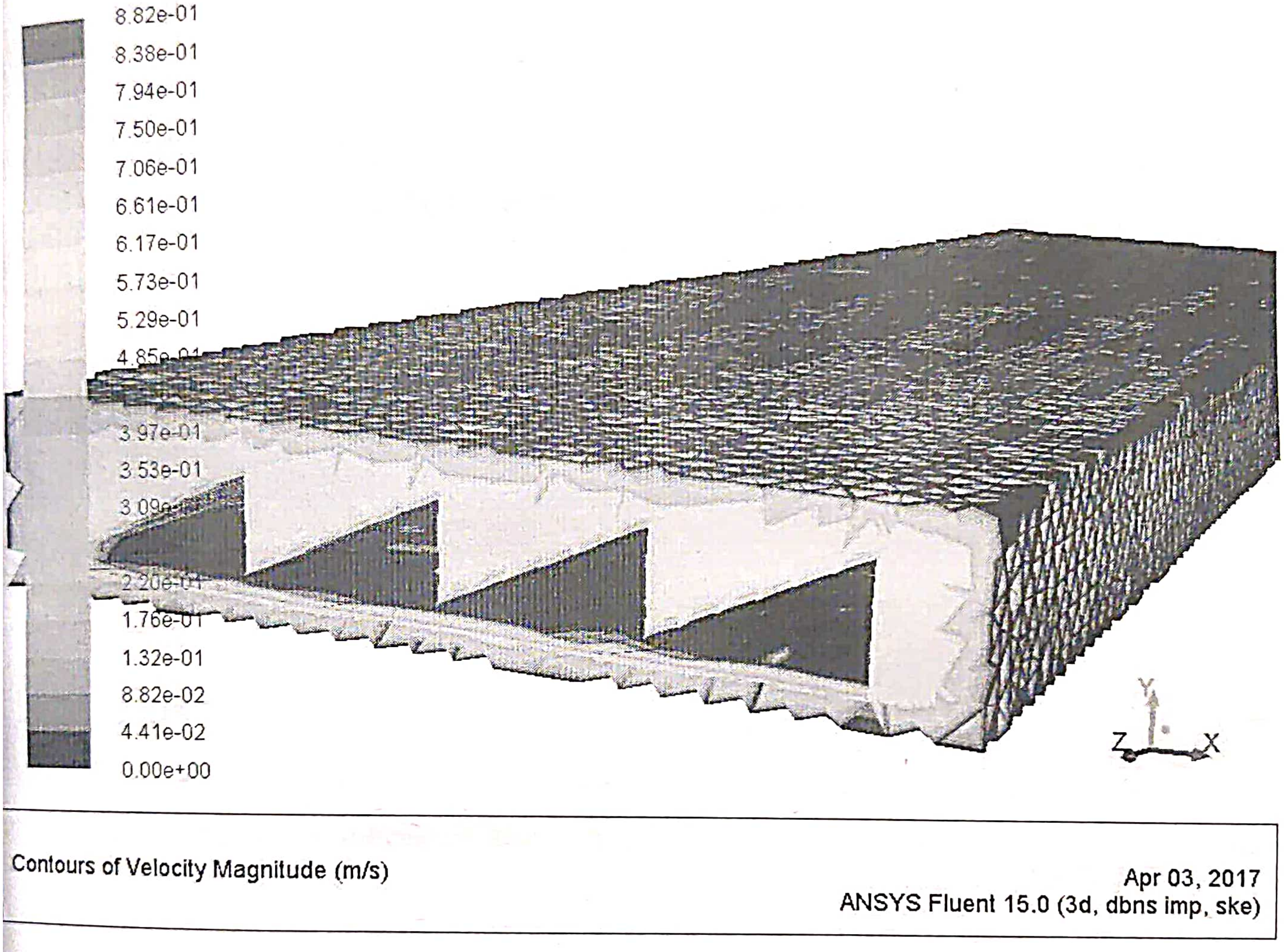
.1 Effect of corrugated geometry on heat transfer enhancement in Solar air heater

t has been observed that the total heat transfer rate, temperature and enthalpy at the outlet of the hannel is increased significantly by applying corrugated geometry inside the boiler tube.

This is due to increased roughness surface area for same mass flow inlet. Lower the low value of mass flow inlet higher will be the surface area and tube get heated uniformly and in relatively shorter time as compare to high mass flow inlet and smooth tube.

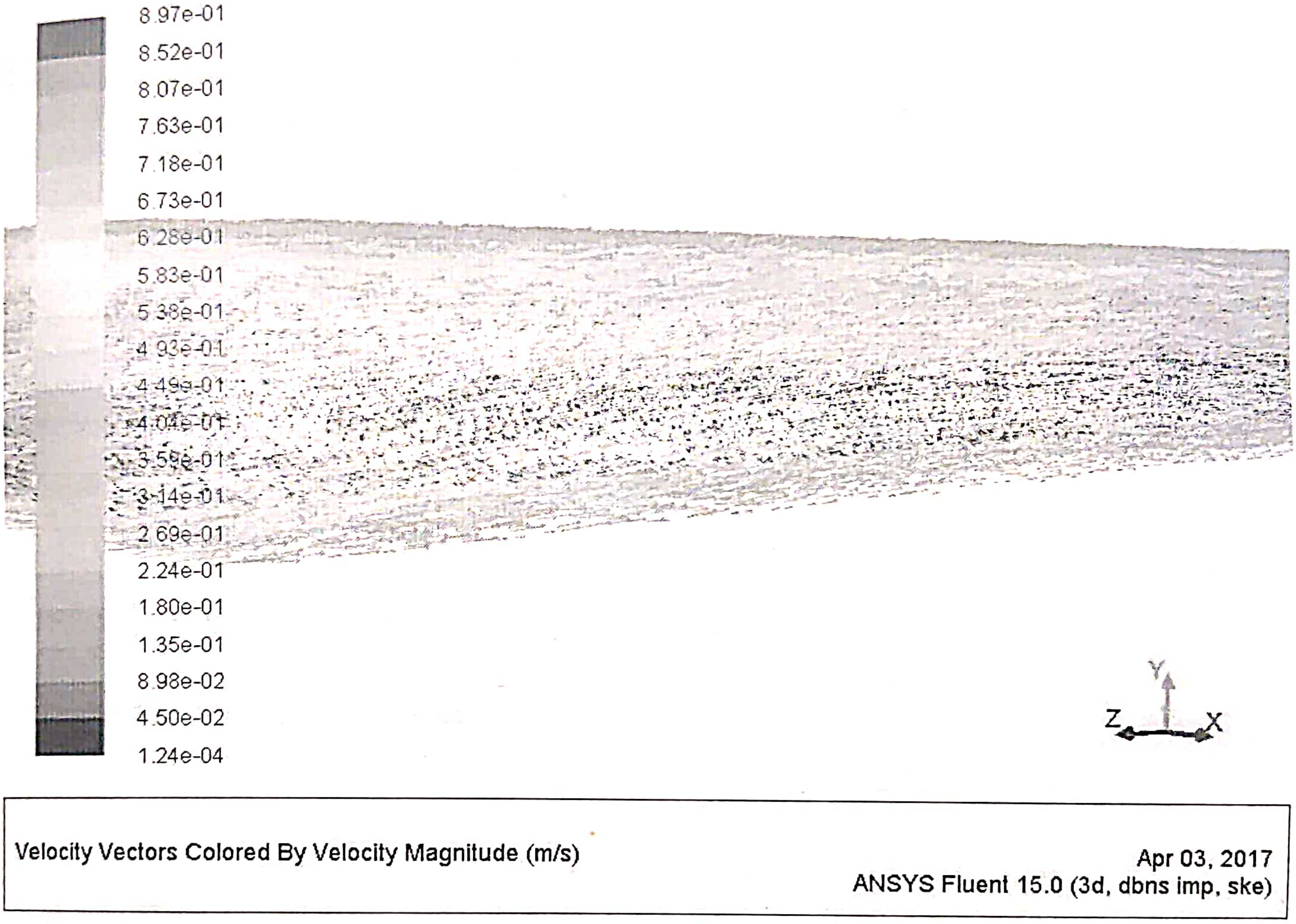
Heat losses to the ambient and poor heat convection from absorber plate to the air stream are highlighted as major drawback of large scale promotion of conventional solar air heaters. The present study revealed that the aforementioned demerits can be alleviated to a certain extent by using matrix collectors.

n increase in velocity increases the convective heat transfer coefficient of air, which reduces the seful heat gain by increasing the top losses which in turn affects the increase in effective fficiency for the rise in velocity in the solar air heater. The effect of mass flow rate, the number f glass covers, heat flux, velocity and variation in width of duct on thermal and effective fficiencies of roughened solar air heater are presented in the form of plots in the present study.



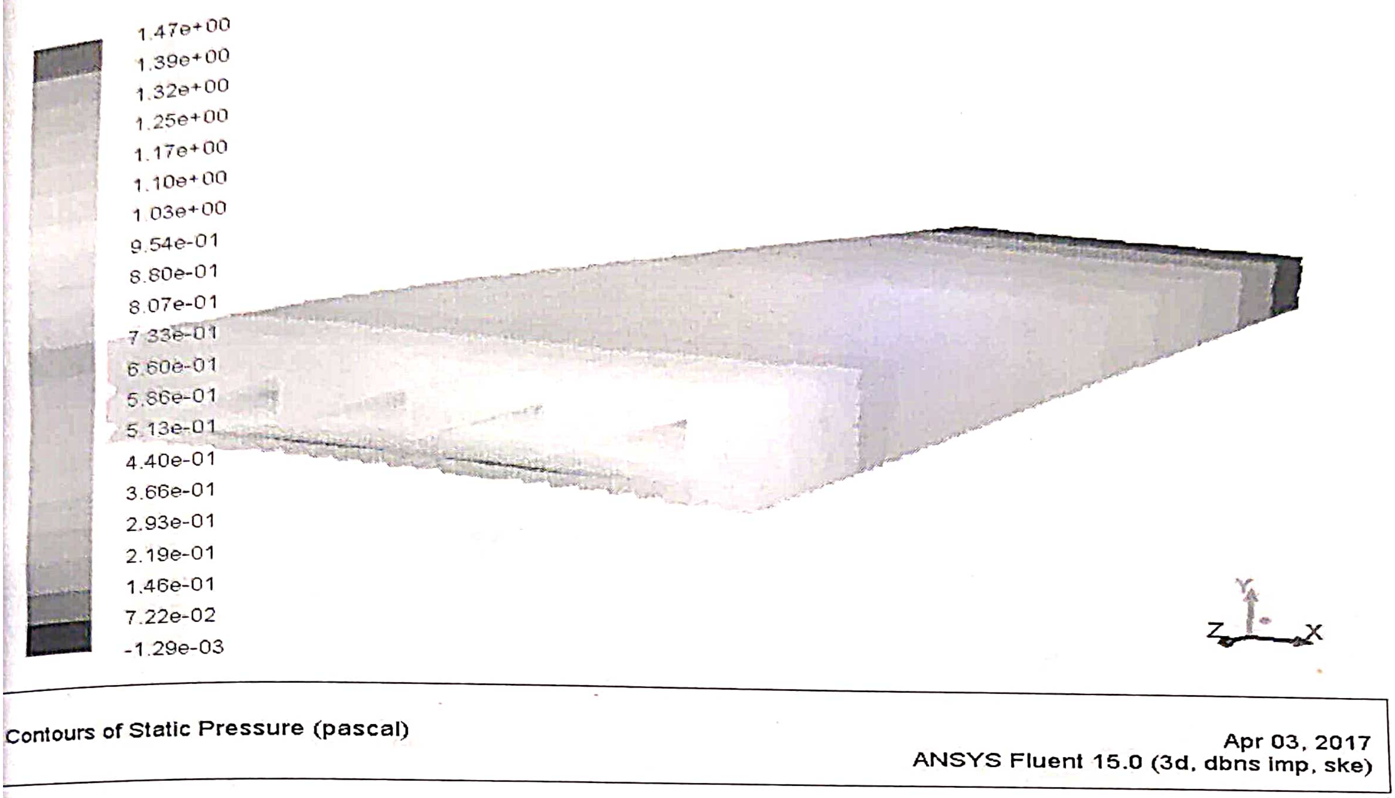
Velocity distribution in Channel

When fluid to be heated enters into the channel, due to convective heat transfer the fluid inside the heater get heated. Due to wall heat transfer coefficient and area product fluid get heated initially up to a certain length and after a long time fluid reaches to a uniform temperature.

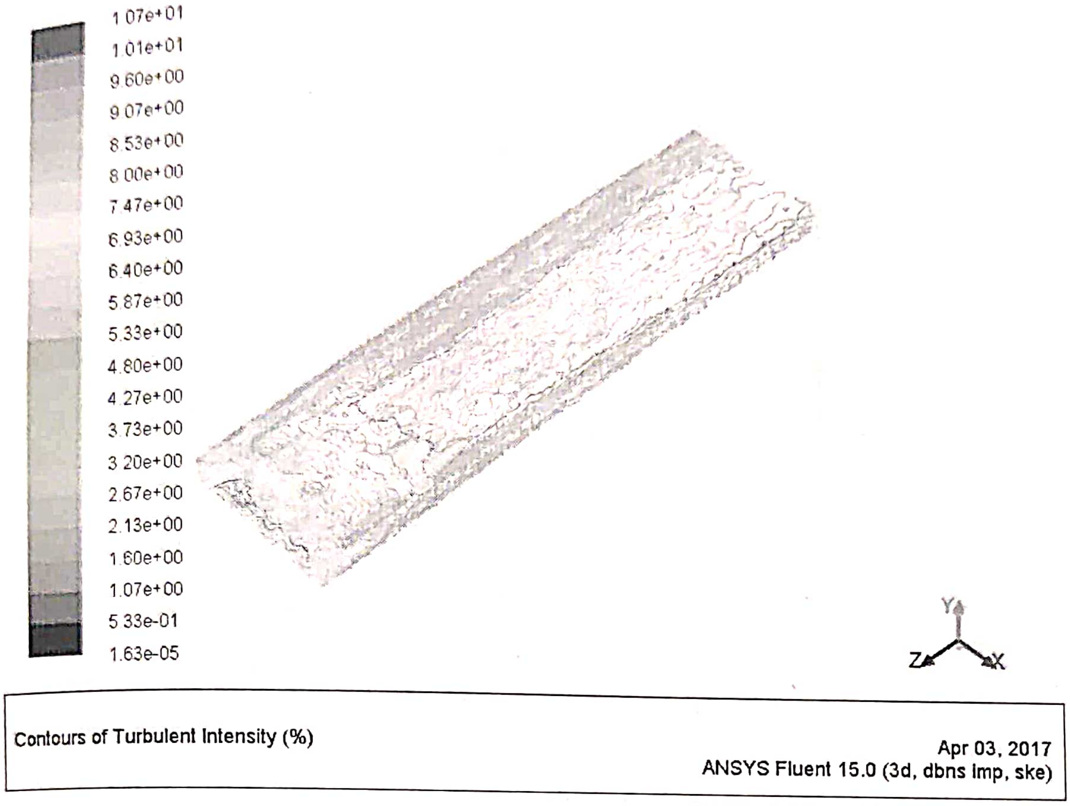


Velocity vector distribution in air heater

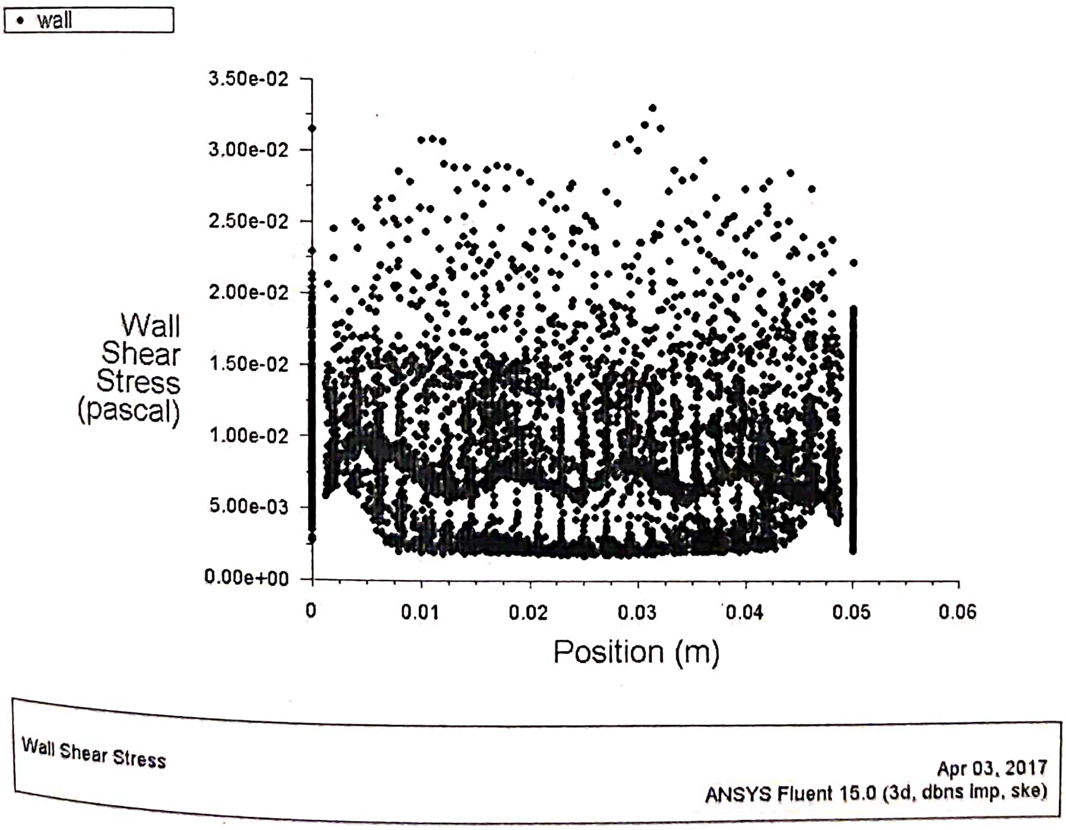
:om the above contour we can see the flow of air inside the heater. Due to corrugated geometry ıside the channel the flow is going to be turbulent and hence to increase the heat transfer rate in r heater. The mean temperatue of the working fluid increases with the distance from the entrance ; expected. Also expected, the increase is rapid nearer the wall because of high heat transfer rates ı this region due to higher temperature differences between the wall surface and the fluid.



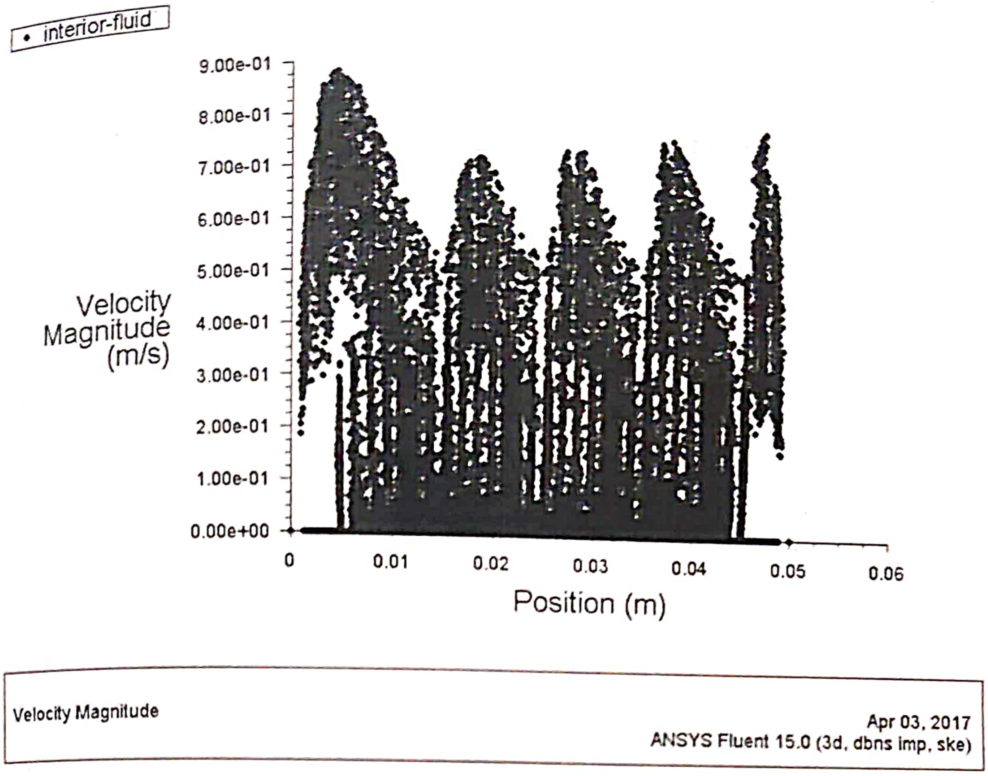
Contours of static Pressure



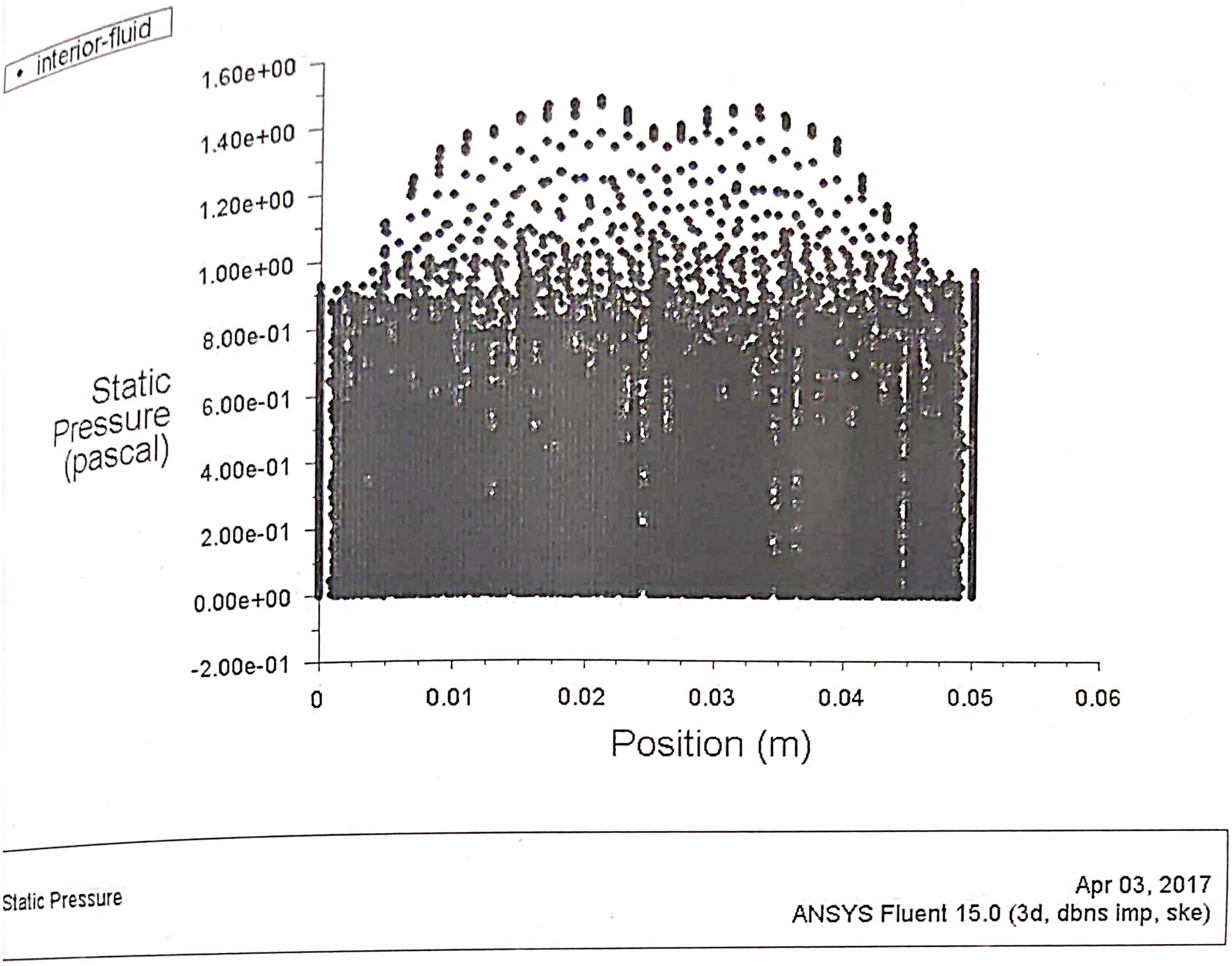
Contours of Turbulent Intensity



Wall shear stress in air heater plate



Velocity graph throughout the length of air heater



Pressure variation in Solar air heater

CHAPTER 5

CONCLUSIONS AND FUTURE SCOPE

5 CONCLUSIONS AND FUTURE SCOPE

5.1 CONCLUSIONS

n this chapter, main results of the project and scope for further study have been given. The heat ransfer in the air heater has been analyzed, and its dependency on geometry is reported in this tudy.

nhancement in the heat transfer in air heater will reflect in the performance of the solar air heater. he effective efficiency of the system increases with an addition in the number of glass covers and idth of the duct. Thermal and effective efficiency increases with the increase in solar insolation. n increase in velocity increases the convective heat transfer coefficient of air, which reduces the seful heat gain by increasing the top losses which in turn affects the increase in effective fficiency for the rise in velocity in the solar air heater. The effect of mass flow rate, the number f glass covers, heat flux, velocity and variation in width of duct on thermal and effective fficiencies of roughened solar air heater are presented in the form of plots in the present study.

The heat transfer enhancement effect in corrugated channel is primarily due to induced urbulence which gives higher heat transfer rate. As shown from the study, different geometries give different results for the same flow conditions (mass-flow rate). The choice of appropriate geometry does not always depend on mass-flow rate values, but predictability and almost constant behavior through a low mass-flow rate range, plays an important role.

The model was created using Solid Works 2013 and meshed with Fluent, and the flow nalysis is done with Ansys 14.5. The results showing that the heat transfer is increased.The study how that the improvement in heat transfer can be achieved by changing the internal plane surface o a corrugated shape.

In the present work it is assumed that constant mass flow rate of fluid in all the design are going through the channel respectively. In actual practice this is not the case as mass flow rate is not always constant in actual practise for the applications. Inclusion of these varying flow rates is a future scope of this present work.

Based on the literature review discussed in this work, following suggestions and recommendations have been made for the development of SAHs:

1. Using SAH with azimuth sun tracking and just adjusting of the optimum elevation of the day.
2. More researches may be carried out for SAH with reflectors as the sun tracking system is more effective and it is capable of improving the efficiency of the heater with reflectors compared to fixed solar heater.
3. Further more studies should be conducted to improve heat transfer efficiency through new materials and geometry for the aim of selecting cheaper and scalable fabrication methods to reduce manufacturing costs, in addition trying to obtain simple configurations.
4. More research works are required in the area of artificially roughened of double pass SAHs duct as few studies are available on this aspect either experimentally and theoretically.

Experimental work can be done in order to validate the present work, evaluated parameters.This is an another future scope of this present work.

REFERENCES

1] WomacAR,TompkinsFD,DebuskKE.Evaluation of solar air heaters for crop drying. Energy ,gric 1985;4:147–57.

2] Karim MA,MNAHawlader. Development of solar air collectors for drying applications. nergy Convers Manag2004;45:329–44.

3] Hachemi A. Experimental study of heat transfer and flow friction in solar air heaters with and ithout selective absorbers.RenewEnergy1999;17:155–68.

4] Mohamad AA.High efficiency solar air heater.SolEnergy1997;60:71–6.

5] SaxenaA,El-SebaiiAA.A thermodynamic review of solar air heater. Renew Sustain nergyRev2015;43:863–90.

] Al-kamil MT,A-GhareebAA. Effect of thermal radiation inside solar air heaters. nergyConversManag1997;38:1451–8.

] Hachemi A. Technical note comparative study on the thermal performances of solar air heater ollectors with selective and non selective absorber-plate. RenewEnergy1999;17:103–12.

] HachemiA.Thermal heat performance enhancement by interaction between the radiation and nvection in solar air heaters. Renew Energy1997;12:419–33.

] Mahdi NAL,NSLBaharna. Thermal performance of an n-pass solar air heater. enewEnergy1991;3:527–32.

10] Soulayman SS. Theoretical and experimental study of at wo channel air heater with perforated rst absorber.RenewEnergy1991;2:331–4.

11] Verma Ratna, ChandraRam, GargHP. Optimization of solar air heaters of different esigns.RenewEnergy1992;2:521–31.

12] SunWei,JiJie, HeWei. Influence of channel depth on the performance of solar air eaters.Energy2010;35:4201–7.

13] Choudhury C,GargHP. Design analysis of corrugated and flat plate solar air heaters. enewEnergy1991;1:595–607.

14] Singh Yadav Anil, Kumar Thapak M. Artificially roughened solar air heater: experimental nvestigations.RenewSustainEnergyRev2014;36:370–411.

15] Verma Ratna,ChandraRam,GargHP.Parametric studies on the corrugated solar irheaterswithandwithoutcover.RenewEnergy1991;1:361–7.

16] Varun SainiRP,SingalSK. A review on roughness geometry used in solar air heaters. olEnergy2007;81:1340–50.

17] Gupta MK,KaushikSC.Performance evaluation of solar air heater for various artificial oughness geometries based on energy, effective and exergy efficiencies. enewEnergy2009;34:465–76.

8] Bhushan Brij,SinghRanjit. A review on methodology of artificial roughness used in duct of olar air heaters.Energy2010;35:202–12.

9]GawandeVB,DhobleAS,ZodpeDB. Effect of roughness geometries on heat transfer nhancement in solar thermal systems – a review. Renew Sustain Energy Rev2014;32:347–78.

20] YadavAnilSingh,Thapak ManishKumar. Artificially roughened solar air heater: experimental vestigations. RenewSustainEnergyRev2014;36:370–411.

21] Singh D,BansalNK. A simplified theory for a matrix solar collector. IntJ Energy es1983;7:173–7.

22] Suggett GJ.Comments on a simplified theory for a matrix solar collector. IntJ Energy es1984;8:305–23.

23] Bansal NK.Solar crop drying. In:GargHP,DayalM,FurlanG,SayighAAM, editors. Physics and chnology of solar energy.Netherlands:Springer;1987. p. 413–45.

24] Gupta CL,GargHP.Performance studies on solar air heaters. SolEnergy 1967;11:25–31.

25] Choudhury C,GargHP.Performance of air-heating collectors with packed airflow assage.SolEnergy1993;50:205–21.

26] Lalji MK,SarviyaRM,BhagoriaJL .Exergy evaluation of packed bed solar air heater. RenewSustainEnergyRev2012;16:6262–7.

[27] Ozturk HH,DemirelY. Exergy- based performance analysis of packed-bed solar air heaters.IntJEnergyRes2004;28:423–32.

[28] Alta D,BilgiliE,ErtekinC, YaldizO. Experimental investigation of three different solar air heaters: energy and exergy analyses. Appl Energy 2010;87:2953–73.

[29] KurtbasI, Durmus A. Efficiency and exergy analysis of a new solar air heater. RenewEnergy2004;29:1489–501.

[30] Sodha MS,BansalNK,SinghD,BharadwajSS.Performance of a matrix air heater. JEnergy1982;6:334–9.