COMPARISION OF V-RIB SOLAR ABSORBER PLATE WITH DIFFERENT RELATIVE GAP WIDTH ON EXPERIMENTAL AND CFD SIMULATION

Jouhri Kumar Akale [1], Dr. S.K Nagpure[2],
Mtech.Scholar[1],HOD[2],
Department of Mechanical Engineering[1,2]
Scope College of Engineering. Bhopal, India

Abstract* — In a Computational Fluid Dynamics (CFD), the trial has been conducted on the heat transfer and Friction factor, for a solar air heater with Circular ribs with different relative gap width. In the CFD investigation it was assumed that, the rib height is 2mm, the pitch of the rib (P) is 20, the Relative Roughness pitch (P/e) is 10, and the angle of attack (α) is 60°. This investigation was carried out on a Reynolds Number ranging in between 3000-15000.When the desired heat transfer was carried with smooth and rough ribs with symmetrical gaps it was found that; The smooth ribs could not transfer the desired heat due to absence of friction; so, they are not preferred practically. The rough ribs were efficient enough to transfer the desired heat, but they are not economical and are very complex in design and construction. Whereas, the Circular ribs with different relative gap width gives more area of contact. So, there is enough time to transfer the heat from the ribs to the passing air which touches the ribs and the heat transfer takes place efficiently. Thus, the Circular ribs with different relative gap width, when compared with the smooth and rough ribs with symmetrical gaps, it was concluded that the related Nusselt Number and Friction Factor for Circular ribs with different relative gap width was more efficient as well as economical, than that of smooth and rough ribs with symmetrical gaps for flow Reynolds Number.

***Keywords* —CFD Analysis, FLUENT, GAMBIT, Solar Duct, Friction factor, and Nusselt number.**

I. INTRODUCTION

1.1 Solar power

As we all know that the natural resources of fossil energy are limited this is available in the form of oil and solid substance like crude oil, coal and many others. They are used at very large scale due to this they are depleting much faster rate. Hence is the need of the current scenario to find alternative source of energy. Solar energy is finding as the one of the most easily available, most promising and important renewable source of energy. It is available in abundant form at anywhere in earth. It is also very easy to capture and utilized it. The easiest way to utilized and store solar energy is to convert it in to heat energy which is basically utilized for heating purposed. Heating air through solar power form the major component of solar power exercise system. After burning the fossil fuels harmfully greenhouse gases (CO_2, SO_2, NO_x) can remain as byproduct, which causes higher levels of acid rain during the rainy season, it also increases the amount of harmful atoms in air which create air pollution, due to the increase of chlorine atom in atmosphere depletion of ozone layer is also happening and also causing global warming. It is predicted that globally it is going to increase very faster rate in future due to expectations of a considerable increase in power and heat demand. The heated air energy demand relates with different sector is quite significant. Solar power air heaters have been used with the aim to reducing the percentage of consumption of conventional fuels to a very large extent. The residential and industrial sectors are larger consumers of fossil energy. Therefore, the heating system and air conditioning devises of residential and industrial buildings generate an large amount of CO_2 and many other gases which is responsible for global warming. Fossil energy required to heat and energy required to maintain air-conditioning of the buildings can also be reduced by using renewable sources as alternative to fossil energy (Sanjay and Vilas, 2014)[14]. Heating of air using solar radiation is a technology where the radiation coming from the sun, that is solar radiation, is entrapped by an absorbing medium and utilized for air heating. It is a technology that uses inexhaustible energy for conditioning of air or maintaining the temperature of buildings or for different other purposes (Omajaro and Aldabbagh, 2010) [2].

Solar heater is the most economical and efficient solar technologies, which is widely used due to their easiness in space heating, removing the moisture from timber, used for drying the industrial products, vegetables and fruits. They may be also used in combination with photovoltaic solar absorber panels which is used to manufacture photovoltaic thermal hybrid solar energy collectors (hybrid PV/T systems or PVT) to produce heating effect or to generate electricity. The basic advantages of solar power collectors are: the fluid which is flowing inside the collectors does not get freeze or boil, they cannot create noise during flowing, the operating of solar panel system is very safe and the operating cost is also very less, system cannot produce any kind of harmful wastes and the running life of solar system is also long enough life cycle (Abdullah and Bassiouny, 2014).[5] but solar power collectors have some following drawbacks: low density, the thermal absorption capacity of solar panel is low and the thermal conductivity of air is also low which lead to low thermal efficiency, high cost system installation and non-uniform rate of heat generation

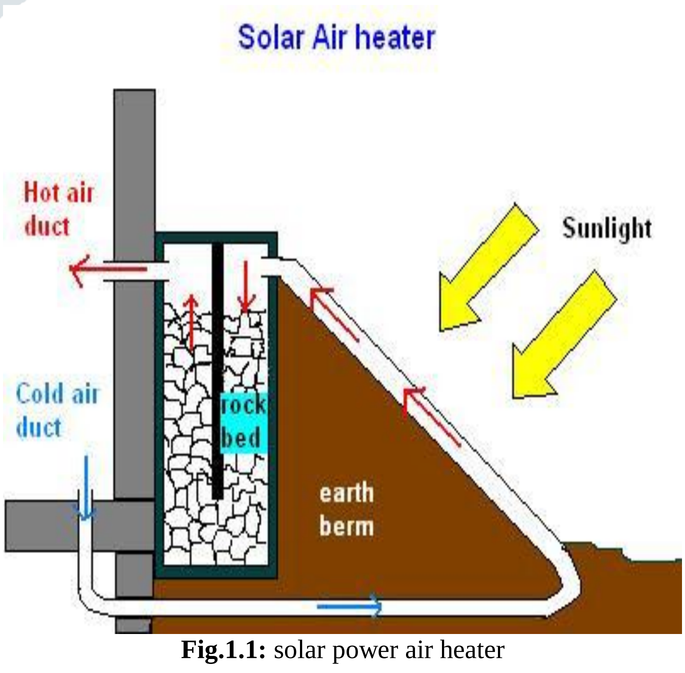

Fig.1.1: solar power air heater

1.2 Classification of solar air heater system based on different type's solar power collector

Collectors are commonly classified by their air ducting methods as one of three types

- Through-pass collectors
- Front-pass
- Back-pass
- Combination front and back pass collectors

II. SOLAR ENERGY COLLECTORS

Solar collectors are the key component of active solar-heating systems. Solar collectors gather the sun's energy, transform its radiation into heat, and then transfer that heat to water, solar fluid, or air. The solar thermal energy can be used in solar water-heating systems, solar pool heaters, and solar space-heating systems.

Solar energy collectors are classified as:

1. Flat plate collectors.
2. Concentrating collectors.

If the area of interception of solar radiation is same as the area of absorption, the collector is known as flat plate collector.

2.1 Flat Plate Collectors

Flat-plate collectors are the most common solar collector for solar water-heating systems in homes and solar space heating. A typical flat-plate collector is an insulated metal box with a glass or plastic cover (called the glazing) and a dark-colored absorber plate. These collectors heat liquid or air at temperatures less than 100°C.
The major components of Flat Plate collectors are:

- The absorber plate used for absorbing solar radiations, normally metallic with a black surface. A wide variety of other materials can be used with air heaters. It is usually one plate or an assembly of metal sheets or plates forming a nearly continuous surface coated with radiation absorbing black paint, black porcelain enamel or a metallic oxide.
- A transparent cover which may be one or more sheets of glass or radiation transmitting plastic film or sheet. As the number of covers increases, the loss of heat from top of collector decreases while intensity of radiation incident on absorber plate also decreases.
- Tubes, passages or channels are integral with the collector absorber plate or connected to it, which carry the water, air or other fluid to transfer energy from absorber plate to the fluid.
- Insulation, provided at the back and sides to minimize heat losses.
- The casing or container, which encloses the components and protects them from the weather.

2.2 Concentrating Collectors

A concentrating collector utilizes a reflective parabolic-shaped surface to reflect and concentrate the sun's energy to a focal point where the absorber is located. To work effectively, the reflectors must track the sun. These collectors can achieve very high temperatures because the diffuse solar resource is concentrated on a small area. In fact, the hottest temperatures ever measured on the earth's surface have been at the focal point of a massive concentrating solar collector. Concentrating collectors have been used to make steam that spins an electric generator in a solar power station. This is sort of like starting a fire with a magnifying glass on a sunny day.

III.PERFORMANCE EQUATIONS FOR A SOLAR COLLECTOR

The performance of solar collector is described by an energy balance that indicate the distribution of incident solar energy into useful energy gain (Qu) and heat losses like bottom (Qb) and top (Qt) as shown in Fig.1.2. The details of the performance analysis of a solar collector are discussed by Duffie and Beckman [31] and Goswami [41].The heat transfer in a solar collector takes place by simultaneous radiation, convection and conduction. The heat transfer from the top takes place by convection and radiation while from the side and bottom is by conduction. The net rate of useful energy collected per unit area is the difference of the amount of solar energy absorbed and the energy loss by the collector to the surroundings

IV. LITERATURE REVIEW

Yadav and Bhagoria (2013) [5] - This investigation the study of heat transfer and fluid flow processes in an artificially roughened solar air heater by using computational fluid dynamics (CFD). The effects of small diameter of transverse wire rib artificial roughness on heat transfer and fluid flow have been investigated. The situation for optimum performance has been determined in term of thermal enhancement factor. A maximum value of thermal enhancement factor has been found to be 1.65 for the range of parameters investigated. We found Nusselt number are also increases with an increase of Reynolds number..

Yadav and Bhagoria (2013) [6] - This investigation is solar air heater is one of the basic equipment through which solar energy is converted into thermal energy. Computational fluid dynamics (CFD) investigation is also carried out to select best turbulence model for the design of a solar air heater. CFD simulation result to found to be in good arrangement with experimental result and with the standard theoretical approaches. A two-dimensional CFD analysis has been carried out to study heat transfer and fluid flow behavior in a rectangular duct of a solar air heater with one artificial roughened wall having circular transverse wire rib roughness.

Yadav and Bhagoria (2013) [7] - This investigation is conducted to analyze the two-dimensional incompressible Navier-Stokes flows through the artificially roughened solar air heater for relevant Reynolds number ranges from 3800 to 18,000.Atwo-dimensional CFD model of an artificially roughened solar air heater having equilateral triangular sectioned rib roughness on the absorber plate has been proposed and used to predict the heat transfer and flow friction characteristics. Further, we found the Nusselt number tends to increase as the Reynolds number increases in all cases.

Yadav and Bhagoria (2014) [8] - A numerical investigation on the heat transfer and fluid flow characteristics of fully developed turbulent flow in a rectangular duct having repeated transverse square sectioned rib roughness on the absorber plate has been carried out.The two-dimensional fluid flow and heat transfer processes in a rectangular duct of a solar air heater with one artificial roughened wall having square sectioned transverse rib roughness are analyzed numerically, and a detailed description of the average heat transfer and flow friction factor, i.e. Nusselt number and friction characteristics, are obtained. Further, we found the Nusselt number tends to increase as the Reynolds number increases in all cases.

V. COMPUTATION FLUID DYNAMICS

5.1 Introduction

Computation Fluid Dynamics (CFD) is the branch of fluid science which deals with a variation occurs on fluid flow, basically computational fluid dynamics opt an finite volume method as methodology and for base equation it follows the Eulerian equation, i.e. when gravity forces were not considered, pressure force and viscous force are used to simulate the desired fluid flow problem.

5.2 Fluent Solver

Computation Fluid Dynamics consists of several domains to solve fluid flow problem like CFX, fluent (poly flow), fluent (blow moulding), fluent, fluent solver works under computational fluid dynamics, it obeys the three governing equation with respect to base equation (Eulerian equation) i.e. energy equation, momentum equation and continuity equation by applying or solving through this algorithm, the further results were obtained and variation could be determine.

5.3 Finite volume method

Finite Volume Method is used to solve the fluid flow problems by obtaining the convergence of Eulerian equation and governing equation, this method works on volume of fluid or volume of fraction, it consists of energy equation, momentum equation and continuity equations with respect to pressure force, viscous force or gravity force to solve the fluid flow problem, in case of heat exchanger, radiation, turbulence, laminar flows, acoustics and also deals with aerodynamics, HVAC

5.4 Governing equations:

5.4.1 Continuity equation:

$$A_1V_1=A_2V_2$$

A_1 = area of inlet
V_1= velocity at inlet
A_2= area of outlet
V_2= velocity at outlet

This equation shows the flow is pressure based or density based i.e. if a flow is pressure based the vortices and stream line of fluid is normal, if the flow is density based the fluid flow and stream line is in a high pressure.

5.4.2 Momentum Equation

This equation justified that the flow of fluid consists of definite mass and product of velocity with respect to mass to determine the momentum of fluid flow.

$$\frac{\partial}{\partial x_i}(\rho u_i u_j) = \frac{\partial}{\partial x_i}\left(\mu \frac{\partial u_i}{\partial x_i}\right) - \frac{\partial p}{\partial x_j} \qquad (4.1)$$

5.4.3 Energy Equation

This equation works on present simulation model when heat flux and radiation were applied on boundary condition to determine the temperature variation on fluid flow and on heat transfer solid element to determine temperature variation.

$$\frac{\partial}{\partial x_i}(\rho u_i T) = \frac{\partial}{\partial x_i}\left(\frac{k}{c_p}\frac{\partial u_i}{\partial x_i}\right) \qquad (4.2)$$

5.5 Procedure for solving problem with fluent:

- Pre- processor
- Solver
- Post- processor

5.5.1 Pre-processor

It is a process on which model is created for simulation, meshing of the domain is done and boundary conditions were applied i.e. inlet, outlet, heat flux, wall, etc.

5.5.2 Solver

It is used to apply the governing equation and base equation on pre-processor to determine the variation on fluid flow.

5.5.3 Post processor

It is used to determine the results obtaining from fluent solver in a form of contour plots, in a form of a velocity and stream line contour plots etc.

5.5.3 Turbulence Modeling

Turbulent flows are characterized 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 small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations.

FLUENT provides the following choices of turbulence models:

- Spalart-Allmaras model
- k-ε models
  - Standard k-ε models
  - Renormalization-group (RNG) k-ε models
  - Realizable k-ε models
- k-ω models
  Standard k-ω models
  Shear-stress transport (SST) k-ω models
- v2-f model (addon)
- Reynolds stress model (RSM)
  Linear pressure-strain RSM model
  Quadratic pressure-strain RSM model
  Low-Re stress-omega RSM model
- Detached eddy simulation (DES) model
  Spalart-Allmaras RANS model
  Realizable k- ε RANS model
  SST k- ω RANS model
- Large eddy simulation (LES) model
  Smagorinsky-Lilly subgrid-scale model
  WALE subgrid-scale model
  Kinetic-energy transport subgrid-scale model

5.5.4 Choosing a Turbulence Model

It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a special class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation.

VI. MODELING AND ANALYSIS

Geometry is framed in demonstrating programming UNIGRAPHICS and its foreign made to the ANSYS workbench where lattice is finished, and sends out the work to FLUENT 15.0. The limit conditions, material properties, and including properties are set through parameterized case records. Familiar tackles the issue until either as far as possible is met, or the measure of emphases determined by the client is accomplished. The procedure for resolving the problem is:

- Create the geometry.
- Meshing of the domain.
- Set the material properties and boundary conditions.
- Obtaining the solution

6.1 PREPARATION OF THE CAD MODEL

The measurements of the computational area sun based pipe depended on the work by Rajesh Maithani, J.S. Saini. After this procedure the imperative are connected and along these lines the model is accomplished in demonstrating programming UNIGRAPHICS The accompanying section (4.1.1) demonstrates the parameters of sun based air radiator channel roughened misleadingly with V-ribs and Semicircular V-ribs.

6.1.1 Modeling of duct with smooth absorber plate

The geometry of conduit with smooth safeguard plate is made and coincided on UNIGRAPHICS. At that point subsequent to putting the limit conditions this fit record is keep running on FLUENT 15.0 and the outcome acquired from familiar are utilized for approval of the outcome with the base paper.

6.1.2 Solver setting and Boundary Conditions

In CFD examination before the running of geometry on FLUENT 15.0, solver defining and limit conditions are given. In the present case taking after solver defining and limit conditions are connected

VII. RESULT AND DISCUSSION

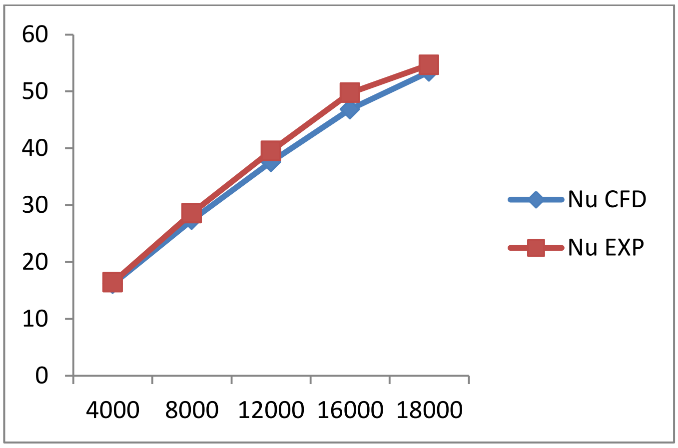

Figure 7.1 Graph of Nusselt Number versus Reynold Number

This diagram demonstrates that Nu number got from CFD Follows the way of Nu Obtained Experimentally , Average variety of Nu number in the middle of CFD and Experimentally acquired is 3.76 %.

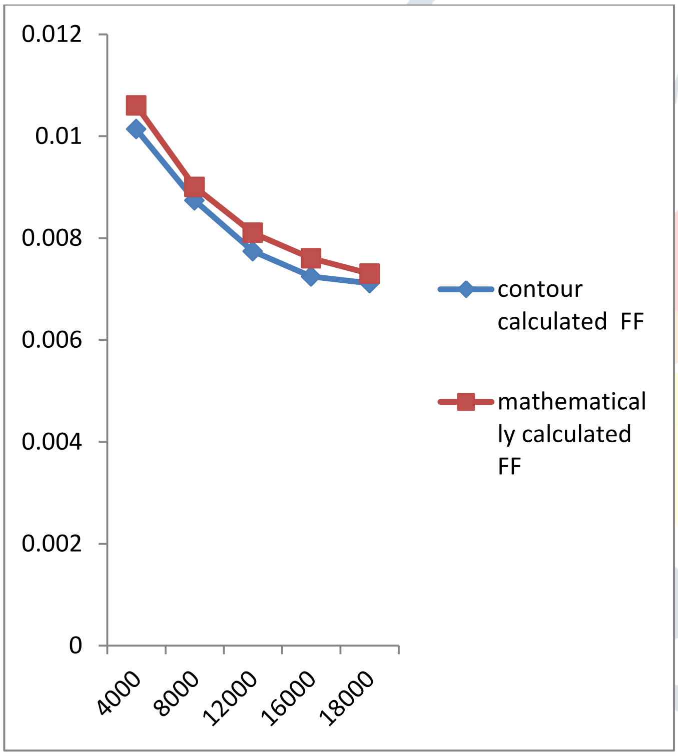

Figure 7.2 Graph of Friction factor versus Reynolds Number

This diagram demonstrates that Friction consider got from CFD Follows the way of Friction component Obtained Experimentally , the normal variety of Friction figure between CFD and Experimentally acquired is 3.91 %.3.91 %

Table no. 5.1 Comparison CFD & Experimental nusslet number and Friction facto

Validation				
Rey no.	Nu no. EXP	Nu no. CFD	F F EXP	FF CFD
re=4000	16.416	16.133	0.011	0.0101
re=8000	28.582	27.394	0.009	0.0087
re=12000	39.533	37.590	0.0081	0.0077
re=16000	49.764	46.860	0.0076	0.0072
re=18000	54.681	53.477	0.0073	0.0071

Table no.5.1.3: Shows friction factor and nusselt no. values for both experimental and CFD on relative gap width = 2

Relative gap width = 2				
Rey no.	Nu no. EXP	Nu no. CFD	F F EXP	F FCFD
3000	41	45.765	0.0333	0.0492
5000	81	101.183	0.0300	0.0418
7000	143	190.623	0.0280	0.0389
9000	159	199.403	0.0265	0.0368
11000	168	204.151	0.0260	0.0368
13000	171	206.893	0.0260	0.0335
15000	173	212.8664	0.0260	0.0353

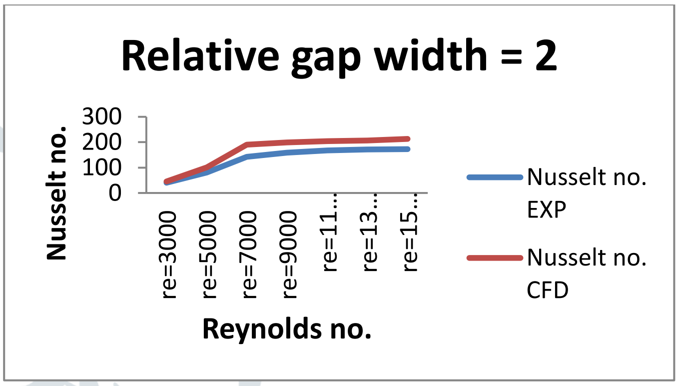

Figure 7.3 Variation in Nusselt no. with respect to Reynolds no. on relative gap width 2

This diagram demonstrates that Nu number acquired from CFD Follows the way of Nu Obtained by Author Experimentally, the normal variety of Nu number in the middle of CFD and Experimentally got is 17.01%.

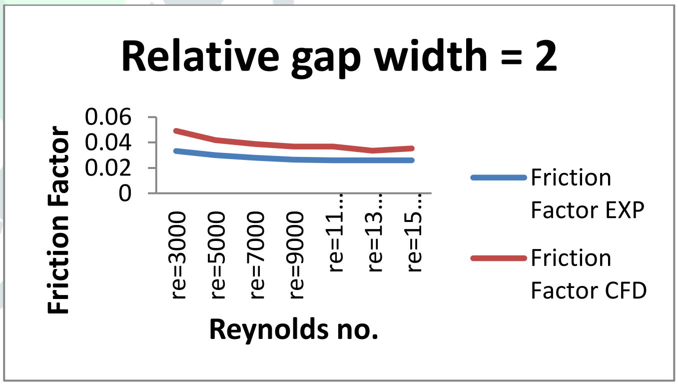

Figure 7.4 Variation in friction factor with respect to Reynolds no. on relative gap width 2.

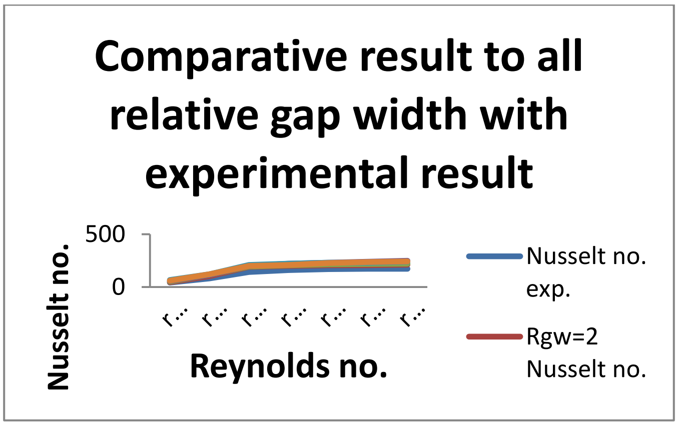

Figure 7.5 Comparison relative gap width with reynolds no. of circular geometry w.r.t Reynolds no

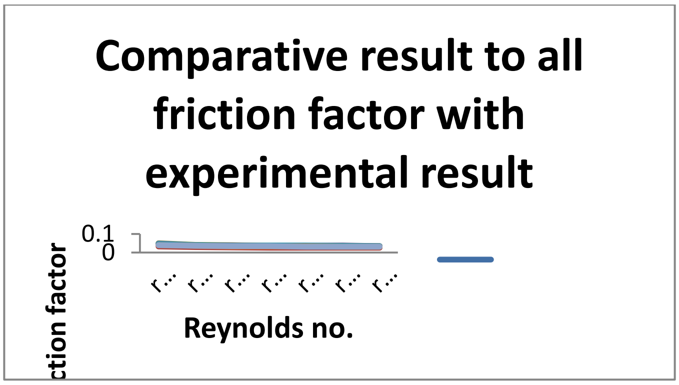

Figure 7.6 Comparison friction factor with reynolds no. of circular geometryw.r.t Reynolds no

7.1CONTOUR OBTAINED FROM FLUENT 15.0 FOR SOLAR DUCT WITH ROUGHENED ABSORBER PLATE WITH RELATIVE GAP WIDTH 2 RIBS

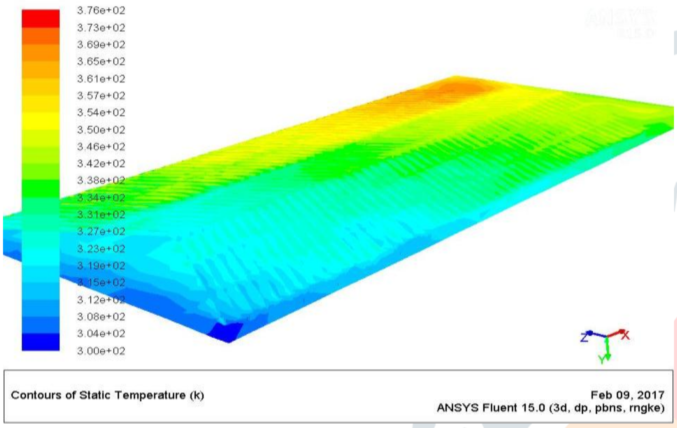

Figure 7.7 Temperature coloured contour plots of air at outlet at Reynolds number 3000

For inlet temperature =300
For outlet temperature =365
Plate temperature = 336
Putting the value of air outlet and plate temperature at Reynolds No. 6000 in calculation 5.1.1.1, Convective heat transfer coefficient obtained is, h= 23
Whereas, Nusselt Number obtained from above h
Nu=45.76523 (NU obtained from CFD for circular rib with different relative gap width geometry)
Nu=41 (NU of author for circular geometry)
Percent increase in Nu number = (45.76523 - 41)/ 45.76523 ×100
Percent increase in Nu number = 10.412%

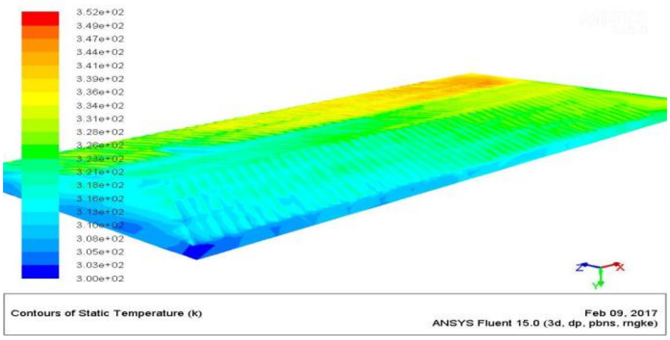

Figure 7.8 Temperature coloured contour plots of air at outlet at Reynolds number 5000

Putting the value of air outlet and plate temperature at Reynolds No. 5000 in calculation 5.1.1.1, Convective heat transfer coefficient obtained is, h= 52
For inlet temperature =300
For outlet temperature =344
Plate temperature = 327
Whereas, Nusselt Number obtained from above h
Nu=101.18354 (NU obtained from CFD for circular rib with different relative gap width geometry)
Nu=81 (NU of authors circular geometry)
Percent increase in Nu number = (101.18354 -81)/ 101.18354 ×100
Percent increase in Nu number = 19.9474%
Similarly, Nu number at Reynolds number 3000, 5000, 7000, 9000, 11000, 13000, 15000

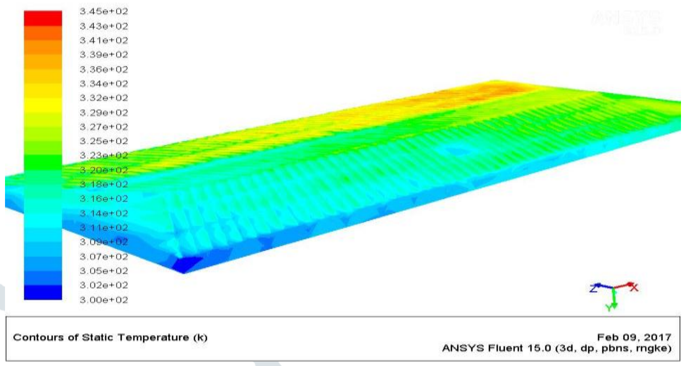

Figure 7.9 Temperature colored contour plots of air at outlet at Reynolds number 7000

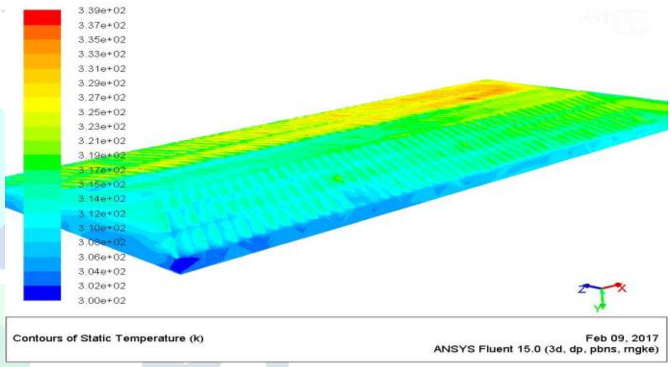

Figure 7.10 Temperature colored contour plots of air at outlet at Reynolds number 9000

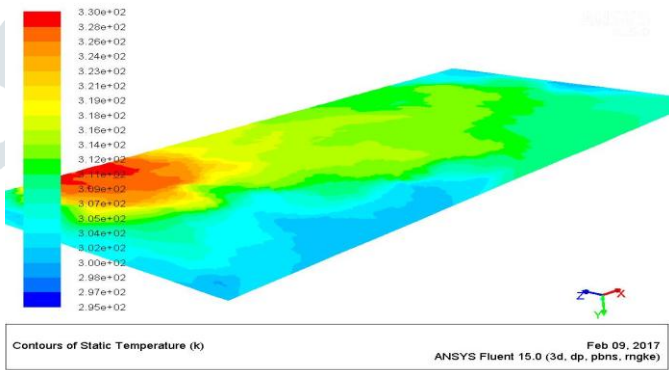

Figure 7.11 Temperature coloured contour plots of air at outlet at Reynolds number 11000

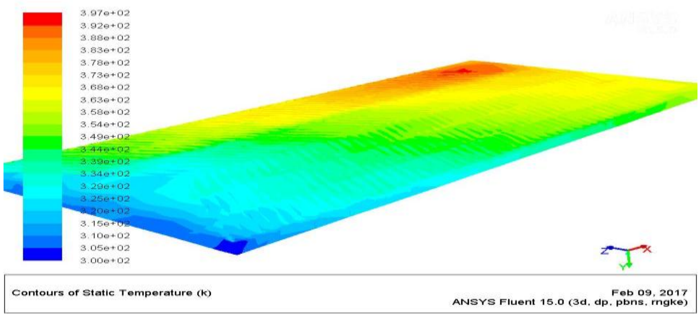

Figure 7.12 Temperature coloured contour plots of air at outlet at Reynolds number 13000

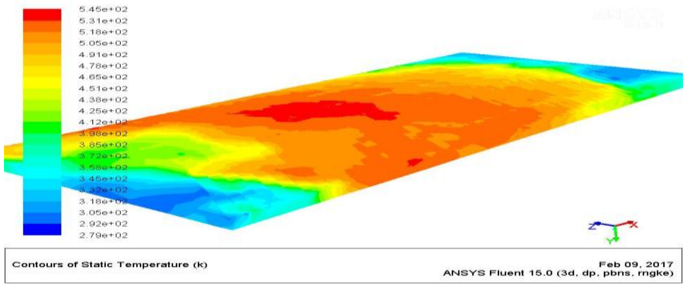

Figure 7.13 Temperature coloured contour plots of air at outlet at Reynolds number 1500

7.2 Friction factor for Reynolds number

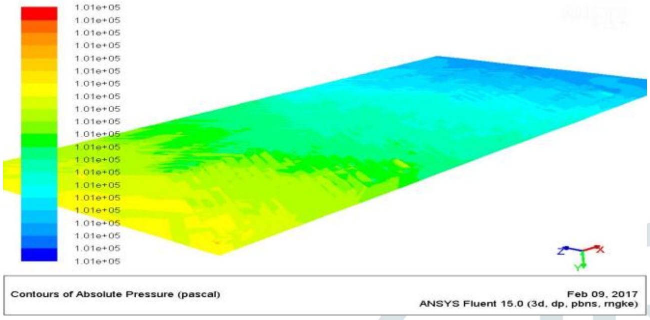

Figure 7.14 Inlet pressure coloured contour plots at 3000

From the coloured contour plots inlet pressure = 101327
From the coloured contour plots outlet pressure = 101323
Pressure difference = 101327– 101323 =4
Hydraulic diameter $D = (2\times W\times H)/(W+H) = (2\times0.3\times0.025)/(0.3+0.025) = 0.046154$
Density of air = 1.225kg/m3
Length of the test plate =1.1m
Velocity of air at Reynolds number 3000= 1.3 (calculated in the Nusselt number)
Putting the given values in the friction factor formula, we get friction factor = 0.04926 (calculated from CFD for circular rib with different relative gap width geometry)
Friction factor from the base paper is= 0.0333(for authors circular geometry)
Percent increase = (0.04926-0.0333) / 0.04926 ×100
Percent increase = 32.27784 %
Similarly, friction factor is calculated for Reynolds number 3000, 5000, 7000, 9000, 11000, 13000, 15000

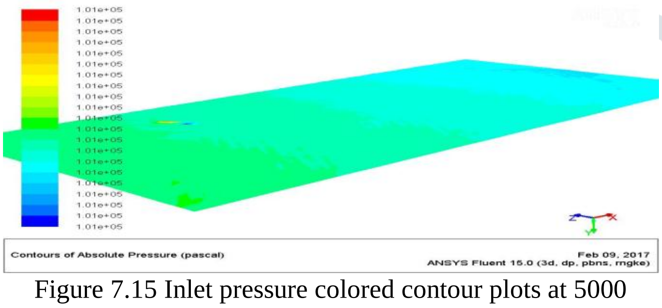

Figure 7.15 Inlet pressure colored contour plots at 5000

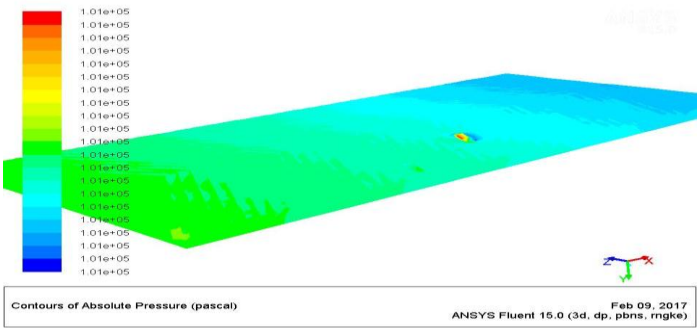

Figure 7.16 Inlet pressure colored contour plots at 7000

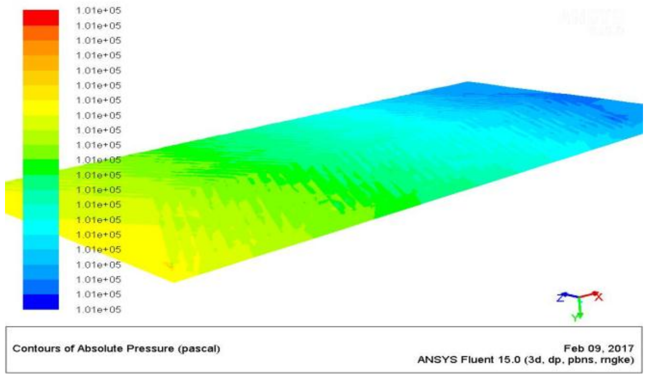

Figure 7.17 Inlet pressure coloured contour plots at 9000

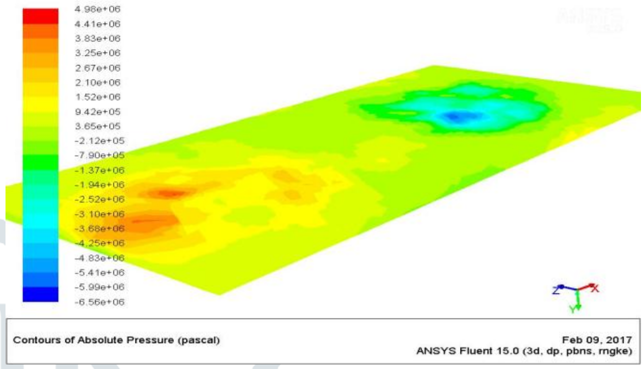

Figure 7.18 Inlet pressure coloured contour plots at 11000

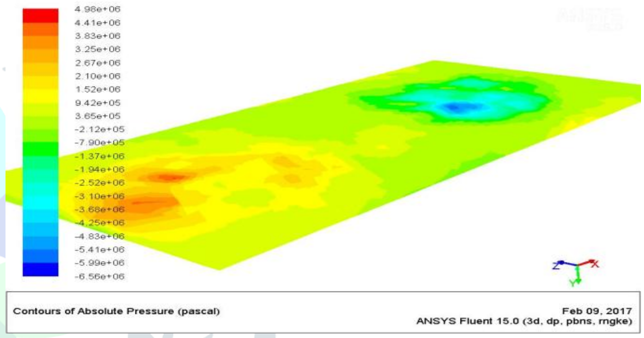

Figure 7.19 Inlet pressure coloured contour plots at 13000

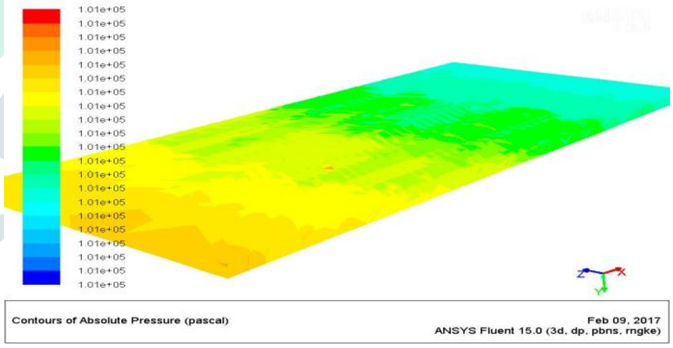

Figure 7.20 Inlet pressure coloured contour plots at 15000

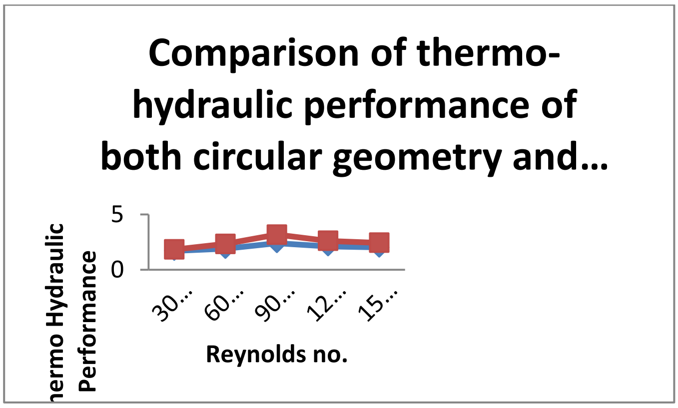

Figure 7.21 Graph of Thermo-Hydraulic performances for different circular gap width and circular geometry at different Reynolds number.

VIII CONCLUSION

8.1 CONCLUSION

- Average deviation of result obtained from CFD for smooth & circular geometry for Nu number & Friction factor lies within the range, average Nu Number is deviate 3.76% for smooth plate and Average Friction factor is deviate 3.91% for smooth plate as compared to experimental work of the Author.
- Average deviation of results obtained for different circular gap width from CFD in Nu Number is deviated by 17.01 % i.e., Nu Number increases for circular geometry at each Reynolds number taken for analysis.
- Average deviation of result obtained for different circular gap width from CFD in Friction factor is deviated by 20.15% i.e., Friction factor increases for circular geometry at each Reynolds number taken for analysis.
- Thermo-Hydraulic performance increases at Reynolds number 3000 for different circular gap width by 6.7%; and for Reynolds number 3000, 5000, 7000, 9000, 11000, 13000, 15000it increases by 18.63%, 24.12%, and 19.35%, 16. 97% respectively.
- This CFD analysis clearly indicates that different circular gap width 3roughness increases the turbulence in the air; and at the contact, the heat transferring area of air is increased which results in the increase in Nu Number and Friction factor as compared to circular geometry.

8.2 FUTURE SCOPE:-

- Simulation could be done for the solar duct of roughened circular rib with different relative gap width profiles by varying the Relative gap width or no. of gap.
- Simulation could be done for the solar duct of roughened circular rib with different relative gap width profiles by varying heat input.
- Simulation could be done for the solar duct of roughened circular rib with different relative gap width profiles by varying all three parametersi.e. relative gap width, no. of gap, roughened profile, and velocity.
- Simulation could also be done by varying Angle of Attack

REFERENCES

[1] M.M. Sahu, J.L. Bhagoria, "Augmentation of heat transfer coefficient by using 908 broken transverse ribs on absorber plate of solar air heater" Renewable Energy 30 (2005) 2057–2073

[2] Arvind Kumar , J.L. Bhagoria, R.M. Sarviya, "Heat transfer and friction correlations for artificially roughened solar air heater duct with discrete W-shaped ribs Energy Conversion and Management 50 (2009) 2106–2117.

[3] AtulLanjewar, J.L. Bhagoria, R.M. Sarviya, "Heat transfer and friction in solar air heater duct with W-shaped rib roughness on absorber plate"Energy 36 (2011) 4531-4541.

[4] A. Lanjewar, J.L. Bhagoria, R.M. Sarviya, "Experimental study of augmented heat transfer and friction in solar air heater with different orientations of W-Rib roughness Experimental Thermal and Fluid Science 35 (2011) 986–995.

[5] Anil Singh Yadav, J.L. Bhagoria, "A CFD (computational fluid dynamics) based heat transfer and fluid flow analysis of a solar air heater provided with circular transverse wire rib roughness on the absorber plate Energy 55 (2013) 1127-1142

[6] Anil SinghYadav ,J.L.Bhagoria, "Heat transfer and fluid flow analysis of solar air heater: A review of CFD approach, Renewable and Sustainable Energy Reviews 23 (2013) 60–79.

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

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

[9] S.C. Lau, R.T. Kukreja, R.D. Mcmillin, "Effects of V-shaped rib arrays on turbulent heat transfer and friction of fully developed flow in a circular rib with different relative gap width channel" Int. J.Heat Mass Transfer. Vol. 34, No. 7 .pp 1605-1616, 1991.

[10] J.C. Han and Y.M Zhang "High performance heat transfer ducts with parallel broken and V-shaped broken ribs" Int. J.Heat Mass Transfer. Vol. 35, No. 2 .pp 513-523, 1992

[11] TabishAlam, R.P.Saini, J.S.Saini "Effect of circularity of perforation holes in V-shaped blockages on heat transfer and friction characteristics of rectangular solar air heater duct" Energy Conversion and Management 86 (2014) 952–963

[12] RajendraKarwa "Experimentally investigated that heat transfer and friction factor in rectangular duct with rectangular cross-section rib on one broad wall in transverse inclined V-continues and V-discrete pattern" Int. Comm. Heat Mass Transfer Vo.30,No.2,pp.241-250,2003

[13] Dhanjay Gupta, S.C. Solanki, J.S. Saini "Thermo hydraulic Performance On Solar air Heater With Roughened Absorber Plates" Solar Energy Vol. 61, No. 1, pp. 33-42, 1997

[14] R. Karwa, S.C Solanki ,J.S Saini "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 (1999) 1597-1615

[15] P.R. Chandra, C.R Alexandra, J.C. Han "Heat transfer and friction behaviors in rectangular channels with varying number of ribbed walls" International Journal of Heat and Mass Transfer 46 (2003) 481–495

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

[17] Abdul-Malik EbrahimMomin, J.S. Saini S.C. Solanki "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 (2002) 3383–3396

[18] M.M. Sahua, J.L. Bhagoria "Augmentation of heat transfer coefficient by using 90degree broken transverse ribs on absorber plate Of solar air heater" Renewable Energy 30 (2005) 2057–2073

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

[20] RajendraKarwa, B.K. Maheswari, NitinKarwa "Experimental study of heat transfer enhancement in an Asymmetrically heated rectangular duct with perforated baffles" International Communications in Heat and Mass Transfer 32 (2005) 275–284

[21] A.R. Jaurker, J.S. Saini , B.K. Gandhi "Heat transfer and friction characteristics of rectangular Solar air heater duct using rib-grooved artificial roughness" Solar Energy 80 (2006) 895–907

[22] K.R. Aharwala, B.K. Gandhi, J.S. Saini "Experimental investigation on heat-transfer enhancement due to a gap in an inclined continuous rib arrangement in a rectangular

duct of solar air heater" Renewable Energy 33 (2008) 585–596

[23] R.P. Saini, JitendraVerma "Heat transfer and friction factor correlations for a duct having dimple-shape artificial roughness for solar air heaters" Energy 33 (2008) 1277–1287

[24] S.V. Karmare, A.N. Tikekar "Heat transfer and friction factor correlation for artificially roughened duct with metal grit ribs" International Journal of Heat and Mass Transfer 50 (2007) 4342–4351

[25] S.K. Saini a, R.P. Saini "Development of correlations for Nusselt number and friction factor for solar air heater with roughened duct having arc-shaped wire as artificial roughness" Solar Energy 82 (2008) 1118–1130

[26] Arvind Kumar, J.L. Bhagoria, R.M. Sarviya "Heat transfer and friction correlations for artificially roughened solar air heater duct with discrete W-shaped ribs" Energy Conversion and Management 50 (2009) 2106–2117

[27] Sukhmeet Singh , SubhashChandera, J.S. "Saini Heat transfer and friction factor correlations of solar air heater ducts artificially roughened with discrete V-down ribs" Energy 36 (2011) 5053-5064

[28] Rajesh Maithani , J.S. Saini "Heat transfer and friction factor correlations for a solar air heater duct roughened artificially with V-ribs with symmetrical gap" Experimental Thermal and Fluid Science 70 (2016) 220–227.

[29] Giovanni Tanda "Performance of solar air heater ducts with different types of ribs on the absorber Plate" 0.1016/j.energy.2011.08.043.

[30] Vipin B. Gawande a, A.S. Dhoble a, D.B. Zodpe a, Sunil Chamoli "Experimental and CFD investigation of convection heat transfer in solar air heater with reverse L-shaped ribs" 10.1016/j.solener.2016.02.040.

[31] Chao Ma⇑, Xiaoling Chen, Jianfei Wang, Shusheng Zang, Yongbin Ji "An experimental investigation of heat transfer characteristics for steam cooling and air cooling in a rectangular channel roughened with parallel ribs" 0.1016/j.expthermflusci.2015.02.009.

[32] Pongjet Promvonge "Thermal performance in square-duct heat exchanger with quadrupleV-finned twisted tapes" 10.1016/j.applthermaleng.2015.08.047.

[33] Bilgen E., Bakeka B.J.D., (2008), Solar collector systems to provide hot air in rural applications, RenewableEnergy, **33**, 1461-1468..