CFD Analysis of Solar Air Heater for Enhancement of Heat Transfer

Jitendra Singh[1], H. S. Sahu[2]
[1]Research Scholar, [2]Assistant Professor
[1, 2]Department of ME, MIT, Bhopal, India

Abstract - To investigate heat transfer and fluid flow behavior in a rectangular duct of a solar air heater with one roughened wall having a mixture of circular and square transverse wire rib roughness, a three-dimensional CFD analysis has been conducted. The Reynolds Effect the quantity, roughness height, pitch, relative pitch, and relative roughness Studies have been done on the effects of height on the friction and heat transfer coefficients. In order to confirm Results from the current numerical model have been contrasted with existing experimental data. under comparable flow circumstances. A medium-sized CFD investigation has been conducted. Reynolds flow of numbers (Re 143800e18, 000). As Reynolds number rises, the average Nusselt number also rises. The highest amount for relative roughness pitch of 5mm, the average Nusselt number is determined to be 43.577, and for At a higher Reynolds number of 10,000, the relative roughness height is 0.06 mm. As Reynolds number increases, the average friction factor falls. The highest amount of 1.698 is the average friction factor for relative roughness pitches of 5 mm and relative 0.06 mm of roughness height at 5000, a lower Reynolds number.
Keywords: CFD Analysis, Absorber plate, Enhancement Factor, Reynolds's No., Nusselt No

***How to cite this article**: Jitendra Singh, H. S. Sahu, "CFD Analysis of Solar Air Heater for Enhancement of Heat Transfer" Published in International Journal of Scientific Modern Research and Technology (IJSMRT), ISSN: 2582-8150, Volume-12, Issue-1, Number 3, July 2023, pp.14-18, URL: www.ijsmrt.com/wp-content/uploads/2023/08/IJSMRT-23120103.pdf*

IJSMRT-23120103

I. INTRODUCTION

Researchers are very interested in the topic of improving heat transmission since it results in energy and money savings. Due to the sharp rise in global energy consumption, it is becoming more and more important for design and operation engineers for many systems to reduce energy loss due to inefficient usage and increase energy in the form of heat. Numerous studies on improving heat transmission have been conducted during the last several decades. These studies concentrated on developing a method that would increase heat transmission while simultaneously reaching great efficiency. Various improvement approaches may increase heat transfer rates, which can lead to more compact, affordable, and thermally efficient equipment with significant energy savings. The technology for enhancing heat transfers has advanced and is now extensively employed in heat exchanger applications, including those related to refrigeration, automobiles, the process sector, and the chemical industry. Inserting various shaped objects with various geometries in channel flow is one of the extensively utilized heat transfer enhancement techniques.

The definition of energy is the capacity or aptitude to do labor. The essential component for sustaining life and growth is energy. Work include lifting, warming, and lighting various objects. The many human-made devices are powered by a variety of energy sources. Researchers are very interested in the topic of improving heat transmission since it results in energy and money savings. Due to the worldwide growth in energy demand, it has become more important than ever for design and operation engineers to reduce

ISSN: 2582-8150

energy loss due to inefficient usage and improve energy in the sense of heat for many systems. India is fortunate to have a lot of water, sunshine, and biomass. People from all walks of life are now more aware of the advantages of renewable energy, particularly decentralized energy where it is needed in villages and in urban or semi-urban areas, as a result of the vigorous efforts made over the last 20 years. The greatest renewable energy scheme in the world is in India. The Department of Non-Conventional Energy Sources (DNES) was established by the government in 1982.

II. LITERATURE SURVEY

Using computational fluid dynamics (CFD), Yadav and Bhagoria [Feb. 2022] studied heat transfer and fluid flow processes in an artificially roughened solar air heater. Investigations have been done into how tiny diameter transverse wire rib roughness affects heat transmission and fluid movement. The design factors considered are the Reynolds number, relative roughness pitches (P/e), and relative roughness heights (e/D). The ANSYS FLUENT 12.1 code is used to do a two-dimensional CFD simulation. The best suited model is determined to be the Renormalization-group (RNG) k-model. By contrasting findings with existing experimental findings, outcomes are verified.

Bhagoria and Yadav [August 2021], An efficient method for accelerating the rate of heat transmission is the application of artificial roughness in the form of repeating ribs on a surface. The features of fully developed turbulent flow in a rectangular duct with repeated transverse square sectioned rib roughness on the absorber plate have been investigated numerically.

Yadav and Thapak [May 2019], The most affordable and widely used solar energy gathering equipment is a solar air heater, which is used for drying agricultural goods, room heating, seasoning wood, and curing industrial items. An efficient method to increase the rate of heat transfer to fluid flow in the duct of a solar air heater is to utilize an artificially roughened surface. For the last thirty years, researchers have been studying the use of artificial roughness in solar air heaters. The relative effectiveness of several kinds of artificially roughened solar air heaters is investigated. This article's goal is to do such research. This article compares twenty various shapes and orientations of roughness components that are known to exist. In MATLAB-7, programs are created in order to get the results numerically.

Sarvaiya[2017], Bhagoriya, Lanjewar,conducted an experimental examination of the rectangular duct's heat transmission and friction factor properties. According to the author, thermo-hydraulic performance increased with flow angle of attack, relative roughness height, and maxima occurred at 600 degree angles of attack. A roughened duct for a solar air heater with wedge-shaped rib roughness was used in the trials to assess the effects of relative roughness pitch, relative roughness height, and wedge angle on heat transfer and friction. For the range of parameters examined, authors discovered a maximum increase of the Nusselt number up to 2.4 times and a maximum enhancement of the friction factor up to 5.3 times.

Prasad and Saini [2016] developed the formulas to get the Stanton number and average friction factor for artificially roughening absorber plates with wires of tiny diameter.
These relationships were utilized to compare the effects of the height and pitch of the roughness element on heat transmission and friction factor to experimental data that was previously available. By assuming that the cumulative shear forces from the one rough wall in a four sided rough duct and the shear forces from the three smooth walls in a four sided smooth duct are about equal, the friction factor for one side rough duct can be calculated. They made an analogy between the friction similarity law and heat momentum transfer.

III. PROBLEM IDENTIFICATION

According to the research and findings from these studies, the following issues are identified:

• To create a specimen with a thinner wall.

• It is difficult to conduct further roughness investigations below room temperature.

• To create a model on the SAH plate with continuous heat delivery.

IV. RESEARCH OBJECTIVES

The main objective of the endeavor is to decrease the consumption of diesel via mixes.

• The best form for enhancing heat transfer and fluid flow dynamics in a rectangular duct were explored.
• The friction factor, heat transfer coefficient, roughness height, and relative roughness pitch may all be evaluated to determine the effect of the Reynolds number.
• Validation of experimental results acquired in current numerical models under flow conditions similar to those of CFD analysis.
• The shape of the plate and its impact on thermal resistance and the rate of heat transfer.

V. METHODOLOGY

A simulation technique for investigating fluid flow, heat transport, and associated phenomena including chemical reactions is known as computational fluid dynamics (CFD). For the study of flow and heat transfer in this project, CFD is used. Application fields include biomedical engineering (simulating blood flow via arteries and veins), power plant combustion, chemical processes, heating/ventilation, and aerodynamic lift and drag (i.e., aircraft or windmill wings). CFD studies performed in many sectors are employed in the research and development and production of combustion engines, airplanes, and many other industrial goods.

Contrary to conventional experimental-based studies, which have a cost directly correlated with the number of configurations requested for testing, CFD allows for the production of vast numbers of data at almost no additional cost. Thus, compared to tests, parametric analyses for equipment optimization using CFD are quite affordable.

This section explains the CFD tools needed to conduct a simulation and the steps one must take to use CFD to solve an issue. The pre-processor, processor, and post-processor—three crucial components of processing CFD simulations—as well as the necessary hardware are discussed.

Three essential components are required to perform a simulation:

1. Pre-processor— A pre-processor is used to create the control volume mesh and establish the geometry for the computational domain of interest. In general, the solution is more accurate when the mesh is finer in the regions with significant changes. The required computer hardware and computation time depend on the grid's fineness. Salomé is the name of the open-source preprocessor that was utilized for this project.

2. Solver—The solver performs the computations using a methodology for numerical solution, which may include spectral, finite element, or finite difference approaches. Finite volumes, a unique finite difference technique, are used by the majority of CFD programs. The fluid flow equations are first integrated over the control volumes, resulting in the precise conservation of pertinent properties for each finite volume. These integral equations are then discretized, converting the integral fluid flow equations into algebraic equations, and the algebraic equations are then solved iteratively. (More information on the discretization methods and finite volume approach is provided in the next sections.

3. Post-Processor – The post-processor enables visualization of the outcomes and has the capacity to show geometry/mesh and produce vector, contour, and 2D and 3D surface plots. The model may be altered (i.e., scaled, rotated, etc.) and particles can be monitored during a simulation, all in full color animated visuals. The open-source post-processor utilized for this project is called Para View.

VI. RESULTS AND ANALYSIS

Following a review of the fundamental CFD procedures to be used, an analysis using real data of the flow within a duct was performed on the solar air heater. To run the simulation, the following three actions must be taken.

Preprocessing: CAD modeling, meshing, solver type, physical model, material property, boundary condition, and six more concepts are discussed here.

For analysis of solar air heaters, a rectangular duct-shaped 3-dimensional model is created. Pre-processor is used to construct the model geometry. Through the use of the pre-processor ANSYS DESIGN MODELER, the model geometry will be produced.

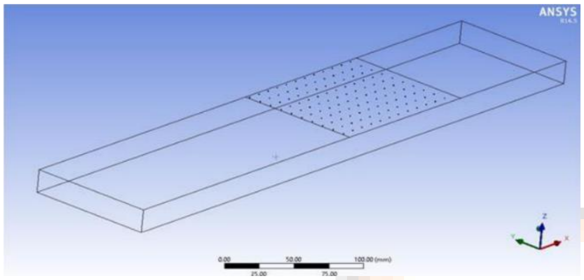

Figure 1: 3-D Domain Sah Duct with Combination of Circular and Square RIB with E =0.5 MM

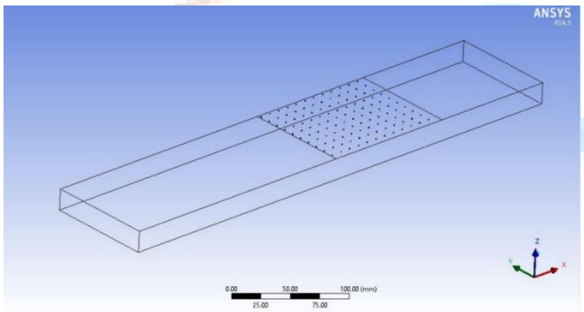

Figure 2: 3-D Domain Sah Duct with Combination of Circular and Square RIB with E =0.75MM

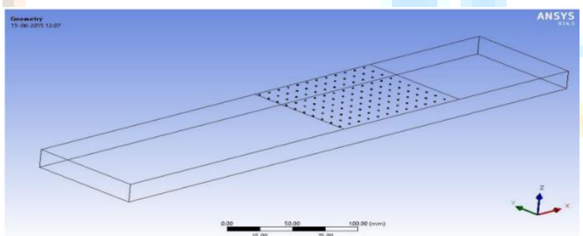

Figure 3: 3-D Domain Sah Duct with Combination of Circular and Square RIB with E =1.00MM

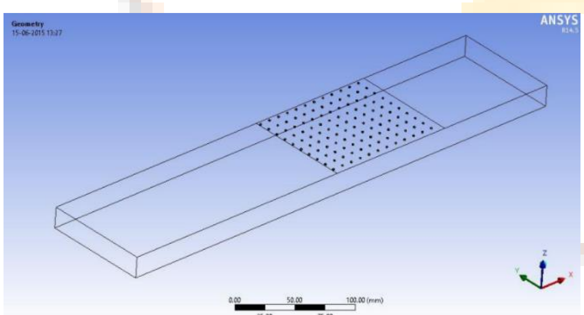

Figure 4: 3-D Domain Sah Duct with Combination of Circular and Square RIB with E =1.25 MM

VII. CONCLUSIONS

The buffer plate designed geometry was defined with the prior results in terms of REYNOLD NUMBERS RANGE 3800-18000 AND AFTER results is same, RIB ROUGHNESS ON THE ABSORBER PLATE before is transverse square sectioned rib and after result is circular and square section transverse rib, TRANSVERSE RIB ROUGHENED DUCT WITH P/E 14 (value must be minimum for better thermal enhancement factor) is 10.71 and after result is 5, Additional findings from the current investigation include the ones below:

1. As Reynolds increases, the average Nusselt number rises. At a higher Reynolds number of 10,000, the highest value of the average Nusselt number is determined to be 43.577 for relative roughness pitch of 5 and relative roughness height of 0.06.

2. As the Reynolds number rises, the average friction factor falls. At a lower Reynolds number, 5000, the highest value of average friction factor is determined to be 1.698 for relative roughness pitch of 5 and relative roughness height of 0.06.

In order to analyze heat transfer and fluid flow behavior in a rectangular duct of a solar air heater with one roughened wall having a mix of circular and square transverse wire rib roughness, a three-dimensional computational fluid dynamics (CFD) analysis was conducted. The heat transfer coefficient and friction factor have been examined in relation to Reynolds number, roughness height, roughness pitch, relative roughness pitch, and relative roughness height. findings have been compared with accessible experimental findings obtained under comparable flow circumstances in order to verify the current numerical model. Medium Reynolds number flow Re (¼)3800e18, 000 has been used for the CFD investigation.

REFERENCES

1. Yadav A.S, Bhagoria J.L, A CFD 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 (2022) 1127–1142

2. Yadav A.S, Bhagoria J.L, Modeling and simulation of turbulent flows through a solar air heater having square sectioned transverse rib roughness on the absorber plate, Sci. World J. 2013 (2021).

3. Lanjewar A, Bhagoria J.L, Sarvaiya R.M "Augmentation of heat transfer coefficient by using 900 broken transverse ribs on absorber plate of solar air heater". Renewable Energy 36 (2011) 4531–4541. (2020).

4. Prasad K and Saini J.S, A experimental investigation in transverse ribs on the absorber plate by assuming that the total shear force, Heat and mass transfer 40 976-993 (2015).

5. Karwa R, Solanki S.C, Saini S.C, Thermo-hydraulic performance of solar air heaters having integral chamfered rib roughness on absorber plates 26 (2001) 161–176 (2014)

6. Bhagoriya J.L, Lanjewar A, Heat transfer coefficient and friction factor correlations for rectangular solar air heater duct having transverse wedge shaped rib roughness on the absorber plate, Renewable Energy 25 (2013) 341– 369.

7. Prasad K, Mullick S.C, Heat transfer characteristics of a solar air heater used for drying purposes, Appl. Energy 13 (2) (2013) 83–93.

8. Saini R.P, Verma J, An experimental setup on the fluid flow and heat transfer of solar air heater duct having dimple –shaped artificial roughness, Energy 33 (2012) 1277–1287.

9. Verma S.K, Prasad B.N, Investigation for the optimal thermo-hydraulic performance of artificially roughened solar air heaters, Renewable Energy 20 (2012) 19–36.

10. Bopche S.B, Tandale M.S, Experimental investigations on heat transfer and frictional characteristics of a turbulator roughened solar air heater duct, Int. J. Heat Mass Transfer 52 (2011) 2834–2848.