Engineering Universe for scientific Research and management
ISSN (Online): 2319-3069
Vol. 11 Issue 3
March 2019

CFD INVESTIGATION ON RECTANGULAR DUCT WITH ARC SHAPED RIB WITH GROOVE'S ON THE ABSORBER PLATE

Pritesh Kharate[1], Sanjay Kalraiya[2], Sachin Baraskar[3]
[1,2,3] Department Mechanical Engineering, SSSUTMS, Sehore, India
pritesh.kharate27@gmail.com

ABSTRACT

Solar air heater is an important device to convert solar energy into heat energy economically. A CFD analysis is conducted through different turbulence models to study the performance of a solar air heater using absorber plate. A modern CFD code ANSYS FLUENT v.14.5 is used to simulate fluid flow and heat transfer through the solar air heater. A numerical analysis of convective heat transfer enhancement in solar air heater with artificially roughened absorber is presented in this paper. CFD numerical simulation were carried out to analyze the flow and heat transfer in the air duct of a solar air heater provided with arc shape with grooves ribs. The air flow in forced convection and the absorber is heater with uniform flux. Since the flow is cerate turbulent, we used the Reynolds-averaged navir – stokes RANS formulation to model the flow. We solved momentum and energy, continuity equation used Finite volume method, finite element method, finite difference method also in turbulent mode using closure models: k- ε RNG, k- ε RZ, k- ω standard k- ω SST. A two – dimensional non uniform grid was generated according to used geometry .with local refining near the wall. In order to critically examine the flow and heat transfer in the inter-rib regine. The effect of major parameter (Reynolds number, Nusselt number, friction factor, global thermo hydraulic performance parameter etc) is examined and discussed.

Keywords - *CFD Analysis, Absorber plate, Solar air heaters, Nusselt number, Reynolds number.*

1. INTRODUCTION

The thermal efficiency of solar air heaters has been found to be generally poor because of their inherently low heat transfer capability between the absorber plate and air flowing in the duct. In order to make the solar air heaters economically viable, their thermal efficiency needs to be improved by enhancing the heat transfer coefficient. In order to attain higher heat transfer coefficient, the laminar sub-layer formed in the vicinity of the absorber plate must be broken and the flow at the heat-transferring surface is made turbulent by introducing artificial roughness on the surface. However, the artificial roughness results in higher frictional losses leading to excessive power requirement for the fluid to flow through the duct. It is, therefore, desirable that turbulence must be created only in a region very close to the heat-transferring surface to break the viscous sub-layer for augmenting the heat transfer, and the core flow should not be unduly disturbed to limit the increase in friction losses. This can be done by keeping the height of the roughness elements small in comparison to the duct dimensions. Various investigators have studied different types of roughness geometries and their arrangements to enhance the heat transfer from heat transferring surfaces. Solar air heaters form the major component of solar energy utilization system which absorbs the incoming solar radiation. A large number of research investigations have been undertaken both theoretically and experimentally to enhance the thermal performance of flat plate solar air heaters.

Computational fluid dynamics is a computer based simulation process for analyzing fluid flow, heat transfer, and related phenomenon such as chemical reactions. This effort uses CFD for analysis of flow and heat transfer. Some examples of application areas are: aerodynamic lift and drag (i.e. windmill wings or airplanes), power plant combustion, chemical processes, heating/ventilation, and even biomedical engineering (simulating blood flow through arteries and veins). CFD analyses carried out in the different industries are used in R&D and manufacture of aircraft, combustion engines, as well as various other industrial products. It can be beneficial to use

Computational fluid dynamics over usual experimental based analyses, since experiments have a expenditure directly proportional to the number of configurations desired for testing, unlike with Computational fluid dynamics, where huge amounts of results can be produced at practically no added cost. In this way, parametric studies to optimize equipment are extremely inexpensive with Computational fluid dynamics when compared to experiments. This segment briefly describes the common concepts and theory related to using Computational fluid dynamics to analyses heat transfer and fluid flow, as relevant to this project. It begins with a evaluate of the tools needed for carrying out the Computational fluid dynamics analyses and the processes required, followed by a summary of the governing equations and turbulence models and a discussion of the discretization schemes and explanation algorithms is presented.

2. LITERATURE REVIEW

K R Aharwal et.al The effect of geometrical parameters, especially, the gap width and gap position has been investigated. The roughened duct has a width to height ratio (W/H) of 5.83. The relative gap position (d/W) and relative gap width (g/e) has been varied from 0.16 to 0.5 and 0.5–2.0, respectively. Experiments have been carried out for the range of Reynolds number from 3000 to 18,000 with the relative roughness pitch (P/e) range of 4–10; relative roughness height (e/D) range of 0.018–0.037; and angle of attack (α) range of 30–90°. The optimum values of parameters for rib arrangement have been obtained and discussed. For Nusselt number, the maximum enhancement of the order of 2.83 times of the corresponding value of the smooth duct has been obtained, however, the friction factor has also been seen to increase by 3.60 times of that of the smooth duct. The maximum enhancement is observed at a relative gap position of 0.25 for relative gap width of 1.0, relative roughness pitch of 8.0, angle of attack of 60° and relative roughness height of 0.037. Based on the experimental data, correlations for Nusselt number and friction factor have been developed as function of roughness parameters of inclined discrete square ribs and flow Reynolds number.

Bhagoria et al. (2002) have experimentally shown that as compared to the smooth duct, the presence of ribs yields Nusselt number up to 2.4 times while the friction factor rises up to 5.3 times.

Karim et al (2006) from their experiments found that the v-corrugated collector has better thermal performance than the flat plate collector. Lin et al (2006) found that cross corrugated solar air-heaters has a much superior thermal performance to that of the flat-plate collector.

Jaurker et al. (2006) experimentally showed that the heat transfer coefficient for rib-grooved arrangement is higher than that for the transverse ribs. Gao et al (2007) experimentally found that the cross-corrugated collectors have higher thermal performance as compared to flat plate collectors due to higher surface area for heat absorption. Mittal et al. (2007) found that solar air heater having inclined ribs as roughness elements is found to have better effective efficiency in the higher range of Reynolds number.

Sachin Baraskar et al heat transfer and friction factor characteristics of a rectangular duct roughened with repeated v-shape ribs with and without gap on one broad wall arranged at an inclination of 60° with respect to the flow direction. A rectangular duct of aspect ratio of (W/H) of 8, relative roughness pitch (p/e) of 10, relative roughness height (e/Dh) of 0.030, and angle of attack 60. The heat transfer and friction characteristics of this roughened duct have been compared with those of the smooth duct under similar flow condition. The effect of gap in v shaped rib has been investigation for the range of flow Reynolds numbers from 5000 to 14000. The maximum enhancement in Nusselt number and friction factor is observed to be 2.57 and 2.85 time of that of the smooth duct.

Yadaba Mahanand et.al The 2D heat transfer and fluid flow phenomenon through an artificially roughened solar air heater is investigated by means of a numerical model at a constant heat flux of 1000 W/m^2. A modern CFD code ANSYS FLUENT v 14.5 is used to simulate fluid flow and heat transfer through the solar air heater. The duct wall, Absorber plate and roughness materials are assumed to be homogeneous & isotropic and also the thermal conductivity is independent of temperature. The present work show that the Renormalization-group k-epsilon model provides the results close to those, worked out from available empirical co-relation for two-dimensional steady flow solar air heaters. The maximum enhancement of average Nusselt number has been found to be 2.3104 times that of smooth duct for relative roughness pitch of 7.14 and for relative roughness height of 0.042.

Vishavjeet et al. (2009) concluded that using computational fluid dynamics (CFD) models, analysis of heat transfer and flow characteristics of roughened solar air heaters needs to be carried out to predict optimum roughness element parameters.

Atul et al. (2012) conducted an experimental investigation of heat transfer and friction factor characteristics of rectangular duct roughened with W-shaped ribs. From the above literature review it is

clear that a CFD analysis has not been explored to find the efficacy of using surface corrugations for improving the thermal efficiency of solar air heaters.
Vishavjeet et al. (2009) in their review paper have stressed the need for a CFD analysis to understand the flow phenomena and thermal behavior of air heater and hence in the present study, CFD analysis has been carried out to understand the heat transfer mechanism of a solar open collector.

3. COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics (CFD) is the use of applied mathematics, physics and computational software to visualize how a gas or liquid flows -- as well as how the gas or liquid affects objects as it flows past. Computational fluid dynamics is based on the Navier-Stokes equations. These equations describe how the velocity, pressure, temperature, and density of a moving fluid are related. Computational fluid dynamics has been around since the early 20th century and many people are familiar with it as a tool for analyzing air flow around cars and aircraft. As the cooling infrastructure of server rooms has increased in complexity, CFD has also become a useful tool in the data center for analyzing thermal properties and modeling air flow. CFD software requires information about the size, content and layout of the data center. It uses this information to create a 3D mathematical model on a grid that can be rotated and viewed from different angles. CFD modeling can help an administrator identify hot spots and learn where cold air is being wasted or air is mixing. Simply by changing variables, the administrator can visualize how cold air will flow through the data center under a number of different circumstances. This knowledge can help the administrator optimize the efficiency of an existing cooling infrastructure and predict the effectiveness of a particular layout of IT equipment. For example, if an administrator wanted to take one rack of hard drive storage and split the hard drives over two racks, a CFD program could simulate the change and help the administrator understand what adjustments would be need to be made to deal with the additional heat load before any time or money has been spent.

About Flow: The physical discipline of fluid dynamics evolved as the sciences started to classify the natural power and associated reaction of air, water or gases. This provided a systematic structure that embraced empirical laws and was derived from the idea of flow measurement that is used to solve practical problems. A typical fluid dynamics problem involves basic fluid properties like flow velocity, pressure, density, and temperature, in relation to time and space. In everyday life, we can find fluid flows in meteorology (rain, wind, floods, hurricanes), heating, ventilation and air conditioning, aerodynamic design, engines combustion, industrial processes, or the human body—for example, blood flow—and so on. Fluid dynamics has a wide range of applications, including calculating forces on aircraft, determining the mass flow rate of petroleum through pipelines, and predicting weather patterns. The development of CFD has been closely associated with the evolution of high-speed computers.

Importance of CFD Analysis: With a CFD analysis, we can understand the flow and heat transfer throughout a design process. The basic methodology for any engineering CFD analysis is based on a few procedures:

- Understanding flow model - Flow separations, transient effect, physical interactions
- Proving assumed model - Experimental results validation, parametric studies, structural simulations
- Model optimizing - reducing pressure drops, flow homogenization, improving laminar and turbulent mixing.
- Without numerical simulations of fluid flow, it is very difficult to imagine how Meteorologists can forecast the weather and warn of natural disasters.
- Vehicle designers can improve aerodynamic characteristics
- Architects can design energy-saving and safe-living environments
- Oil and gas engineers can design and maintain optimal pipes networks;
- Doctors can prevent and cure arterial diseases by computational hemodynamic.

CFD Computational Tools: This section describes the Computational fluid dynamics apparatus required for carrying out a simulation and the method one follows in order to solve a problem using CFD. The hardware required and the three main elements of processing Computational fluid dynamics simulation: the pre-processor, processor, and post-processor are described. There is a assortment of commercial Computational fluid dynamics software are available such as ANSYS CFX, Fluent, ACE, as well as a broad range of appropriate hardware and associated expenses, depending on the difficulty of the mesh and size of the calculations. Commercial Computational fluid dynamics packages can cost up to about $20000 per year for license, protection, and support. Complicated transient cases with fine meshes will require extra powerful computer processors and

RAM than simpler cases with rough meshes. A typical engineering workstation (i.e. 32 GB processing RAM with quad processors) at a cost of US Dollars approximately $3000-$5000, or a combination of a number of processors running in equivalent, is probably the minimum investment needed to get started. The effort for this assignment was carried out on a HP Pavilion laptop with dual processors totaling 2 GHz RAM, operation on Linux Operating System downloaded free from Caelinux. The download from Caelinux included open source software Salomé for geometry construction and meshing, open FOAM for the Computational fluid dynamics calculations, preview for hallucination of results, along with other useful scientific and mathematics related software. Calculations for this assignment were carried out for approximately 50,000 cells (CFD calculations are often made for one to two million cells or more). On my system, the steady state solvers took between one to three hours required to end calculations, while the transient simulation took two to three days running in corresponding on both processors. One of the purposes of this assignment is to use all open source Computational fluid dynamics software in its place of commercial software for the simulations. This type of software is beneficial for smaller companies to use, as the rate of commercial CFD package licenses can be unaffordable.

CFD Algorithm

To run a simulation, three main elements are needed:

(a) **Pre-Processor:** A pre-processor is used to identify the geometry for the computational domain of concentration and produce the mesh of control volumes (for calculations). Normally, the finer the mesh in the areas of large changes, the more correct the solution. Fine quality of the grid also determines the computer hardware and calculation time needed. The open source pre processor used for this assignment is called Salomé.

(b) **Solver:** The solver makes the calculations using a mathematical solution technique, which can use finite diversity, finite element, or spectral methods. Mainly Computational fluid dynamics codes use finite volumes, which is a unique finite difference method. Primary the fluid flow equations are integrated more than the control volumes, then these significant equations are discredited (producing algebraic equations through converting of the integral fluid flow equations), and finally an iterative method is used to solve the algebraic equations. Open FOAM Computational fluid dynamics code is used for solving the simulation in this task.

(c) **Post-Processor:** The post processor provides for hallucination of the results, and includes the capability to display the geometry/mesh, create vector, contour, and 2Dimention and 3Dimension surface plots. Throughout a simulation, Particles can be tracked and the model can be manipulated and all in full color animated graphics. Preview is the open source post processor used for this assignment.

Basic Steps to Perform CFD Analysis

1. Preprocessing
- CAD Modeling: Creation of CAD Model by using CAD modeling tools for creating the geometry of the part assembly of which you want to perform FEA. CAD model may be 2D or 3D.
- Meshing:-Meshing is a critical operation in computational fluid dynamic. In this operation, the CAD geometry is discretized into large numbers of small Element and nodes. The arrangement of nodes and element in space in a proper manner is called mesh. The analysis accuracy and duration depends on the mesh size and orientations. With the increase in mesh size (increasing no. of element), the CFD analysis speed is decreased but the accuracy increased.
- Type of Solver: Choose the solver for the problem from Pressure, based and density based solver
- Physical model: Choose the required physical model for the problem i.e. laminar, turbulent, energy, multiphase, etc.
- Material Property: Choose the Material property of flowing fluid.
- Boundary Condition: Define the desired boundary condition for the problem i.e. velocity, mass flow rate, temperature, heat flux etc.

2. Solution
- Solution Method: Choose the Solution method to solve the problem i.e. First order, second order.
- Solution Initialization:- Initialized the solution to get the initial solution for the problem.
- Run Solution: Run the solution by giving no of iteration for solution to converge.

3. Post processing
- Post processing for viewing and interpretation of result. The result can be viewed in various formats: graph, value, animation etc.

ANSYS Fluent Software

Fluent is one of the widely used software package. It contains the wide range of physical modeling capabilities which are needed to model flow, turbulence and heat transfer for industrial application.

Features of ANSYS Fluent software are:-

- ✓ **Turbulence:** ANSYS Fluent offers a number of turbulence model to study the effect of turbulence in a wide range of flow regimes.
- ✓ **Mesh flexibility:** ANSYS Fluent software provides mesh flexibility. It has the ability to solve flow problems using unstructured meshes. Mesh types that are supported in fluent includes triangular, quadrilateral, tetrahedral, hexahedral, pyramid, prism (wedge) and polyhedral. The techniques which are used to create polyhedral meshes save time due to its automatic nature. A polyhedral mesh contains fewer cells than the corresponding tetrahedral mesh. Hence convergence is faster in case of polyhedral mesh.
- ✓ **Dynamic and Moving mesh:** The user set up the initial mesh and instructs the motion, while fluent software automatically change the mesh to follow the motion instructed.
- ✓ **Post–Processing and Data export:** User can post –process their data in fluent software, creating among other things contours, path lines, and vectors to display the data.

4. SETUP DETAILS OF CFD

In ANSYS FLUENT 14.5, Design of artificially roughness duct is done using the work-bench. The details of the set-up, procedure result and the method used to develop suitable correlations are described in the following section.

There are numerous decisions to be completed before setting up the problem in the Computational fluid dynamics code. a few of the decisions to be made can include: whether the problem should be 2-D or 3-D , which type of boundary conditions to use, whether or not to estimate pressure/temperature variations based on the air flow density, 29 which turbulence model to use, etc. The assumptions made should be reduced to a level as easy as possible, yet still retaining the largest part significant features of the difficulty to be solved in order to reach an accurate explanation.
Once the above decisions are made, the geometry and mesh can be produced. The grid should be made as fine as required to make the simulation 'grid independent'. To check it out the fineness required, a grid reliance revise is normally accepted by making a series of refinements on an primarily course grid, and carrying out simulations on each to determine when the key results of attention don't change, at which point the grid is measured free. In this assignment, a grid with approximately 50,000 cells was chosen once carrying out such a revision. To reach a converged solution, relaxation factors and acceleration devices can be selected. In this assignment, relaxation factors for all the parameters to be solved and the GAMG smooth solver for pressure were used to sustain in union and speed optimization. Lastly, to make sure accuracy of the simulations, they should be validated against tentative data. This project's simulation results are compared to an tentative study reported in the literature .Details of the heat exchanger geometry, initial boundary conditions related to flow and temperature were followed as closely as possible when building this Computational fluid dynamics model.

Nomenclature:

D	Hydraulic diameter of duct, mm
Ph	Wetted perimeter, mm
A	Cross-sectional area, m2
h	Heat transfer coefficient, W/m2K
k	Thermal conductivity of air, W/mK
m	Mass flow rate, kg/s
f	Friction facto
Nu	Nusselt number
Pr	Prandtl number
Re	Reynolds number
W	Width of duct, mm
H	Depth of duct, mm
e	Rib height, mm
P	Rib Pitch, mm
W/H	Duct aspect ratio

PARAMETER

Range of parameter for heat transfer and friction factor studies:

S.N.	Roughness and flow parameter	Range of parameter
1	Reynold number (Re)	3000-18000 (4 values)
2	Relative roughness height (e/Dh)	0.045
3	Relative roughness pitch (P/e)	8.0
4	Relative gap position (d/W)	0.30 to 0.55 (3 values)
5	Relative gap width (g/e)	1.5
6	Relative of attack (α)	50^{o} - 60^{o}
7	Aspect Ratio	8.0

5. CONCLUSION

In the present work, the study of enhancement of heat transfer is performed by using the ribbed are shaped wire as artificial roughness in the solar air heater. A numerical investigation has been carried out to generate data on Nusselt number and friction factor that that can be utilized to develop heat transfer coefficient and pressure loss correlation respectively. It is proposed to collect data on Nussel number and friction factor as function of roughness parameter like (relative roughness height), relative roughness pitch ,aspect ratio of duct and Reynolds number of flow.

REFERENCES

[1] K R Aharwal, 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", Renew Energy, Vol 33, 2008, pp. 585–96.

[2] S. Singh, S. Chander, J.S. Saini, "Heat transfer and friction factor correlations of solar air heater ducts artificially roughened with discrete V-down ribs", Journal of Renewable and Sustainable Energy, Vol. 3(1), 2011

[3] Sachin Baraskar, K.R.Aharwal, A.Lanjewar, "Experimental Investigation of Heat Transfer and Friction Factor of V-shaped Rib Roughed Duct with and without Gap" International Journal of Engineering Research and Applications, Vol. 2(6), pp. 1024-1031

[4] RP Saini, J. Verma, "Heat transfer and friction factor correlations for a duct having dimple-shaped artificial roughness for solar air heaters". Journal of Energy, Vol. 133, 2008, pp. 1277–87.

[5] V.S. Hans, R.P. Saini, J.S. Saini, "Heat transfer and friction factor correlations for a solar air heater duct roughened artificially with multiple V-ribs", Solar energy, Vol. 84(6), 2010, pp. 898-911

[6] A. Kumar, R.P. Saini, J.S. Saini, "Development of correlations for Nusselt number and friction factor for solar air heater with roughened duct having multi v-shaped with gap rib as artificial roughness" Renew Energy, Vol. 58, 2013, pp. 151-163

[7] A K Patil, J S Saini, K Kumar, "Nusselt number and friction factor correlations for solar air heater duct with broken V-down ribs combined with staggered rib roughness", J Renew Sustain Energy, Vol. 4, 2012,

[8] A K Patil, J S Saini, K. Kumar, "Heat transfer and friction characteristics of solar air heater duct roughened by broken V-shape ribs combined with staggered rib pieces", J Renew Sustain Energy, Vol. 4(1), 2011

[9] A Layek, J S Saini, S C Solanki, "Heat transfer coefficient and friction characteristics of rectangular solar air heater duct usin grib-grooved artificial roughness", Int J Heat Mass Transf, Vol. 50, 2007, pp. 4845–54.

[10] 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", Sol Energy, Vol 80, 2006, pp. 895–907.