Mechanics and Mechanical Engineering
Vol. 20, No. 4 (2016) 359–370

Performance Evaluation of Multipurpose Solar Heating System

R. Venkatesh
Department of Mechanical Engineering
Srinivasan Engineering College
Perambalur, Tamilnadu, India

V. Vijayan
Department of Mechanical Engineering
K. Ramakrishnan College of Technology
Samayapuram, Trichy, Tamilnadu, India

Received (29 May 2016)
Revised (26 July 2016)
Accepted (25 September 2016)

In order to increase the heat transfer and thermal performance of solar collectors, a multipurpose solar collector is designed and investigated experimentally by combining the solar water collector and solar air collector. In this design, the storage tank of the conventional solar water collector is modified as riser tubes and header and is fitted in the bottom of the solar air heater. This paper presents the study of fluid flow and heat transfer in a multipurpose solar air heater by using Computational Fluid Dynamics (CFD) which reduces time and cost. The result reveals that in the multipurpose solar air heater at load condition, for flow rate of 0.0176 $m^3/s\ m^2$, the maximum average thermal efficiency was 73.06% for summer and 67.15 % for winter season. In multipurpose solar air heating system, the simulated results are compared to experimental values and the deviation falls within ± 11.61% for summer season and ± 10.64% for winter season. It proves that the simulated (CFD) results falls within the acceptable limits.

Keywords: solar water collector, solar air collector, fluid flow, heat transfer and Computational fluid dynamics.

1. Introduction

Solar energy plays an important role in low–temperature thermal applications, since it replaces a considerable amount of conventional fuel. Flat plate collectors are, therefore, the best candidates for solar heating system. The total system efficiency is based on the performance of flat plate collectors. Hence most of the research work so far has been focused on the performance improvement of collector area K.S.Ong (1995).

The absorber plate efficiency, selective coatings, various design of absorber plate, thermal losses and insulation, effect of tilt angle, various working fluids and characteristics of thermosyphon system which have been analyzed by many researchers Whiller A et al (1965).

To enhance the performance of solar air heaters, efforts have been made to reduce top heat losses from absorber, increase heat transfer coefficient and increase contact area between the absorber plate and the air stream. In order to achieve this, different modifications have been suggested by many researchers R.S. Gill et al (2012).

A computational analysis of heat transfer augmentation and flow characteristic due to rib roughness over the absorber plate of solar air heaters were presented by Chaube et al (2006). Sahu & Bhagoria (2005) investigated experimentally the heat transfer coefficient by using 90° broken transverse ribs on the absorber plate of a solar air heater. They concluded that the roughened absorber plates increase the heat transfer coefficient 1.25–1.4 times as compared to smooth rectangular duct under similar operating conditions at higher Reynolds number.

The heat transfer and friction characteristics of rectangular solar air heater duct using rib-grooved artificial roughness was studied by Jaurker et al (2006). They inferred that as comparison to the smooth duct, the presence of rib-grooved artificial roughness yields Nusselt number up to 2.7 times while the friction factor rises up to 3.6 times in the range of the parameters investigated.The performance of solar air heaters having v-down discrete rib roughness on the absorber plate was investigated by Karwa & Chauhan (2010). Analysis of fluid flow and heat transfer in a rib grit roughened surface solar air heater was presented by Karmare & Tikekar (2010). The previous studies on rib roughness over the absorber plate of the solar air heaters indicated that the artificial roughness results in the desirable increase in the heat transfer rate with the penalty of the undesirable increase in the pressure drop due to the increased friction.

This paper presents the study of fluid flow and heat transfer in a multipurpose solar air heater by using Computational Fluid Dynamics (CFD).In the present work, multipurpose solar collector is designed by adding the solar water heater and the solar air heater. The storage tank of the conventional solar water collector is modified as riser tubes and header. It is fitted in the bottom of the solar air heater as an absorber in the normal air heater. Heat energy stored in water from the solar water heater is being sent into the solar air heater. The heat energy absorbed in the air heater is also added into the previous heated water to increase the energy content of the system.

2. Theory

Computational fluid dynamics (CFD) is concerned with the efficient numerical solution of the partial differential equations that describe fluid dynamics. Dynamics of fluids are governed by coupled nonlinear partial differential equations, which are derived from the basic physical laws of conservation of mass, momentum and energy. In general, most of the engineering problems are governed by non–linear partial differential equations for which the analytical solutions are possible only if partial differential equations are converted into linear form. Hence, Anderson et al

(1984) obtaining the solutions of partial differential equations by analyzed numerical methods.

CFD techniques are used in many areas of engineering where fluid behavior is the main element. Numerical analysis is applied to fluid flow and heat transfer problems.

All CFD codes contain three main elements:

1. A pre–processor which is used to input the problem geometry, generate the grid and define the flow parameters and the boundary conditions to the code.
2. A flow solver which is used to solve the governing equations of the flow subjects to the conditions provided.

   There are four different methods used as a flow solver:

   (a) Finite difference method,
   (b) Finite element method,
   (c) Finite volume method,
   (d) The spectral method.
3. A post–processor is used to display the data and show the results in graphical way in order to be easy to read format.

In all these approaches, the following basic procedure is followed

- The geometry of the problem is defined.
- The space occupied by the fluid is divided into discrete cells, known as mesh.
- Boundary conditions are defined. This involves specifying the fluid behavior and properties at the boundaries of the problem.
- The equations are solved iteratively.
- Analysis or visualization of the solution.

2.1. *Governing equations*

The governing equations of fluid flow represent mathematical statements of conservation laws of physics. The following equations are solved by Versteeg & Malalasekera (1995) and Ferziger & Peric (2002), which subjects to the boundary conditions of respective problems to get the solution of that problem.

1. Continuity equation:

$$\frac{\partial}{\partial t}(\rho m_k) + \nabla(\rho U m_k) = -\nabla.(j_k) + S_k \tag{1}$$

2. Momentum equation:

$$\frac{\partial}{\partial t}(\rho U) + \nabla.(\rho UU) = -\nabla.\pi + \rho g + F \tag{2}$$

3. Energy equation:

$$\frac{\partial}{\partial t}(\rho h) + \nabla.(\rho U h) = -\nabla.(q) + \frac{Dp}{Dt}(\tau : \nabla U)\nabla\left[\sum_k h_k j_k\right] + S_h \qquad (3)$$

3. Experimentation

In a multipurpose solar heating system, solar water heater and air heater are combined together, and act as a multipurpose solar air heater by closing the valve V_1 & V_4. Initially, the valve V_1 is opened and V_4 is closed to fill the riser tubes of both water and air heater with cold water every morning before commencement of the experiment. The period of testing for each run was between 08:30 am and 04:30 pm. No water was withdrawn from the storage during the experiment. Experiments were conducted for several days of summer and winter.

The heat energy is gained by the water heater which is stored in the riser tubes and is transferred to the air available in the air heater. The air available in the air heater will absorb the heat from the solar radiation that directly falls on it and is reheated by the energy stored in the riser tubes. The experiment was conducted at no load and with load on different days in order to analyze the thermal performance and to compare the performance of this MPSAH with that of the conventional solar air heater. The characteristics of this system are described below.

4. CFD modelling

A three–dimensional numerical model was developed using the CFD numerical package, FLUENT which is based on the control volume method. An experimental model was used to evaluate the flow patterns. Flow visualization was used to investigate the flow structure. Simulation was performed in Cosmos Express commercial CFD software. The inlet parameters from the experimental studies were used as the input data for simulation. The output parameters from the simulation studies were compared with the calculated output parameters for the different set of conditions obtained by changing variable parameters like mass flow rate of air and water, inlet temperature and intensity of solar radiation.

4.1. *Computational domain*

The 3–D computational domain of the solar air heater, with height (H) of 200 mm, width (W) 1035 mm and total length of 1035 mm, used for CFD analysis is in Fig. 1.

The generations of the geometrical parameters of multipurpose solar water and air heater for computational analysis are shown in Tab. 1.

These are created by using the method of volume splitting with face. This splitting of volume is necessary in order to create meshes with less number of elemental volumes to enable the defined problem to be solved in solver. After creating the geometry, boundary layer is created in the entrance and exit faces of the riser tube. This is necessary in order to incorporate the wall effect on the flowing fluid. Turbulence model used in this model was k-epsilon, it has more significance than the other models like k-omega, spalartallmarasand large eddy current models.

Table 1 Parameters of the multipurpose solar collector for computational analysis

Component	**MPSWH**	**MPSAH**	
	Specifications		
Collector Area	2035×1035 mm	1070×1070 mm	
Absorber Plate Dim.	2 m × 2m	1 m × 1m	
Riser tube diameter	12.5 mm	25mm	
Riser tube length	1980 mm	700 mm	
Computational Domain			
	Min	**Max**	**Number of cells**
X1	- 0.519 m	0.519 m	**64**
Y1	- 0.0001 m	0.1001 m	**6**
Z1	- 1.103 m	1.0196 m	**130**

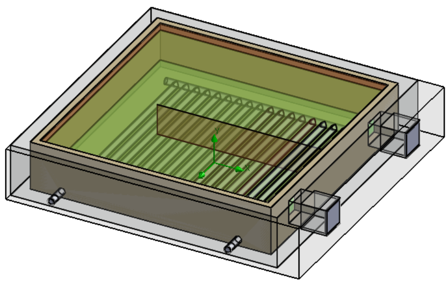

Figure 1 Three dimensional computational domain of solar air heater

In the present study, Cosmos Express Version 2014 is used for analysis. The following assumptions are imposed for the computational analysis.

1. The flow is steady, fully developed, turbulent or laminar and three dimensional.

2. The thermal conductivity of the collector wall, absorber plate and riser tube material are independent of temperature.

3. The collector wall, absorber plate and riser tube material are homogeneous and to be completely adiabatic, for simplification of the boundary condition.

4. The working fluid, water & air are assumed to be incompressible for the operating range of solar collectors since variation in density is very less.

5. Negligible radiation heat transfer and other heat losses.

4.2. *Mesh generation*

After defining the computational domain, uniform meshing is done by rectangular elements or hex elements. In creating this mesh, it is desirable to have more cells near the plate, because the turbulent boundary layer should be resolved, which is very thin compared to the height of the flow field. The computational time and accuracy depends on grid size. High number of grid will lead to good accuracy and vice versa. In this CFD modeling study, the grid independency test were considered and taken into calculations. Also, the grid system test was satisfied a good balance between the computational time and calculation accuracy. Fig. 2 shows the quadrilateral meshing.

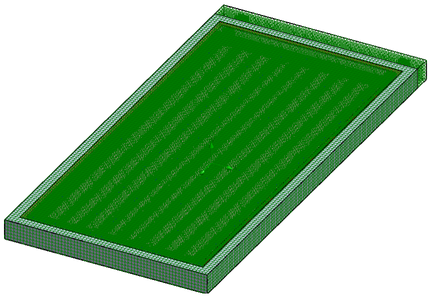

Figure 2 Meshing of solar water heater

4.3. *Boundary Condition*

The collector plate boundary conditions mainly include solar radiation intensity, ambient temperature, and mass flow rate of air in inlet and outlet. The specified boundary conditions are as detailed here.

1. The inlet of the domain allows fluid at a mass flow rate in a direction along the axis of the tube and the density and temperature are mentioned and zero relative static pressure is applied to the exit plane due to the water comes without any pressure head in it. (Al-Abbas, AH & Naser, J 2011& 2012)

2. The transient heat flux is applied on the surface of the absorber plate for the given time.

3. The wall surfaces are described with no slip conditions and an adiabatic boundary condition is applied to the outer surface of the riser tube.

4. An unstructured mesh is applied to the computational domain with the refined mesh density near the wall boundaries.

4.4. *Solver*

The full geometry and computational mesh for CFD simulations were created in the Solidworks software, which is used as a pre–processor for the CFD solver and Cosmosexpress V 2014 is used for post–processor.

The computational domain has been solved as a steady state conjugate heat transfer problem and the solution process is performed until convergence and an accurate balance of mass and energy are achieved. The solution process is iterative and each of the iteration in the steady state problem is treated as a pseudo–time step. In the iterative scheme, all the equations are solved iteratively, for a given time step, until the convergence criteria are met.

The simulations are carried out for the following cases

- Riser tube with absorber plate of conventional solar water heater for both laminar and turbulent flow.
- Riser tube with absorber plate of multipurpose solar water heater for both laminar and turbulent flow.
- Riser tube with absorber plate of conventional solar air heater for both laminar and turbulent flow.
- Riser tube with absorber plate of multipurpose solar air heater for both laminar and turbulent flow.

5. Simulation results and validation

Simulation is performed on riser tube with absorber plate in multipurpose solar heating system and is discussed in detail below. The inlet parameters from the experimental studies were used as the numerical description for simulation.

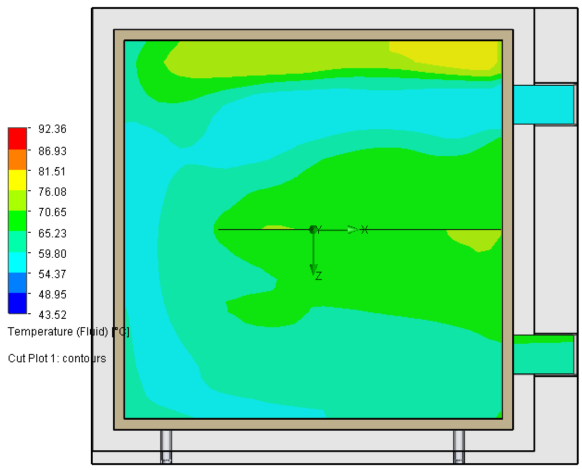

Figure 3 Temperature distribution simulation of the conventional solar air heater

5.1. *Solar Air Heater*

The experimental dimensions and thermal performance of conventional solar air heater, for natural circulation mode, is considered as input for simulation and the simulated results are compared with the multipurpose solar air heater.

5.1.1. *Conventional solar air heater*

The simulation module for the conventional solar air collector is shown in Figure 3. The module's initial conditions were applied as in the actual experimental one. As it is shown in Fig. 3, the maximum registered temperature of the collector was 57.30 $^\circ$C.

5.1.2. *Multipurpose solar air heater*

CFD analysis was carried out for the absorber plate with riser tube under consideration. The boundary conditions for the geometries were applied as per the experimentation. The results obtained by the analysis are discussed in detail below.

The temperature profile of CFD work of MPSAH was obtained using heated water from water heater and with ambient air as inputs. To analyze the heat transfer of the input parameters, analysis was done using different mode of heat transfer like conduction and convection. The results reveal that the high temperature output is obtained due to mixing of the heated air from the riser tube and the heated air through air heater. As a result the temperature of air close to surface is higher than the conventional solar heater. Fig. 4 shows the graph of the temperature profile along the length of the tube.

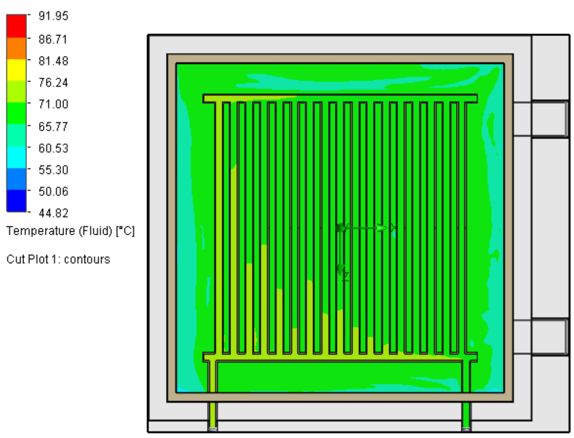

Figure 4 Temperature profile along the length of the plate

Using CFD analysis, the velocity profile of MPSAH was obtained for the different mass flow rates of air. Fig. 5 shows the output of variation in the air velocity across the surface.

It is observed from the Fig. 5 that the velocity profile is not resistance to high heat transfer rate from the surface of riser tube which is fitted below the absorber surface. The fluid flow and heat transfer analysis was carried out by using CFD software. In the present analysis, the multipurpose solar air heater with different mass flow rates has been tested under different climatic condition. The results show that the maximum heat transfer is observed at noon time since the heat energy gained by the water heater is stored in the riser tubes and is transferred to the air available in the air heater.

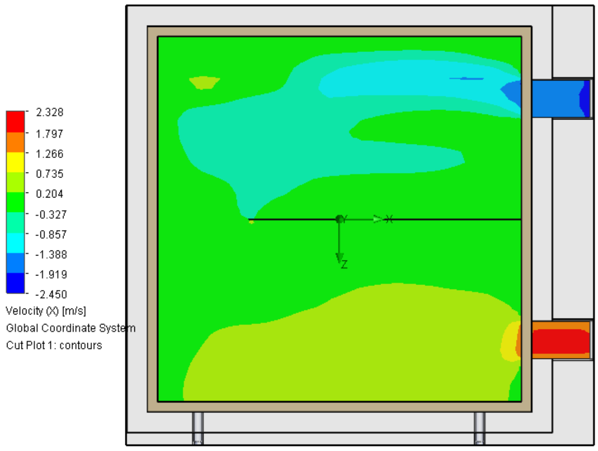

Figure 5 The output of variation in the air velocity across the surface

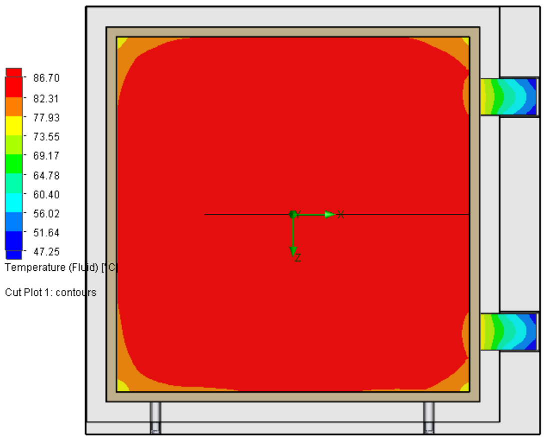

Figure 6 Temperature distribution simulation of the multipurpose solar air heater at no load condition

To find optimum solution, the different parameters such as load and no load conditions were analyzed experimentally as well as by CFD software at varying flow rate. Figure 6 shows that under no load condition, the maximum registered temperature for the absorber plate was 91 $^\circ$C because, maximum value of heat is transferred to the multipurpose solar air heater by additional heat energy supply from the riser tubes under consideration. Hence it is recommended that this new design multipurpose solar air heater can be used globally.

A comparison chart is drawn to indicate the experimental and simulated outlet temperatures versus time of summer and winter season at load condition as shown in Fig. 7 and Fig. 8, respectively.

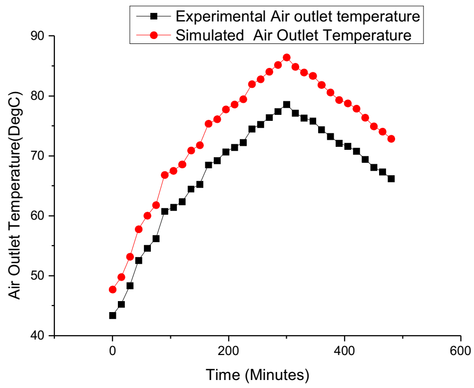

Figure 7 Comparison between experimental and simulated outlet air temperatures during summer season

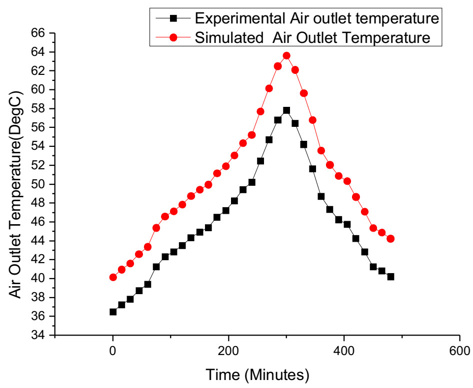

Figure 8 Comparison between experimental and simulated outlet air temperatures during winter season

It reveals that the value of temperature difference between the air inlet and outlet is almost 7.5 $^\circ$ C over all times. Also, the simulated temperature curve has the same behavior as that of the experimental one and they are close to each other.

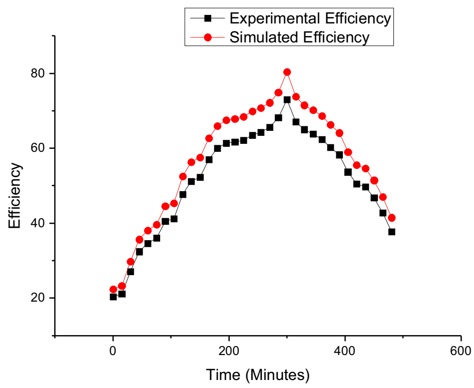

Figure 9 Comparison between experimental and simulated thermal efficiency of MPSAH during summer season

Similarly the maximum thermal performance efficiency of the multipurpose solar air heater at load condition during summer and winter in CFD analysis was registered as 80.15% and 73.86% for the mass flow rate of 0.0093 kg/s and 0.0176 kg/s, respectively.

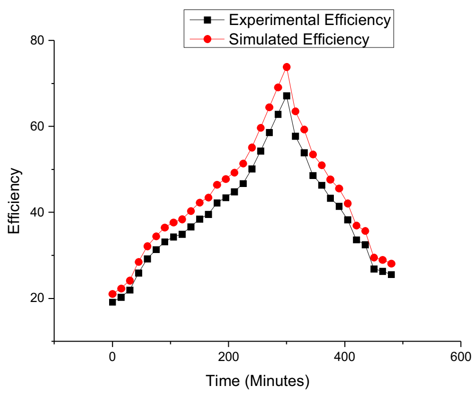

Figure 10 Comparison between experimental and simulated thermal efficiency of MPSAH during winter season

The comparison between experimental and simulated thermal efficiency of multipurpose solar air heater during summer and winter season is shown in Figs. 9 and 10, respectively. The range of thermal performance is lower in a conventional solar air heater compared to a multipurpose solar air heater due to the absence of energy transfer from the water heater.

6. Conclusions

The geometry of absorber plate and riser tube of both solar water and air heater are created, meshed and simulated by commercial CFD software. The simulation of MPSHS has been carried out on the assumption, that air is an incompressible fluid. In multipurpose solar air heating system, the simulated results are compared to experimental values and the deviation falls within ± 11.61% for summer season and ± 10.64% for winter season in addition to ignoring radiation heat transfer and other heat losses. It shows that the result obtained on heat transfer of different parameters experimentally agrees with the results of CFD. It proves that the simulated results falls within the acceptable limits.

References

[1] **Al–Abbas, A. H. and Naser, J.**: CFD modeling of air–fired and oxy–fuel combustion in a large–scale furnace at Loy Yang A brown coal power station, *Fuel*, 102, 646–665, **2012**.

[2] **Anderson, D. A., Tannehill, J. C. and Pletcher, R. H.**: Computational fluid mechanics and heat transfer, *Mc–Graw Hill Publications*, New York, **1984**.

[3] **Chaube, A., Sahoo, P. K. and Solanki, S. C.**: Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater, *Renewable Energy*, 31, 317–331, **2006**.

[4] **Ferziger, J. H. and Peric, M.**: Computational for fluid dynamics, *Springer Publications*, Germany, **2002**.

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

[6] **Ong, K. S.**: Thermal performance of solar air heaters: mathematical model and solution procedure, *Sol. Energy*, 55, 93–109, **1995**.

[7] **Karmare, S. V. and Tikekar, A. N.**: Analysis of fluid and heat transfer in a rib roughened surface solar air heater using CFD, *Solar Energy*, 84, 409–417, **2010**.

[8] **Karwa, R. and Chauhan, K.**: Performance evaluation of solar air heaters having v–down discrete rib roughness on the absorber plate, *Energy*, 35, 398–409, **2010**.

[9] **Gill, R. S., Singh, S. and Pal Singh, P.**: Low cost solar air heater, *Energy Conversion and Management*, 57, 131–142, **2012**.

[10] **Sahu, M. M. and Bhagoria, J. L.**: Augmentation of heat transfer coefficient by using 90^o broken transverse ribs on absorber plate of solar air heater, *Renewable Energy*, 30, 2057–2073, **2005**.

[11] **Versteeg, H. K. and Malalasekera, W.**: An introduction to computational fluid dynamics, The finite volume method, *Longman Scientific and technical*, Malasia, 30, 1345–1357, **1995**.

[12] **Whiller, A. and Saluja G.**: Effect of materials and constructive details on the thermal performance of solar water heaters, *Solar Energy*, 9(1), 21, **1965**.