International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 2, No. 1, January 2013

Research Paper

CFD ANALYSIS OF TRIANGULAR ABSORBER TUBE OF A SOLAR FLAT PLATE COLLECTOR

Basavanna S[1]* and K S Shashishekar[1]

Corresponding Author: **Basavanna S, ✉ basu.guru@rediffmail.com*

Solar flat plate collectors are commonly used for domestic and industrial purposes and have the largest commercial application amongst the various solar collectors. This is mainly due to simple design as well as low maintenance cost. An attempt is being made in this paper to analyze the solar collector using the Computational Fluid Dynamics (CFD) so as to simulate the solar collector for better understanding of the heat transfer capabilities of the collector. In the present work, Fluid flow and heat transfer in the collector panel are studied by means of Computational Fluid Dynamics (CFD). The conjugate heat transfer phenomenon between collector and water is modeled using FLUENT CFD software. The solar radiation heat transfer is not modeled; however radiation effects are taken in to consideration while calculating the heat flux boundary conditions for the collector area. The geometric model and fluid domain for CFD analysis is generated using ANSYS Design Modeler software, Grid generation is accomplished by ANSYS Meshing Software. The numerical results obtained using the experimentally measured temperatures are compared to the temperatures determined by the CFD model and found to have a good similarity between the measured and calculated results.

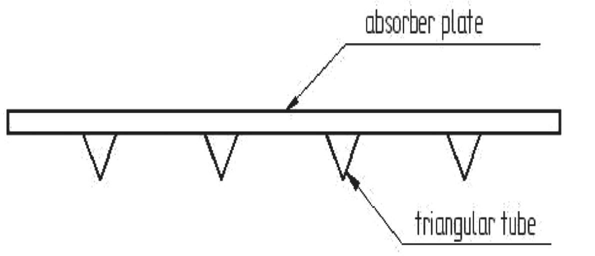
Keywords: Conjugate heat transfer, Solar radiation simulation, Solar collector, Computational Fluid Dynamics (CFD), Fluent CFD software, ANSYS design modeler software

INTRODUCTION

Solar energy collectors are special kind of heat exchangers that transform solar radiation energy to internal energy of the transport medium. The major component of any solar system is the solar collector. Of all the solar thermal collectors, the flat plate collectors though produce lower temperatures, have the advantage of being simpler in design, having lower maintenance and investment cost.The experimentally measured inlet and outlet temperature of triangular tube configuration was obtained and CFD simulations are carried out for triangular tube configuration as shown in Figure 1.

[1] Department of Mechanical Engineering, S.I.T, Tumkur 572103, India.

Figure 1: Triangular Tube Configuration



The Figure 1 shows the absorber plate and the triangular tubes attached below the absorber plate. The heat energy from the sun is absorbed by the absorber plate and conducted to the tubes having surface contact with the absorber and heat is absorbed by the working fluid flowing through the triangular tubes. By this arrangement the surface area of contact between the tube and the plate will increase, resulting in more heat absorption and hence enhanced performance of the collector.

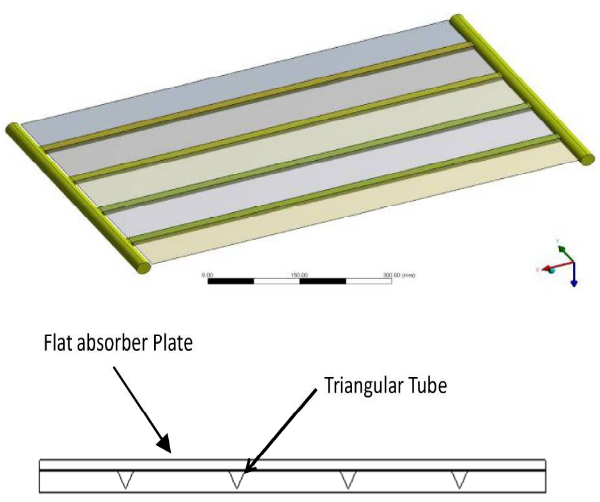
THEORETICAL FORMULATIONS

Problem Statement and Assumptions

The flow domain consists of a flat plate absorber plate with triangular absorber tube attached below the absorber as shown in Figure 2. Water is circulated through the absorber tube. Following assumptions are made in the analysis.

- Water is a continuous medium and incompressible.
- The flow is steady and possesses laminar flow characteristics.
- The thermal-physical properties of the absorber plate, water and the absorber tube are independent of temperature.

Figure 2: Geometric Modelling of Triangular Tube-Absorber Configuration



Collector Configurations for the Analysis

The collector absorber plate length and width for all the design is taken as 1 m and 0.5 m respectively. The absorber tube is 0.017 m × 0.017 m × 0.017 m cross sectional area with a thickness of 0.0002 m. The length of the absorber tube is 0.8m and has a thickness of 0.0005 m.

Numerical Model

To conduct numerical simulation, the computational domain is meshed with control volumes built around each grid using ANSYS Meshing Software which is the preprocessor for FLUENT (version13). Numerical simulation was carried out using steady state implicit pressure based solver which is an in-built in the commercially available software FLUENT (version 13). The governing partial differential equations, for mass and momentum are solved for the steady incompressible flow. The velocity-pressure coupling has been effected through SIMPLE algorithm (Semi Implicit Method for Pressure-Linked Equations) developed by Patankar and Spalding (1972). Second order upwind schemes are chosen for the solution schemes.

Mean Flow Equations

All the equations are presented in Cartesian tensor notation.

Continuity

$$\nabla(\rho V)+\frac{\partial \rho}{\partial t}=0 \qquad ...(1)$$

Momentum equation:

$$\frac{\partial}{\partial x_j}\left(\rho U_i U_j\right)=\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_j}\right)-\rho\overline{u_i u_j}\right] \qquad ...(2)$$

Energy equation

$$\frac{\partial}{\partial x_j}\left(\rho U_i T\right)=\frac{\partial}{\partial x_j}\left[\frac{\mu}{P_r}\frac{\partial T}{\partial x_j}-\rho\overline{u_i t}\right] \qquad ...(3)$$

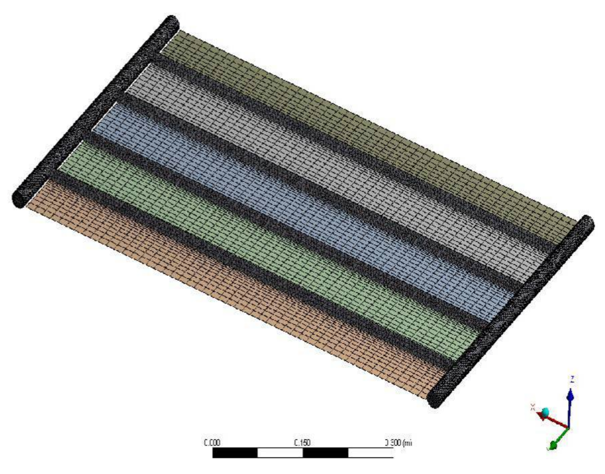
METHOD OF SOLUTION

Numerical Scheme

Conservation equations are solved for the control volume to yield the velocity and temperature fields for the water flow in the absorber tube and the temperature fields for the absorber plate. Convergence is effected when all the residuals fall below 1.0e-6 in the computational domain. The geometric model and fluid domain for CFD analysis is generated using ANSYS Design Modeler software, Grid generation is accomplished by ANSYS Meshing Software. A three dimensional computational domain is built as shown in Figure 3.

The grid independence test is performed to check validity of the quality of mesh on the solution. The influence of further refinement did not change the result by more than 0.75% which is taken here as the appropriate mesh quality for computation.

Figure 3: CFD Meshing of Triangular Tube Absorber Configuration



Boundary Conditions and Operating Parameters

Appropriate boundary conditions are impressed on the computational domain, as per the physics of the problem. Inlet boundary condition is specified as velocity inlet condition. Outflow boundary condition is applied at the outlet. Wall boundary conditions are used to bound fluid and solid regions. In viscous flow models, at the wall, velocity components are set to zero in accordance with the no-slip and impermeability conditions that exist there. The interface between the water and the absorber tube is defined as wall with coupled condition to effect conjugate heat transfer from absorber tube to the water. A varying heat flux equivalent to the solar insolation is applied at the top surface of the absorber plate. The bottom and side surfaces of the absorber plate and the outer surface of the absorber tube are defined as wall with zero heat flux condition to effect insulated conditions. The material used for both absorber plate and the tube is copper. The input parameters used in the analysis are as shown in Table 1.

Table 1: Input Parameters for Simulation

Parameter	Value
Density (Copper)	8978 kg/m^3
Specific Heat (Copper)	381 J/kg-K
Thermal Conductivity (Copper)	386 W/m-K
Density (Water)	998.2 kg/m^3
Viscosity (Water)	0.001003 kg/(m.s)
Specific Heat (Water)	4182 J/kg-K
Thermal Conductivity (Water)	0.6 W/m-K

Source: Manjunath et al. (2011)

RESULTS AND DISCUSSION

CFD simulation is first performed on a triangular tube-flat plate absorber with water as working fluid. In this case pressure drop is increased due to triangular tube. In the straight flow with constant cross-sectional riser tubes, pressure increases from the entrance to the other end of inlet header due to momentum gain due to decrease in mass flow rate, whereas the pressure decreases along the outlet header due to momentum losses.

Figure 4 shows CFD calculated temperatures of the Midplane absorber plate with a tilt angle of 45° when the collector fluid enters the panel at 1.5 l/min and 310.3 K, and is heated by solar irradiance of 1316 W/m^2 (Rhushi *et al.*, 2010). The fluid temperature in the dividing manifold is shown to be the same as the inlet temperature. As the fluid flows along the tube, the working fluid gains heat and its temperature rises. Figure 5 shows the temperature at the end portion of the plate to be high due to the fact that this part of the absorber plate doesn't contain absorber tube which is used to absorb the heat. Thus there is no heat sink at these locations and this result in rise in temperature. There is rise in plate

Figure 4: Middle Plane Temperature Contours for Triangular Tube Absorber at Inlet Temperature of 310.3 K

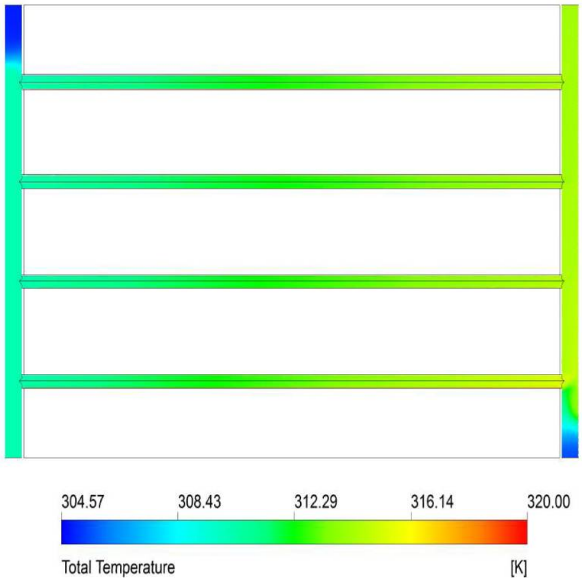


Figure 5: Temperature Contours for Triangular Tube-Absorber at Inlet Temperature of 310.3 K

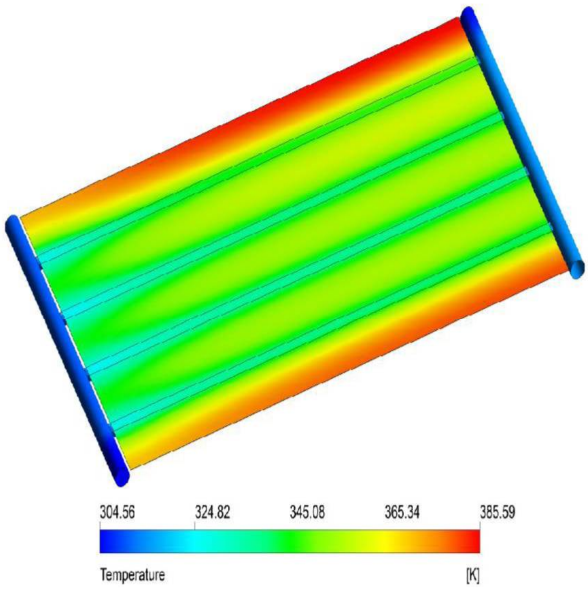


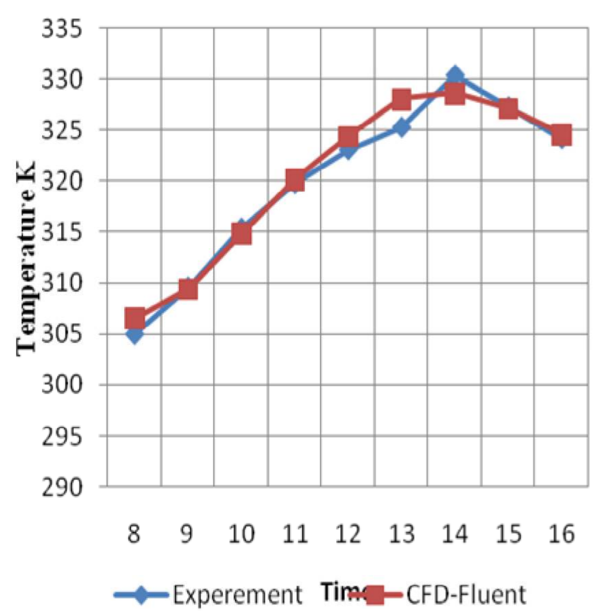
Table 2: Comparison of Outlet Temperature for Triangular Tube

Time	Ambient Temp in K	Radiation in w/m^2	Exp Inlet Temp K	Exp. Outlet Temp K	CFD Outlet Temp K
8	300.5	878.5	305.0	305.0	306.5
9	301.0	1125.8	305.3	309.6	309.4
10	302.0	1316.8	310.3	315.4	314.8
11	302.5	1444.3	312.0	319.8	320.2
12	303.5	1501.8	314.2	323.1	324.4
13	305.0	1485.8	315.3	325.3	328.1
14	307.5	1397.3	319.4	330.4	328.7
15	305.5	1241.3	318.8	327.3	327.2
16	304.0	1025.3	316.9	324.2	324.6

temperature of about 385.59 K for inlet fluid temperature of 310.3 K.

The discrepancy obsevered in Table 2 and Figure 6 is, the outlet temperature of the experimental and CFD results is due to the fact that in experimental observation, the outlet temperature is measured at one particular location of the outlet header, while in CFD analysis the average temperature of the fluid flow through the outlet header is indicated. Hence the observed discrepancy in the outlet temperature. This results in a good comparision between experimental and CFD studies. Though the deviation is approximately 5%.

Figure 6: Outlet Temperature from Experimatal Data and CFD Simulation Data for Triangular Tube-Absorber



CONCLUSION

A triangular tube configuration raises the outlet temperature due to better contact between tube and plate. In the header, the fluid temperature in the dividing header is shown to be the same as the inlet temperature. As the fluid flows through the tube, the collector fluid gains heat and its temperature rises. In the absorber plate the temperature at the end portion of the plate is high due to the fact that this configuration tube has more surface area of contact between the tube and the plate, resulting in more heat absorption and hence enhanced performance of the collector.

REFERENCES

1. Bukola O Bolaji (2006), "Flow Design and Collector Performance of Natural Circulation Solar Water Heater", *Journal of Engineering and Applied Science*.

2. Fluent 13, Fluent Inc., Cavendish Court Lebanon, NH, 03766, USA.

3. Jianhua Fan, Louise Jivan Shah and Simon Furbo (2007), "Flow Distribution in a Solar Collector Panel with Horizontal Fins", Technical University of Denmark, DK-2800 Kgs, Lyngby, Denmark.

4. Manjunath M S, Vasudeva Karanth K and Yagnesh Sharma N (2011), "Three Dimensional Numerical Analysis of Conjugate Heat Transfer for Enhancement of Thermal Performance Using Finned Tubes in an Economical Unglazed Solar Flat Plate Collector", Proceedings of the World Congress on Engineering, Vol. III, WCE 2011, July 6-8, London, UK.

5. Patankar S V and Spalding D B (1972), "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows", *International Journal of Heat Mass Transfer*, Vol. 15, p. 1787.

6. Rhushi Prasad P, Byregowda H V and Gangavati P B (2010), "Experiment Analysis of Flat Plate Collector and Comparison of Performance with Tracking Collector", *European Journal of Scientific Research*, Vol. 40, No. 1, pp. 144-155, ISSN 1450-216X.

7. Vasudeva Karanth K, Manjunath M S, Yagnesh Sharma N (2011), "Numerical Simulation of a Solar Flat Plate Collector Using Discrete Transfer Radiation Model (DTRM)—A CFD Approach", Proceedings of the World Congress on Engineering, Vol. III, WCE 2011, July 6-8, London, UK.