IJETST- Vol.||04||Issue||10||Pages 6266-6271||October||ISSN 2348-9480 2017

International Journal of Emerging Trends in Science and Technology
IC Value: 76.89 (Index Copernicus) Impact Factor: 4.219 DOI: https://dx.doi.org/10.18535/ijetst/v4i10.14

CFD Analysis of Circular Pipe Flat Plate Solar Collector

Authors
Zhang Jun[1], Muhammad Ayaz Akbar[1], Tareq Manzoor[2], Ahsan Ali[3], M.Amjad[4]
[1,2]School of Mechanical Engineering, Dalian University of Technology, Dalian, P.R China
[3,4]Department of Mechanical Engineering, COMSATS Institute of Information Technology, Sahiwal, Pakistan

Abstract
Solar energy is available in abundance but least used energy source of available renewable energy resources. Solar Energy is being used for some household and commercial purposes like producing steam for commercial usage, heating of water to maintain the indoor temperature of houses etc. Water heating requires heat, which is produced usually with burning of fuels (Methane, Gasoline) and these fuels are costly and causes pollution at burning, but if we use solar energy which is available for almost 10-12 hours in almost every country, we can save a lot. This paper attempts to make the numerical simulation of flat plate solar collectors. Simulation is an important tool for the design and operation control. Design engineers use simulation results to design water heating systems, simulation makes it possible to find the optimal design and operating parameters. In this paper, the computational fluid dynamics (CFD) tools have been used to simulate solar collectors to better understand heat transfer capacity. Three-dimensional model of the U-tube collectors and flat absorber plate is obtained using Solid Works. The results obtained by the use of ANSYS FLUENT software. The purpose of this work is to better understand the relative computational fluid dynamics flow of the solar collector and the temperature distribution within solar collector. Outlet air temperature and solar radiation are compared and between them there is a good consistency. Our project is basically in our study design optimization, we can use our results to improve the efficiency of flat plate solar collector theory in the future. These results can also be used for design purposes.
***Keywords*:** CFD; Solar Energy

1. Introduction

Solar energy is the light and heat coming from the sun in the form of radiation which are utilized using different present technologies such as photovoltaic, solar thermal energy etc. Basically, solar energy is a major source of renewable energy. The technologies of solar energy are distinguishing as active solar or passive solar. These types are because of the way they convert solar energy into power or collect and distribute solar energy. In active solar techniques, solar energy is utilized by using solar water heating, photovoltaic systems and concentrated solar power. In passive solar techniques, buildings are aligned in the direction of the sun, then material are selected having suitable light dispersion properties and suitable thermal mass, and at the end spaces are designed which naturally circulates the air.

Solar energy is the highly considerable source of energy due to large magnitude of available solar energy. Distance from sun to sun land 1.495x1011 1.39x109 cm diameter, about the 1353(W / m²) heat flux available through radiation to earth. 170 trillion (kW) Energy reached the world and the reflected solar energy back to space is 30%, 47% to low heat, and 23% in biosphere evaporation / precipitation cycle and less than 0.5% in the Wind, waves and plants used for photosynthesis kinetic energy. The solar heating system consists of many parts. These parts are the most important in heating process in which the heat transfer occurs and the fluid absorbs sun energy (Solar Flux). Dried fruits and vegetables, including grapes, peppers and Papaya is one of those essential applications that are performed by burning of fuel if solar energy is not used. Solar dryers quickly dry fruits in mind, with the traditional fuels potentially not burned. Therefore, the need for solar energy storage system that provides energy during the night. Furthermore, the sun drying applications needs suitable development. The conversion of solar energy to solar collector's heat exchanger radiant energy transfer medium is an especially inside. Main part of any solar system is the solar collector. For all the solar collectors, there is low maintenance costs with the benefit of a simple design, to maintain the temperature there is a flat plate solar water heaters having gained maximum rate established at the cost of solar energy. Solar Water heaters are used widely in many applications, such as are used for residential, industrial and agricultural sectors the solar collector types. Solar collectors is a large part of the active solar thermal system. Receive sunlight, the heat of the sun, and the fluid (air or water) heat transfer.

I. Flat Plate Solar Collectors:

Flat plate collectors by Hottel and Whillier developed in the 1950s, it is the most common type. These include (1) a dark plate absorber, (2) a transparent cover, reduce heat loss, (3) fluid heat transfer (air, water or antifreeze) to remove heat from the absorber, and (4) thermal insulation backing. The absorbent sheet is used as the material with great heat transfer capability and a high melting point and also a relatively cheaper material is preferred and coating of dark color id than applied at top to gain maximum energy. Heat is transferred usually through fluid and plate to an insulated tank. This may be reached directly or through a heat exchanger

II. Aim of Work:

The aim of our work is to simulate the solar collector fluid flow within the flat plate solar collector and understand the heat transfer capability, water outlet temperature of the heat flow at different day times. The water temperature at the outlet of the solar collector is predicted using the numerical estimation method.

III. Numerical Simulation by Software:

Computational fluid dynamics, generally known as CFD, is a branch of fluid mechanics, numerical analysis and algorithms to solve and analyze problems related to fluid flow. Computer used to perform calculations required to simulate the interaction of the boundary conditions defined by the surface of liquids and gases. With the speed of supercomputers, you can get a better solution. Production Software ongoing research, can improve the accuracy and speed of complex simulation programs such as transonic or turbulent. Computational Fluid Dynamics (CFD) is predicted by the numerical solution of mathematical equations governing the collection of fluid flow, heat and mass transfer, chemical reactions, and scientific phenomena related

- Conservation of mass
- Conservation of momentum
- Energy conservation
- Species protection
- Physical impact

CFD analysis of the results is relevant:

- The new design study concept
- The development of detailed product
- Troubleshooting
- Redesign

i. How does CFD works:

ANSYS CFD solver is based on the finite volume method, through the following steps to resolve any problems

- Domain is a set of discrete finite volumes.
- Transport equation of mass, momentum, energy, species, have to be solved
- Conversion of partial discrete differential equations into algebraic equations
- All algebraic equation solutions than present numerical solutions.

ii. CFD Methodology:

CFD code contains following main elements.

1. Identify the problem
   a. define goals
   b. recognition of domain
2. Pre-Processing.
   a. Geometry
   b. Mesh
   c. Physics and solver configuration
3. Solution.
   a. Compute Solutions

4. Post processing
    a. Examine results

1. Identify the problem

This step involves the theoretical work involving with our work. We study the situation we are going to analyze and to analyze that typical problem which sort of conditions are required.

- Define Goals

In analysis of flat plate solar collector, we are looking for change in temperature due to solar energy which is obvious that we are looking to increase the temperature. In this simulation we will limit our results to less than boiling point of our working fluid e.g. for water less than $100°C$ because above this temperature we have not installed a very large water heating system we have approximately 2-meter-long plate. We have to make assumptions to solve this typical problem easily, assumptions are

- 3D model is in a state of equilibrium.
- The ambient temperature is considered constant.
- Flow is assumed to be laminar.

1. Recognition of domain.

In this analysis we are going to model only pipe which are in this case U-Shaped and absorber plate. We are going to neglect the insulation at bottom surface because in ANSYS Fluent we can assume there is no heat transfer from bottom surface of solar collector.

Pre-Processing:

1. Geometry

CFD analysis needs fluid domain and solid part as well, so for this need we have to make two geometries one is pipe and absorber plate merged together and other is fluid domain. The geometry for this problem is constructed on Solid Works 2014.

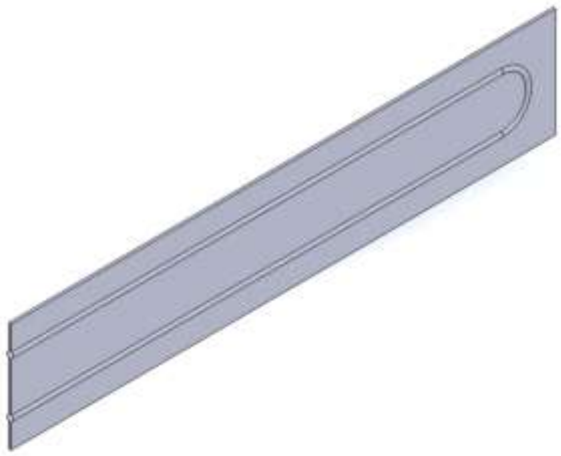

Fig 1: Flat Plate Collector

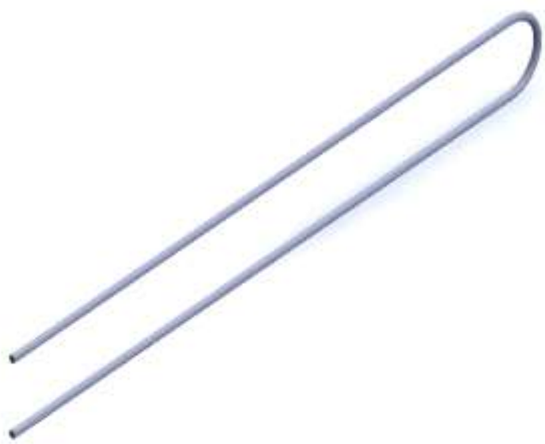

Fig 2: Fluid Domain

2. Mesh

The CFD solver configured to be used the control volume method. Grid is an important part of any mesh, because the grid assembles finite element geometry. The mesh is done for discrete geometry.

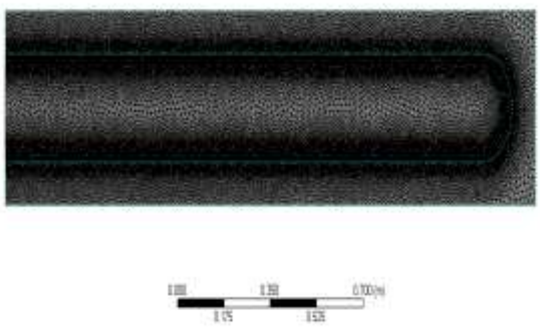

Fig 3: Mesh

2. Physics and Solver Setting

In this step we define fluid and solid's properties according to our need. In this project the solid used as absorber is Copper alloy and fluid is water liquid. Their properties are given as:

Table 1: Properties of Solid and Fluid

Properties	**Value**
Density (kg/m^3)	8920
Cp (Specific Heat) (J/(kgK)	381
Thermal Conductivity (W/m.K)	394
Density (kg/m^3)	998.2
Cp (Specific Heat) (J/kg.K)	4182
Thermal Conductivity (W/m.K)	0.6
Viscosity (kg/m.s)	0.001003

After adjusting all materials properties, we setup solver with setting up input values for the solution as Heat Flux, Inlet Velocity, convection constant and atmospheric/ free stream temperature to allow convection. Beside all these the main thing is heat transfer project we have to identify the region from where heat will transfer through solid into liquid, in this case which is inside walls of pipe at plate.

Table 2: Input Heat Flux with changing Time

Time	Solar Heat Flux
9 am	621.7
10 am	750.51
11 am	879.5
12 pm	909
1 pm	948
2 pm	909.5
3 pm	790
4 pm	597.5
5 pm	353

2. Solve:

3. Compute Solution

In this solution phase after applying all boundary conditions we solve our problem, or start the simulations till convergence, in our project convergence was achieved at 89 iterations on average. Convergence occurs when results of an iteration are similar to previous iteration, means not significant changes between iterations starts to occur.

3. Post-Processing:

4. Examine Results.

In this last step of our simulation we examine our results with the help of contours, vectors, plots, and other possible ways.

After simulating all the analysis, we have summarized to these following results, we can better understand the results through contours, vectors, plots etc. We have shown here some contours for the Temperature distribution along the pipe and plate. At each time hour we have temperature distribution along the pipe and plate.

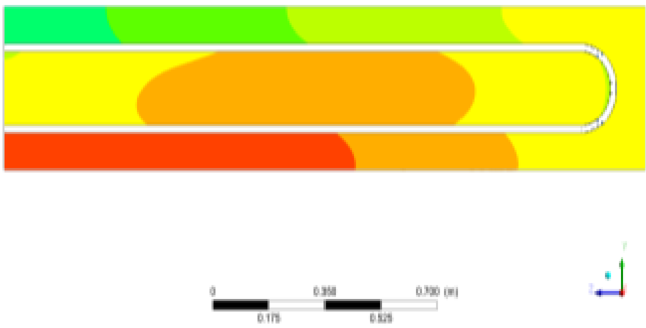

Fig No 4: Temperature Contour (Top Surface)

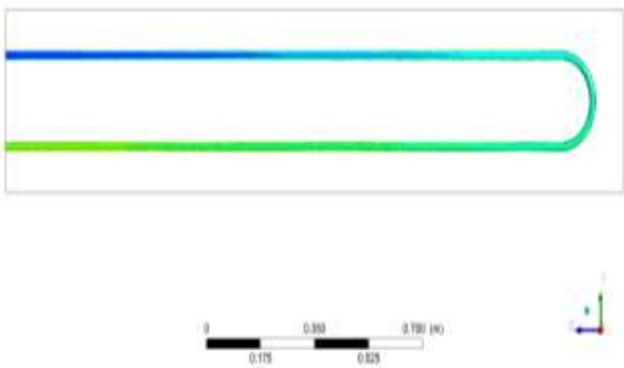

Fig No 5: Temperature Contour (Walls Pipe)

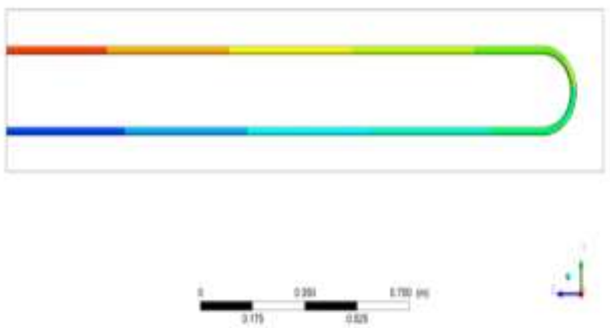

Fig No 6: Pressure Contour (Walls Pipe)

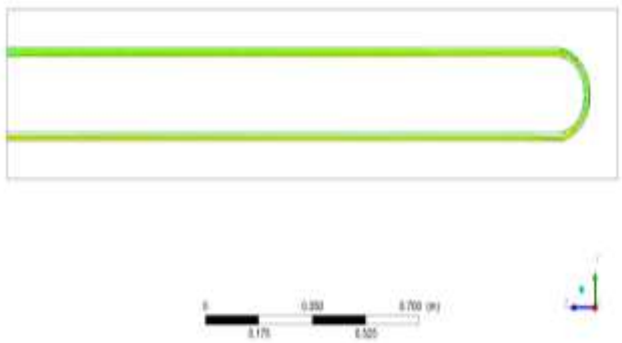

Fig No 7: Velocity Stream Lines

Table 3: CFD Results.

Time Hours	**Solar Intensity** (W/m^2)	**Ambient Temperature** (℃)	**Collector Temperature Obtained by CFD** (℃)
9 am	621.7	30	58.99
10 am	750.51	32.7	67.7
11 am	879.5	35	76
12 pm	909	36.9	79.3
1 pm	948	39.5	83.7
2 pm	909.5	38.9	81.3
3 pm	790	36	72.6
4 pm	597.5	33	60.87
5 pm	357	31	47.65

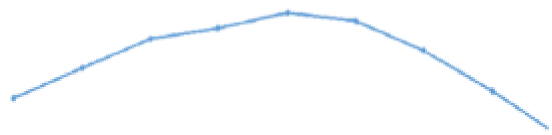

Fig No 8: Variation of Outlet Temperature with Time

2. Conclusions

This CFD Analysis of Flat Plate Solar collector gives feasible results, which could be used in any process where hot water is necessary and fuel is burnt to heat water, then we can use this type of flat plate solar water heater to fulfil our demand. On commercial scale we can attach multiple plate like this with one another and can get steam with temperature above 120 $°C$ easily which could be used in small scale power plants.

.

Acknowledgement

In future more fluid like air can also be used and analysis on another fluid can also be performed. Our project is theoretical in nature, in which design optimization could be studied and the results can be used in future for design of the solar flat plate as well as to enhance the efficiency of the collector. We can also enhance design efficiency by increasing the absorber area.

.

References

1. John A. Duffie , William A. Beckman, “Solar Engineering of Thermal Process” Second Edtition.

2. R. E. H. Sims, R. N. Schock, A. Adegbululgbe, J. Fenhann, I. Konstantinaviciute, W. Moomaw, H. B. Nimir, B. Schlamadinger, “Energy Supply”, in Climate Change 2007: Mitigation. Contribution of Working Group III to the Fourth Assessment Report of the Intergovernmental Panel on Climate Change, Cambridge, United Kingdom and New York, USA: B. Metz, O. R. Davidson, P. R. Bosch, R. Dave, L. A. Meyer, 2007.

3. Tin-Tai Chow .”Performance evaluation of evacuated tube solar domestic hot water systems in Hong Kong.2011
4. G.L. Morrison, I. Budihardjo, M. Behnia (2004), “Water-in-glass evacuated tube solar water heaters”, Solar Energy, Vol.76, pp.135-140.
5. G. L. Morrison, I. Budihardjo, e M. Behnia, “Measurement and simulation of flow rate in a water8in8glass evacuated tube solar water heater”, Solar Energy, vol. 78, no. 2, p. 257–267, fev. 2005.
6. Indra Budihardjo, Graham L. Morrison, Masud Behnia (2007), “Natural circulation flow through water-in-glass evacuated tube solar collectors”, Solar Energy, Vol.81, pp.1460-1472.
7. L. J. Shah e S. Furbo, “Vertical evacuated tubular8collectors utilizing solar radiation from all directions”, Applied Energy, vol. 78, no. 4, p. 371–395, ago. 2004.
8. L. J. Shah e S. Furbo, “Theoretical flow investigations of an all glass evacuated tubular collector”, Solar Energy, vol. 81, no. 6, p. 822–828, jun. 2007.
9. F. O. Gaa, M. Behnia,S. Leong, e G. L. Morrison, “Numerical and experimental study of inclined open thermosyphons”,International Journal of Numerical Methods For Heat & Fluid Flow, vol. 8, no.7, p.748, 1998.
10. Z. Y. Li, C. Chen, H. L. Luo, Y. Zhang, e Y. N. Xue, “All-glass vacuum tube collector heat transfer model used in forced8circulation solar water heating system”, Solar Energy, vol. 84, no. 8, p. 1413–1421, ago. 2010.
11. R. Tang, Y. Yang, e W. Gao, “Comparative studies on thermal performance of water8in8glass evacuated tube solar water heaters with different collector tilt8angles”, Solar Energy, vol. 85, no. 7, p. 138181389, jul. 2011.
12. A. I. Sato, V. L. Scalon and A. Padilha , “Numerical analysis of a modified evacuated tubes solar collector”.2012