www.ijiarec.com

ISSN:2348-2079

Volume-7 Issue-2

International Journal of Intellectual Advancements and Research in Engineering Computations

Analysis of Copper Pipe in Solar Water Heater using Ansys-Fluent

R. Prakash[1], T. Suriyanarayan[2], S. Suwetha[2], R. Vinothkumar[2], M. Sivasurya[2]

[1]Assistant Professor, Department of Mechanical Engineering, Velalar College of Engineering and Technology, Tamil Nadu, India.

[2]UG Students, Department of Mechanical Engineering, Velalar College of Engineering and Technology, Tamil Nadu, India.

ABSTRACT

This project deals with two different models of pipe design and concludes the best possible design for thermal efficiency. Solid Works 2018 version was used to design the pipes. Numerical analysis for thermal testing was conducted by Flow Simulation of ANSYS FLUENT. Pressure drop and thermal output is calculated across the pipes with two different thermal inputs. The significance of the inlet velocity and varying incident solar radiation on the nature of flow and performance of the collector is examined. Mass flow rates have been varied from 0.0002 to 0.03kg/s while the solar insolation considered was in the range of 600 to 1000 W/m2. For a uniform heat flux, with decrease in velocity of the fluid entering the copper tube pipe, temperature of the water obtained at the outlet was found to be higher. This is due to the prolonged interaction of the fluid with the tube walls, which facilitates a higher heat gain from the tube surface to the fluid. Similarly, as the heat flux increases, the magnitude of the tube surface temperature increases which results into a higher outlet temperature.

Keywords: Solar insolation, Ansys-Fluent.

INTRODUCTION

Solar thermal energy is used to heat water or other fluids, and can also power solar cooling systems. They differ from solar photovoltaic (PV) systems, which generate electricity rather than heat. Solar thermal energy has been used for centuries by the harnessing of solar energy for heating and drying. More recently, in a wide variety of thermal processes, solar energy has been developed for power generation, water heating, and mechanical crop drying and water purification among others.

The primary uses for solar thermal systems in rural locations are water heating for domestic use and swimming pools; heating of buildings; and evaporative processes (such as distillation and dryers) for agriculture. All of these processes require an operating temperature of less than 100°C; therefore the solar energy flux is enough to elevate the working fluid to the required temperature without needing to concentrate the sun's energy (i.e. using mirrors or lenses. There are three main types of solar hot water panels: Evacuated tube, glazed flat panel, and unglazed flat panel. Each of these types of panels has different operating conditions where they have better efficiency than the others. To make this project we are using plastic cover in order to reduce costs since we want to focus on low-income places so it will have a higher availability. Also we made this with copper which can be found naturally in all plants and animals. Also copper is a ductile metal found in nature well known for its electrical conductivity and its use for construction and plumbing purposes.

Author for correspondence:
Department of Mechanical Engineering, Velalar College of Engineering and Technology, Tamil Nadu, India.

A Flat Plate Collector is a heat exchanger that converts the radiant solar energy from the sun into heat energy using the well-known greenhouse effect. It collects, or captures solar energy and uses that energy to heat water in the home for bathing, washing and heating, and can even be used to heat outdoor swimming pools and hot tubs.

A solar flat plate collector typically consists of a large heat absorbing plate, usually a large sheet of copper or aluminum as they are both good conductors of heat, which is painted and chemically etched black to absorb as much solar radiation as possible for maximum efficiency. This blackened heat absorbing surface has several parallel copper pipes or tubes called risers, running length ways across the plate which contains the heat transfer fluid, typically water.

These copper pipes are bonded, soldered or brazed directly to the absorber plate to ensure maximum surface contact and heat transfer. Sunlight heats the absorbing surface which increases in temperature. As the plate gets hotter this heat is conducted through the risers and absorbed by the fluid flowing inside the copper pipes which is then used by the household.

The pipes and absorber plate are enclosed in an insulated metal or wooden box with a sheet of glazing material, either glass or plastic on the front to protect the enclosed absorber plate and create an insulating air space. This glazing material does not absorb the suns thermal energy to any significant extent and therefore most of the incoming radiation is received by the blackened absorber.

LITERATURE REVIEW

Mohamed Selmi, Mohammed J. Al-Khawaja, AbdulhamidMarafia [1], present the collector performance, after obtaining 3-D temperature distribution over the volume of the body of the collector, was studied with and without circulating water flow. An experimental model was built and experiments were performed to validate the CFD model. It seems the temperature difference between the water inlet and outlet is almost 9°Cover all times. Also, the simulated temperature curve has the same behaviour as that experimental one and they are close to each other.

Present the investigation describes the construction and experimentation of solar energy absorbers using water as fluid and its simulation in Computational Fluid Dynamics. For Absorber A with rectangular cross section and Absorber B with circular cross section, water temperature was calculated using solar radiation and ambient temperature measurements showing increases of up to 62.5°C for both absorbers.

S.Eswaran, M.Chandru, M.Vairavel, R.Girimurugan, the objective of this study is to validate the mass flow rate of water inside the collector tube in an Ado-Ekidi natural circulation solar water heater system. The existing solar water heating systems the optimum mass flow rate is 0.1 kg / m2. The study of numerical analysis is carried out in reengineered Ado-Ekidi solar water heater by using CFD software. Results show that maximum mass flow rate achieved more than experimental values. At the mass flow rate of 0.6 kg / m2, dynamic pressure of 4.30×105 Pa, flow velocity of 5.91×105 l/m2 & relative temperature of 360° K.

Yuechao D. ET. al. investigated novel flat plate solar water heater using micro heat pipe array with selective coating closely arrangement as shown in fig. MHPA has high heat transfer rate, high reliability, and high compressive strength, low cost and small thermal resistance. This new FPC has high heat transfer ability, relatively low heat loss, resistance to freezing, preventing leakage and most important elimination of welding. It was shown that novel collector has excellent thermal performance. [2]

K.A. Muhammed Yarshi & Dr. Benny Paul (2015) analyses the effect of change in shape of absorber tube for flat plate collector performance. He investigated the effect of mass flow rate and absorber material on FPC efficiency. Set up consist of absorber tube of 0.8 m long with inner dia. 0.012 m. Header pipe of 0.8 m long with dia. 0.24 m, glazing area is 0.5 m2. Overall dimensions are 1×1.5×0.1 m. He studied collector with circular and semi-circular absorber tube performance. It was examine that outlet temperature of semi-circular collector is greater than conventional circular collector due to increasing absorbing area. [3]

Basavanna S. and K.S. Shashishekar investigated triangular absorber tube in FPC. Triangular absorber tube is attached in order to increase the contact surface area between tube and plate, which will enhance heat absorption so that performance of collector increases. Set up consist collector box of 1 m length and 0.5 m width, absorber tube is 0.017×0.017×0.017 m, 0.8 m long and 0.0005 m thick. These results are comparing with CFD simulations. [4]

COMPUTATIONAL ANALYSIS

- Turbulence Model: Ke - Standard wall function.

(The equations of momentum (in three directions x, y, z), continuity, energy, k and ε were solved simultaneously. It is worthy to note that ε and k are only dependent on the flow and geometry of the impeller) [5].

- Inlet: Temperature - 300K
- Outlet: Pressure outlet - gauge pressure.
- Wall: Heat flux - 600 W/mm^2 and 1000W/mm^2

Three steps to be followed for CFD analysis

- Pre processing
- Processing
- Post processing

Preprocessing

This step is a primary and most important step to be followed to prepare the geometry for the future analysis. This involves

- Geometry clean up
- Surface mesh generation
- Setting Quality criteria

For this step, we use ANSA. The below steps are to be followed;

Open the ANSA software

- File> open> browse for the .iges
- Wireframe of the geometry will be displayed in the workspace.
- If the wireframe shows low resolution (instead of circle it will display hexagon), >ctrl+I> settings> resolution> below cons> change perimeter length > 1 (if not recovered, value can be lowered but greater than 0)> ok> ok.
- To enable the surfaces, enable blue cube button in first row.
- Black buttons in the first row are Boolean operations.
- To delete unwanted and over lapping faces, select faces> delete from geometry module window.
- To create a face or a surface, select coons> surface.
- After improving the geometry quality, switch to mesh.
- Select mesh generation> best> visible.
- Mesh will be generated
- Press f11> shells> skewness> fluent> 0.7>ok
- File> output> fluent> select destination folder> save
- File> save as> select destination folder> save

Processing

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 iteration are similar to previous iteration, means not significant changes between iterations starts to occur. This step is a secondary step to be followed to generate volume mesh and to apply boundary conditions for the analysis.

- Open the fluent software.
- In the startup menu- check on – double precision > check on > meshing mode> ok
- File > read> mesh> browse for the surface mesh file> ok.
- Mesh> tet> click init & refine button.
- Volume mesh will be generated.
- File> write> mesh> save to a folder.
- Select switch to solution button below the file menu.
- Display> grid> select wall surfaces> click display.
- In general tab>Select scale button> view geometry in > mm > mesh was generated in > mm> scale> ok
- In model tab> double click> energy> check energy equation> ok [6].

- Double click> viscous> select k-epsilon(2 equation) in model > realizable in k-epsilon model> standard wall function> ok
- In materials tab> choose the fluid and solid material required > ok
- In boundary conditions, select inlet domain, change it to required inlet property> input the required magnitude> select the vector direction> ok
- Likewise all the boundary values are to be defined.
- Select solution initialization> select hybrid initialization> initialize.
- After initialization iterations are completed.
- Select > run calculation> set no of iteration to 1000(Approximately)> click calculate.

The solver begins to solve the problem based on the input condition.

Post Processing

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.

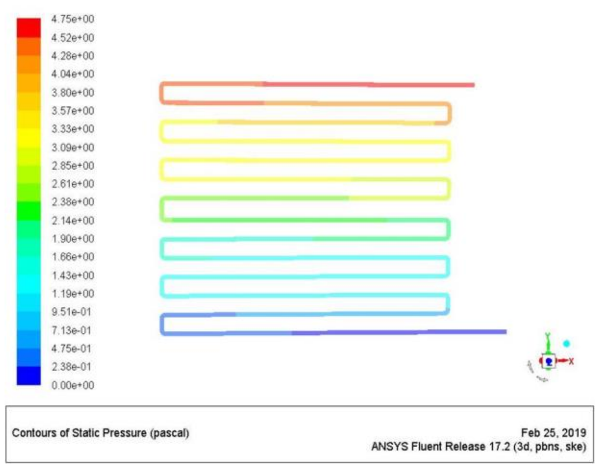
After the solution is done, the results are to be obtained.

- Select graphics and animation from results tab.
- Click sweep surface from animation tab> select the sweep axis> enter 1 for required axis and rest with 0 > create.
- Select contours> check filled> select which contours need to be viewed> select sweep surface created> display.

RESULTS

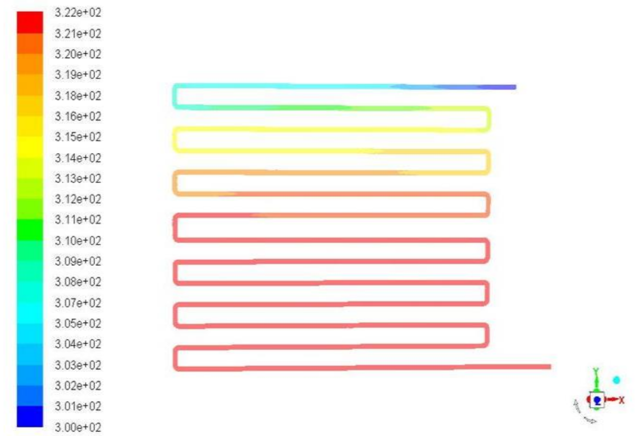
For carrying out this analysis, the mass flow rate has been kept constant and only the magnitude of solar insolation has been varied. It must be noted that, the intensity of heat flux is proportional to the temperature along the tube surface. As the temperature increases, the heat gain by the fluid from the tube surfaces increases. This in turn, helps to raise the water temperature within the storage tank. It can be noted that, for a heat flux of 600 W/m2 the water temperature rises up to 341.42 K. On the other hand, water gets heated to 352.75k, when subjected to a solar insolation of 1000 W/m2.

ZIGZAG PRESSURE



Area-Weighted Average Static Pressure	(pascal)
inlet	4.7514242
outlet_int	0.036015872
Net	2.4270258

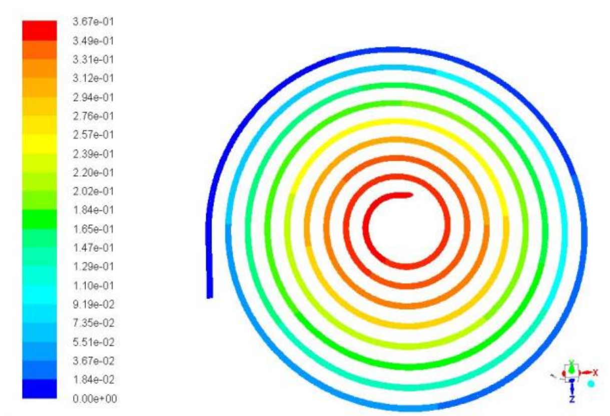
ZIGZAG TEMPERATURE



Contours of Static Temperature (k)
Feb 25, 2019
ANSYS Fluent Release 17.2 (3d, pbns, ske)

Area-Weighted Average Static Temperature	(k)
inlet	300.00033
outlet_int	341.42437
Net	320.41977

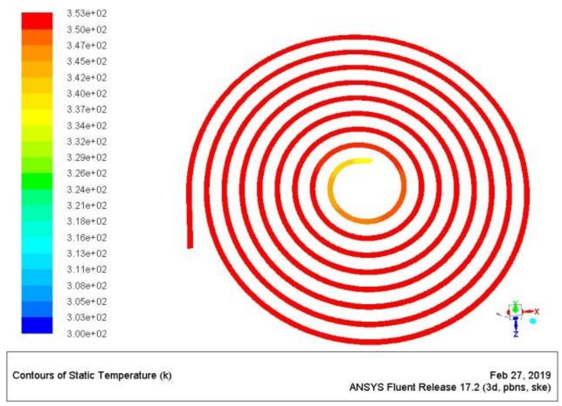
SPIRAL PRESSURE



Contours of Static Pressure (pascal)
Feb 27, 2019
ANSYS Fluent Release 17.2 (3d, pbns, ske)

```
          Area-Weighted Average
                Static Pressure              (pascal)
------------------------------ --------------------
                          inlet            0.36725645
                     outlet_int          0.0038815973
               ---------------- --------------------
                            Net            0.18825885
```

SPIRAL TEMPERATURE




```
          Area-Weighted Average
             Static Temperature                   (k)
---------------------------- --------------------
                        inlet                  300
                   outlet_int            352.75323
             ---------------- --------------------
                          Net            325.98611
```

Two cases of the solar insolation (i.e. 600 W/m2 and 1000 W/m2) at a fixed mass flow rate. It can be noticed that at a constant mass flow rate, the performance of the collector is better at higher solar intensities than at lower incident solar insolation over the tube surfaces.

COMPARISON OF ZIGZAG & SPIRAL TUBE PIPE

TYPE OF SHAPE	SPIRAL	ZIG ZAG
OUTLET TEMPERATURE(k) To @600W/mm^2	341.42	321.92
OUTLET PRESSURE(Pa) Po @ 600W/mm^2	0.00388	0.0360
OUTLET TEMPERATUR(k)To@ 1000W/mm^2	352.75	341.46
OUTLET PRESSURE(Pa) Po @ 1000W/mm^2	0.00388	0.0360

CONCLUSION

By realizing this type of solar heater we could noticed that not invested so much time nor a high amount of money, which helps us to fulfill one of the objectives that is to approach it to communities with few resources because as the theory says, there are places where solar heaters are used for domestic routines or in agriculture. Hence, copper is a material that we can easily find and through this project we prove that it is an economic material and also conducts the heat well enough to heat the necessary water. This study set out to better understand how the performance of a spiral and zigzag copper tube is affected by flow rate as well as the incident solar insolation. Hence, at a lower flow rate and at a higher incident solar radiation, the outlet water temperature obtained was found to be higher when compared with other possible combinations. CFD analysis is an effective tool with which we can stimulate the models on various operating conditions without actually fabricating them and can compare their results. The best solution is then selected for fabrication so that we can save time and money.

REFERENCES

[1] Vilet GC. Natural convection local heat transfer on constant-heat flux inclined surfaces. Journal of Heat Transfer. 1969 Nov; 91(4), 511–6. Crossref

[2] Vilet GC, Liu KC. An experimental study of turbulent natural convection boundary layers. Journal of Heat Transfer. 1969 Nov; 91(4):517–31. Crossref

[3] Shitzer A, Kalmanoviz D, Zvirin Y. Grossman G. Experiments with a flat plate solar water heating system in thermosyphonic flow. Solar Energy.1978 Aug; 22(1):27–35. Crossref

[4] Celentano R, Kirchner R. An experimental study on once through thermosiphon system. Transactions of ASME. 1988 May; 110(2):90–7. Crossref

[5] Huang BJ. Development of long-term performance correlation for solar thermosyphon water heater. Transactions of ASME. 1989 May; 111(2), 124–31. Crossref

[6] Bannerot RB, Yi-Wey T, Scott A, Placke G, Poche T. A simple device for monitoring flow rates in thermosyphon solar water heaters. Journal of Solar Engineering. 1992 Feb; 114(1):47–52. Crossref