International Journal of Scientific Engineering and Research (IJSER)
www.ijser.in
ISSN (Online): 2347-3878
Volume 3 Issue 4, April 2015

Analyze the Effect of Variations in Shape of Tubes for Flat Plate Solar Water Heater

Vishal G. Shelke[1], Prof. Chinmay V. Patil[2]

. [1]PG Scholar, Department of Mechanical Engineering,
Shri Sant Gajanan Maharaj College of Engineering, Shegaon, Dist-Buldana, (M.S.), India-444203

[2]Assistant Professor, Department of Mechanical Engineering
Shri Sant Gajanan Maharaj College of Engineering, Shegaon, Dist-Buldana, (M.S.), India-444203

Abstract: ***Solar water heater is a very simple device and efficient way to absorb energy from the sun rays and use it. Therefore improvement in their operating condition & geometrical would definitely result in saving conventional fuel and cost. The objective of this study was to analysis the effect of variations in shape of tubes for flat plate solar collector. For this study circular tube of 12.7 mm diameter was considered and numerical analysis is carried out with ANSYS CFD FLUENT software. Comparison in inlet and outlet temperature was performed for different heat flux. Further analysis was carried out for different elliptical tube shapes. Outlet temperature of water is compared with circular results and there is a good agreement.***

Keywords: Solar energy collector, CFD simulation, Flat Plate Solar Water Heater, Shape of Tubes

1. Introduction

The solar energy collection as a renewable energy topic has been the primary interests of many engineers and researchers for the last two centuries due to its wide applications such as domestic water heating systems. Today, solar water heating systems are being used for single family houses, apartment buildings, schools, car washes, hospitals, restaurants, agricultural farms and different industries. Solar water heating can reduce domestic water heating costs by as much as 70%. Owners of these buildings have found that solar water heating systems are cost-effective in meeting their hot water needs all over the year. A more intensive attention was given to this topic from 1970s of the last century, particularly, when the worldwide crisis of 1973 has taken place. Since then, the efficiency of solar heating systems and collectors has improved. The efficiencies can be attributed to the use of low iron, tempered glass for glazing (low-iron glass allows the transmission of more solar energy than conventional glass), improved insulation, and the development of durable selective coatings. Thus due to its importance, a flat plate solar collector with and without cover glass was analyzed using computational fluid dynamics (CFD) soft-ware and simulated without water flow.

1.1 Flat-plate collectors

Flat-plate collectors are the most common solar collector for solar water-heating systems in homes and solar space heating. A typical flat-plate collector is an insulated metal box with a glass or plastic cover (called the glazing) and a dark-colored absorber plate. These collectors heat liquid or air at temperatures less than 80°C.

Mohamed Selmi, Mohammed J. Al-Khawaja , Abdulhamid Marafia [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°C over all times. Also, the simulated temperature curve has the same behaviour as that experimental one and they are close to each other.

Nomenclature			
h	heat transfer coefficient, W/m^2 °C	^{T}out, sim	simulation water outlet temperature, °C and K
n	number of iterations	U	velocity component in x-direction, m/s
p	pressure, N/m^2	V	velocity component in y-direction, m/s
q	solar heat flux, W/m^2	W	velocity component in z-direction, m/s
R	residuals		
T	temperature, °C and K	Greek letters	
^{T}in	water inlet temperature, °C and K		fluid density, kg/m^3
		r	

ISSN (Online): 2347-3878
Volume 3 Issue 4, April 2015

Marroquín-De Jesus Angel, Olivares-Ramirez Juan Manuel, Zamora-Antuñano Marco Antonio [2], present the investigation describes the construction and experimentation of solar energy absorbers using water as fluid and its simulation in *Computational Fluid Dynamics* (CFD). 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. In the experimental data for absorber A the highest variation is noted in the middle of the fluid path with values of 347 K, 346 K and 345 K for three different channels. For equivalent locations in Absorber B the results were 345.5 K, 344.5 K and 344.5 K for three different pipes.

S.Eswaran, M.Chandru, M.Vairavel, R.Girimurugan [3], 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 / $m^{2.}$ 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 / m^2, dynamic pressure of 4.30×105 Pa, flow velocity of 5.91×105 l/m^2 & relative temperature of 360° K.

Manjunath M.S, K. Vasudeva Karanth, and N. Yagnesh Sharma [4] , In this paper, an attempt is made to explain in a comparative way the effect of surface geometry of solar collector having dimple geometry with that of a flat plate solar collector of the same size. A CFD analysis was carried out for the two cases, subjected to a constant heat flux of 600W/m2 and 1000W/m2. It can be inferred from the study that the absorber plate temperature shows a rise of average surface temperature of about 50C for the dimple solar collector when compared to a flat plate solar collector. Most importantly, the average exit water temperature shows a marked improvement of about 5.50C for a dimple solar collector as compared to that of a flat plate solar collector.

From the CFD analysis carried for the full three dimensional absorber plate assemblies for plates with and without dimple establishes that with surface geometry enhancements such as having a dimple pocket increases the heat transfer to the absorber tube due mainly to the increase in area for diffusion heat transfer.

Prof. P.W.Ingle, Dr. A. A. Pawar, Prof. B. D. Deshmukh, Prof. K. C. Bhosale [5], this thesis attempts to present numerical simulation of solar collector developed exclusively for grape drying. Solar drying of grapes is much feasible technically and economically. There has been a remarkable achievement in solar drying of grapes due to sustained research and development associated with the adoption of advanced technologies. In the present thesis the computational fluid dynamics (CFD) tool has been used to simulate the solar collector for better understanding the heat transfer capability. 3D model of the collector involving air inlet, wavy structured absorber plate, glass cover plate, and pebble block is modeled by ANSYS Workbench and the unstructured grid was created in ANSYS ICEM. The results were obtained by using ANSYS FLUENT software. It is found from the CFD analysis that the flow of air in the solar flat plate collector is not properly distributed. In order to overcome this issue author had suggested introducing baffles at the inlet of collector which improves the efficiency of solar flat plate collector.

The CFD analysis of the flow and heat transfer in flat plate solar collectors is computationally quite difficult and the number of research works on this subject is quite low.

2. Problem Statement

The objective of present study is to perform CFD simulation for solar water collector. The results obtained by ANSYS Workbench simulation are been validated with numerical results of Mohamed Selmi [1] which done by Computational Fluid Dynamics Research Corporation (CFDRC) software. The overall aim of this work is to understand the temperature distribution of water inside the solar collector and compare the outlet temperature of water with numerical results of Mohamed Selmi.

In this paper, CFD Simulation are done for two shapes,

1. Circular tube
2. Elliptical tube

1. Circular Tube

The Project model consist of aluminium plate of 1500 mm long, 166 mm wide and 1 mm thick used as an the absorber, fixed to it from the top a copper tube of ½ inch (12.7mm) diameter. Copper tube of 1500 mm long and 1 mm thick.

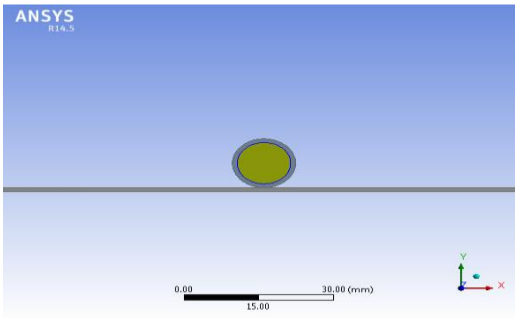


Figure 1: Model geometry of circular tube

2. Elliptical Tube

The Project model consist of aluminium plate of 1500 mm long,166 mm wide and 1 mm thick used as an the absorber, from the top a copper tube for different major and minor axis of elliptical shape as shown in table 1. The copper tube of 1500 mm long and 1 mm thick.

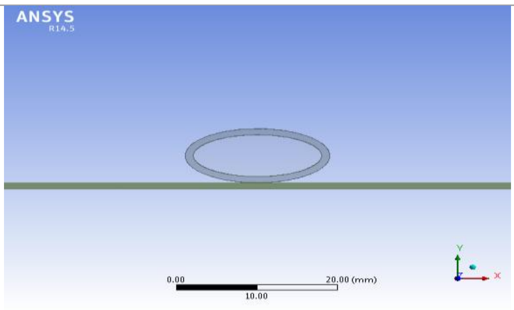


Figure 2: Model geometry of elliptical tube of case no 5 i.e. B = 0.5 A

Table 1: Values of A and B for Different Case

case no	*Relation of A with B*	*Value of A*	*Value of B*	*Value of Major axis= 2A*	*Value of Minor axis= 2B*
1	B = 0.1 A	20.08	2.008	40.16	4.016
2	B = 0.2 A	14.199	2.898	28.398	5.676
3	B = 0.3 A	11.593	3.478	23.1866	6.956
4	B = 0.4 A	10.04	4.016	20.08	8.032
5	**B = 0.5 A**	**8.98**	**4.49**	**17.96**	**8.98**
6	B = 0.6 A	8.2	4.92	16.4	9.8407
7	B = 0.7 A	7.585	5.31	15.171	10.6244
8	B = 0.8 A	7.087	5.67	14.175	11.35
9	B = 0.9 A	6.6933	6.024	13.3866	12.045

3. Numerical Simulation by Software

Computational system dynamics is the analysis of the systems involving fluid flow, heat transfer and associated phenomenon such as chemical reactions by means of computer-based simulation. The technique is very powerful and spans a wide range of industrial and non-industrial applications areas. Some examples are: aerodynamics of aircrafts and vehicles, hydrodynamics of ships, combustion, turbo machinery, electrical and electronic engineering, and chemical process engineering, external and internal environment of buildings, marine engineering, environmental engineering, hydrology and oceanography, metrology, biomedical engineering etc. from the 1960s onwards, the aerospace industry has integrated CFD technique into design, R & D and manufacture of aircrafts and jet engines. More recently the methods have been applied to the design of internal combustion engines, combustion chambers of gas turbines and furnaces. Furthermore, motor manufacturers now routinely predict drag forces, under bonnet airflow and the in-car environment with CFD. Increasingly CFD is becoming a vital component in the design of industrial products and processes.

The ultimate aim of development in the CFD field is to provide a capability comparable to other CAE (Computer-Aided Engineering) tools such as stress analysis codes.

The main reason why CFD has lagged behind is the tremendous complexity of the underlying behaviour, which precludes a description of the fluid flows this is at the same time economical and sufficiently complete. The availability of affordable high performance computing hardware and the introduction of user friendly interference have led to a recent upsurge of interest and CFD is poised to make an entry into the wider industrial community in the 1990s. Clearly the investment costs of a CFD capability are not small, but the total expense is not normally as great as that of a high quality experimental facility.

Moreover, there are several unique advantages of CFD over experimental-based approaches to fluid systems design.

1. Substantial reduction of lead times and costs of new design.
2. Ability to study systems where controlled experimental are difficult or impossible to perform. (e.g. very large systems)
3. Ability to study systems under hazardous conditions at and beyond their normal performance limits. (e.g. safety studies and accident scenarios).
4. Practically unlimited level of detail of results.

In contrast CFD codes can produce extremely large volumes of results at virtually no added expense and it is very cheap to perform parametric studies, for instance to optimize equipment performance **[6].**

A. Basics in CFD

CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. In order to provide easy asses to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. Hence all code contains three main elements:

1. Pre-processor
2. Solver
3. Post-processor

B. Numerical Modelling of Solar Water Collector

The procedure adopted to simulate the solar water collector by CFD tool is as follows:

1. The 3D model is been modelled by using ANSYS WORKBENCH software as shown in Fig.3 and 4
2. After creation of 3D model, the unstructured grid is been created by using ANSYS ICEM software as shown in fig 5 and fig.6
3. The unstructured grid created consists around 7 lakh element in circular tube and 50 lakh elements in elliptical tube.
4. The unstructured grid which is created then imported in ANSYS FLUENT software and the experimental conditions are used while simulating the solar water collector.
5. The model was defined by using 3D segregated solver with steady condition, energy equation, and laminar model.

ISSN (Online): 2347-3878
Volume 3 Issue 4, April 2015

6. The fluid chosen to simulate solar collector is water. The water properties used in this simulation is shown in table no.2
7. The material used for absorber plate is aluminium and for tube copper. The aluminium and copper properties used in this simulation is shown in table no.2
8. After this give cell zone condition to different zone of plate, tube and water fluid.
9. The boundary conditions are as follows.
   Inlet velocity = 0.0170484 m/s
   Inlet temp and Heat Flux change with time shown in table no.3, and fig.7 shows Measured total solar radiation on 7 December 2004, in Doha city**. [1]**
10. After setting all boundary conditions in fluent software, to solve the numerical equations the initialization by inlet is to be done.
11. To visualize the residuals of iterations verses convergence limit, the residual monitor is set to be in ON state condition.
12. To get the final results the numbers of iterations are set around 200. The results for these simulations were converged at around 50 to 60 iterations.
13. As the numbers of elements are more to get the converged results the time taken for these simulations will be more with single processor.
14. Finally after getting the proper converged results the water flow distribution and heat transfer inside the solar water collector is been plotted in the form of Contour plots.
15. The outlet temperature is been calculated from ANSYS FLUENT after getting converged results and been compared with the experimental results.

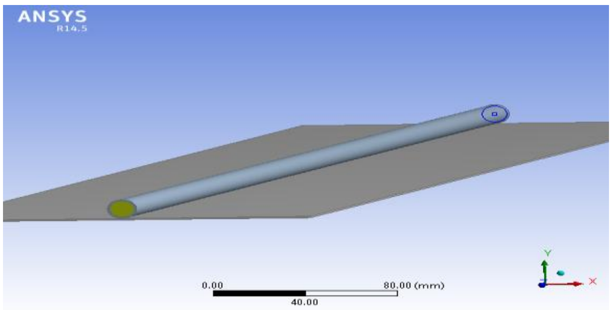


Figure 3: 3D model of solar water collector of circular tube visualizing the absorber plate and water domain

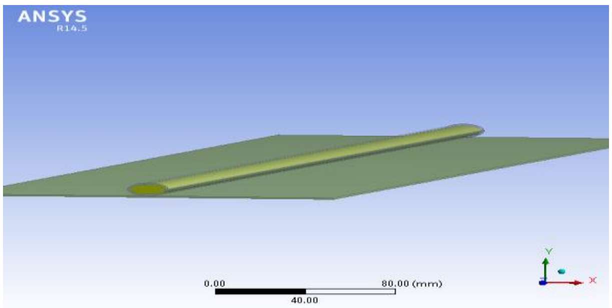
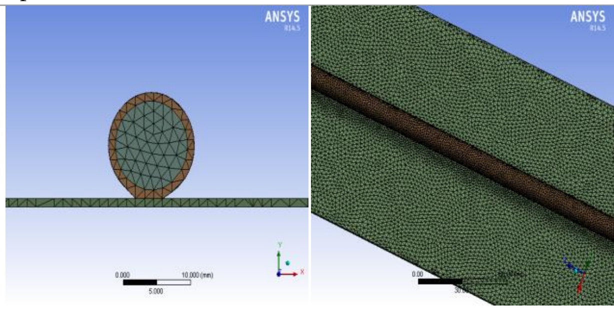
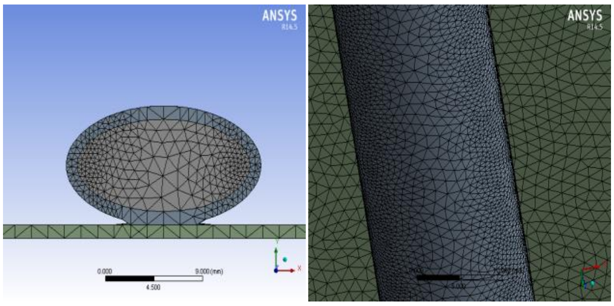


Figure 4: 3D model of solar water collector of elliptical tube visualizing the absorber plate and water domain



a) Front View b) Top View
Figure 5: Meshing by using ANSYS FLUENT of Circular Tube



a) Front View b) Top View
Figure 6: Meshing by using ANSYS FLUENT of Elliptical Tube of case no 5 i.e. B=0.5 A

Table 2: Volume condition setting for properties of blocks

Properties	*Water block*	*Al plate*	*Cu pipe*
Density (kg/m^3)	998.2	2770	8800
Specific heat (J/kg °C)	4230	875	420
Thermal conductivity (W/m °C)	0.569	177	401

Table 3: Varying Heat flux and inlet temp of water

Time	*Heat Flux in W/m^2*	*Inlet Temp of water in K*
9.31 am	470	306
10.15 am	520	311
11.15 am	535	316
11.45 pomp	540	318
12.35 p.m	535	317
13.15p.m.	520	316

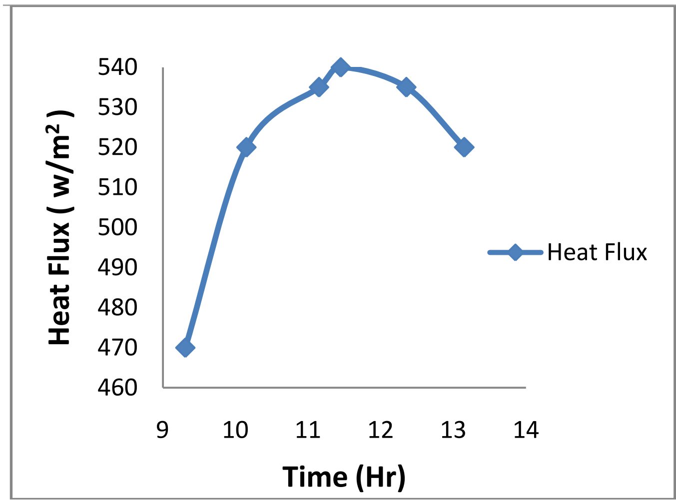


Figure 7: Measured total solar radiation on 7 December 2004, in Doha city. [1]

4. Result and Discussion

The results obtained from the CFD analysis of solar flat plate collector are presented in this section. The simulation is carried out for different times of the day i.e.9.30 am to 1.15 pm. Result obtained by simulation for circular pipe as shown in fig.8.Also result obtained for elliptical pipe for different cases are shown in fig.9 to fig 15.

Then the results obtained by circular compared with best result obtained elliptical as shown in fig 16 and in table 4. The curves are plotted to indicate simulated outlet temperatures of circular pipe and elliptical pipe versus time. From fig.16 it seems that the difference between simulated outlet temperature of circular pipe and elliptical pipe for different times is almost 4.2˚C. The peak outlet temp was obtained in elliptical case for case no 5 i.e. B = 0.5 A. as shown in fig.17.

As in Mohamed Selmi et al. [1] seems the temperature difference between the water inlet and outlet is almost 9 °C over all times. But in elliptical case temperature difference between the water inlet and outlet is almost **14 °C** over all times. That means elliptical tube gives 5 °C more than circular tube.

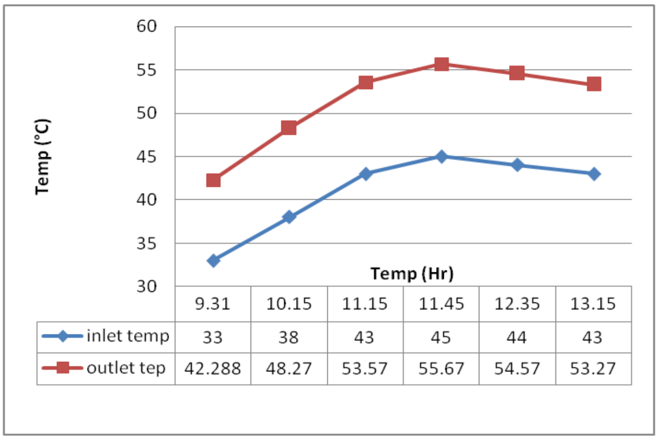


Figure 8: Difference in Inlet & Outlet for Circular Pipe

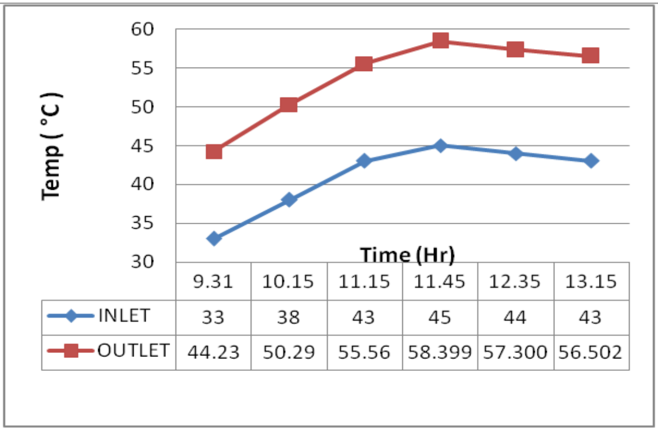


Figure 9: Difference in Inlet & Outlet for Case No.3 i.e. B=0.3 A

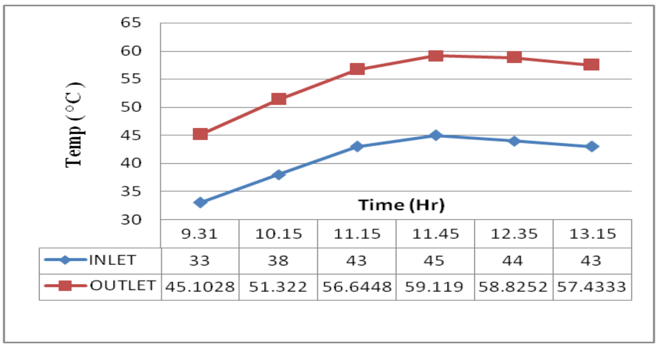


Figure 10: Difference in Inlet & Outlet for Case No.4 i.e. B=0.4 A

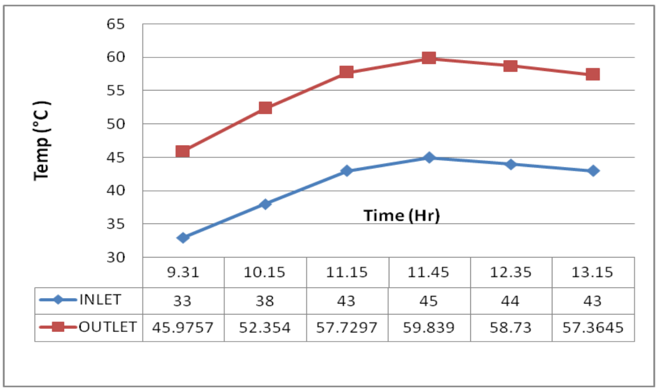


Figure 11: Difference in Inlet & Outlet for Case No.5 i.e. B=0.5 A

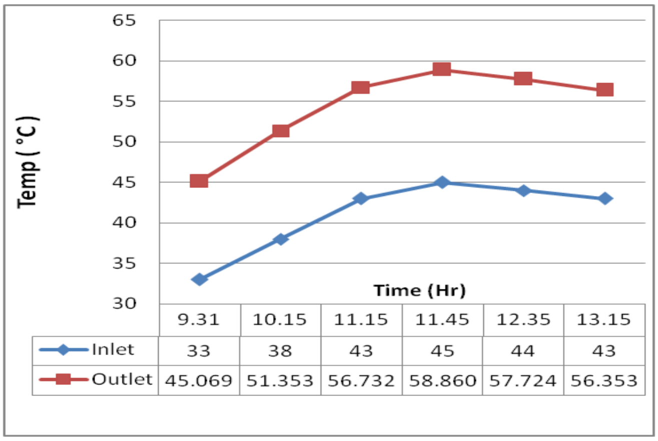


Figure 12: Difference in Inlet & Outlet for Case No.6 i.e. B=0.6 A

ISSN (Online): 2347-3878
Volume 3 Issue 4, April 2015

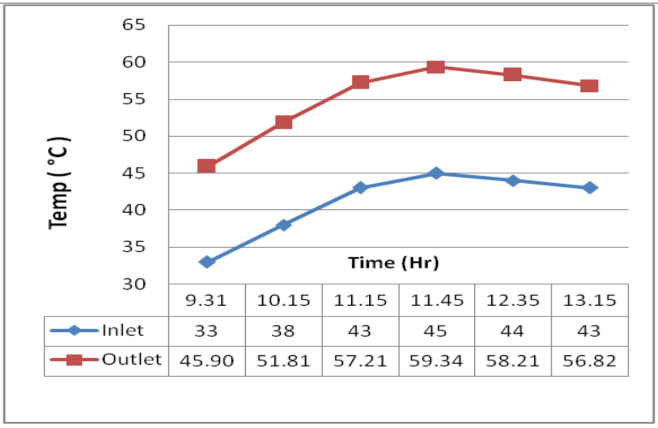


Figure 13: Difference in Inlet & Outlet for Case No.7 i.e. B=0.7 A

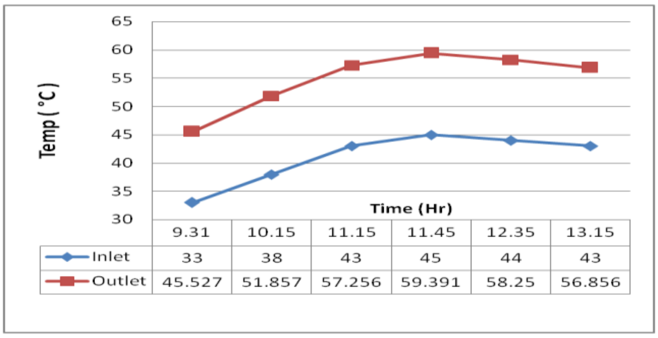


Figure 14: Difference in Inlet & Outlet for Case No.8 i.e. B=0.8 A

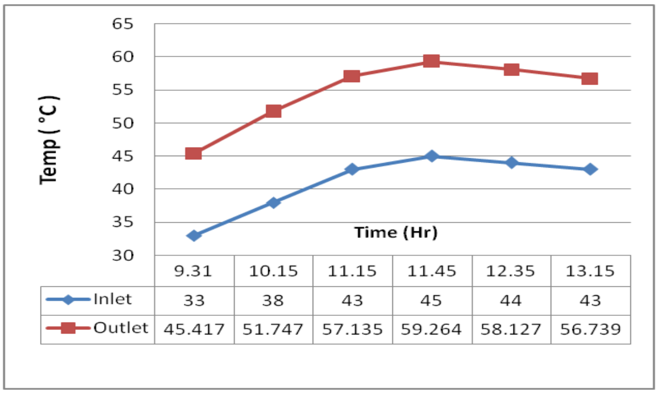


Figure 15: Difference in Inlet & Outlet for Case No.9 i.e. B=0.9 A

Table no 4: Comparison between Inlet and Outlet Temp of water for Circular & Elliptical Tube of case no 5 **i.e. B = 0.5 A**

Time	*Inlet Temp in °C*	*Heat Flux in w/m2*	*Outlet Temp of Circular Tube in °C*	*Outlet Temp of Elliptical Tube for case 5 i.e. B = 0.5 A in °C*	*Difference in Elliptical & Circular Tube in °C*
9.31	33	470	42.288	45.9757	3.6877
10.15	38	520	48.27	52.354	4.084
11.15	43	535	53.57	57.7297	4.1597
11.45	45	540	55.67	59.839	4.169
12.35	44	535	54.57	58.73	4.16
13.15	43	520	53.27	57.3645	4.0945

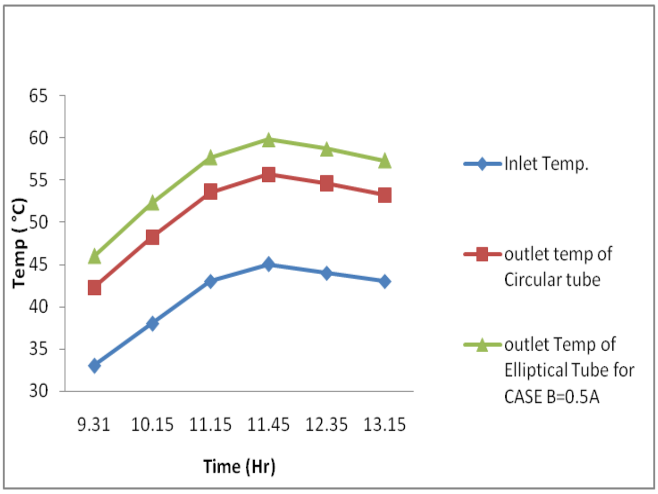


Figure 16: Comparison between Inlet and Outlet Temp of water for Circular & Elliptical Tube of case no 5 i.e. B = 0.5 A

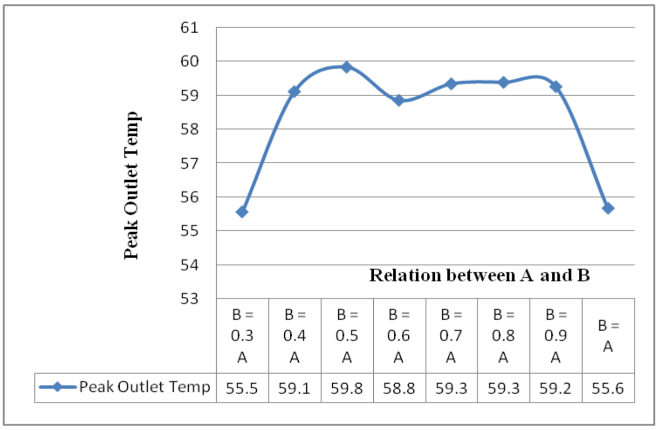


Figure 17: Peak Outlet Temp vs. Relation between A and B

Also the temperature distributions are obtained by CFD simulation. The contour plots obtained for temperature distribution in streamlines of circular and elliptical tube are shown in Figure 18 and Figure 19.

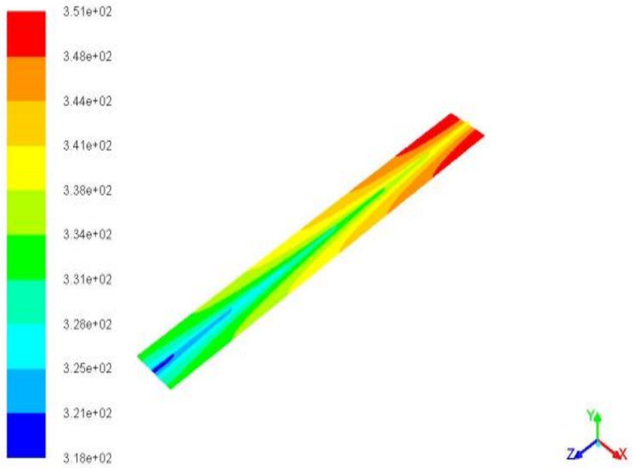


Figure 18: Streamlines for temperature distribution of Circular tube at 11.45 am of the day

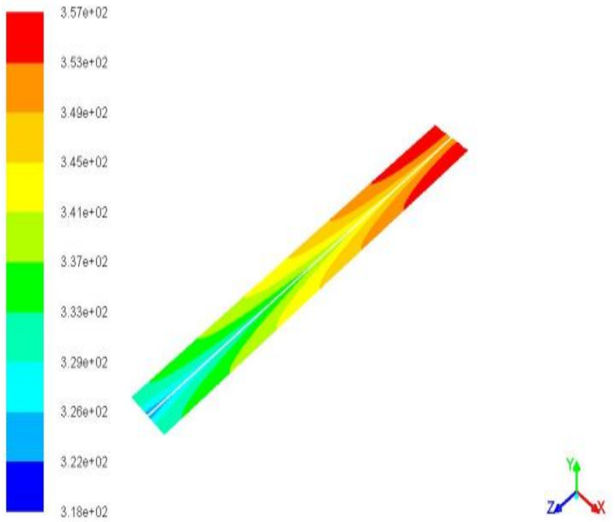


Figure 19: Streamlines for temperature distribution of Elliptical tube at 11.45 am of the day

5. Conclusion

From the above study it is concluded that elliptical tube of case no. 5 (i.e. B=0.5A) gives the maximum outlet temperature of water for the same heat flux and inlet temperature in comparison with circular and other elliptical geometries. It also shows the peak outlet temperature difference between circular and elliptical tube is 4.17 ˚C. This shows that elliptical tube is beneficial in future for domestic purpose.

References

[1] Mohamed Selmi, Mohammed J. Al-Khawaja and Abdulhamid Marafia, "Validation of CFD simulation for flat plate solar energy collector," Renewable Energy 33 (2008) 383–387.
[2] Marroquín-De Jesus Angel, Olivares-Ramirez Juan Manuel, Zamora-Antuñano Marco Antonio, Jiménez-Sandoval Omar and Encinas-Oropesa Armando, "Analysis of Flow and Heat Transfer in a Flat Solar Collector with Rectangular and Cylindrical Geometry Using CFD," Ingeniería Investigación y Tecnología, volumen XIV (número 4), octubre-diciembre 2013: 553-561.
[3] S.Eswaran, M.Chandru, M.Vairavel, R.Girimurugan, "Numerical Study on Solar Water Heater using CFD Analysis," International Journal of Engineering Sciences & Research Technology [Eswaran, 3(3): March, 2014, [1485-1489].
[4] Manjunath M.S, K. Vasudeva Karanth, and N. Yagnesh Sharma, "A Comparative CFD study on Solar Dimple Plate Collector with Flat Plate Collector to Augment the Thermal Performance," World Academy of Science, Engineering and Technology Vol:6 2012-10-21.
[5] Prof. P.W.Ingle, Dr. A. A. Pawar, Prof. B. D. Deshmukh, Prof. K. C. Bhosale, "CFD Analysis of Solar Flat Plate Collector," International Journal of Emerging Technology and Advanced Engineering, ISSN 2250-2459,Volume 3, Issue 4, April 2013.
[6] Fabio Struckmann, "Analysis of a Flat-plate Solar Collector", 2008 MVK160 Heat and Mass Transport, May 08, 2008, Lund, Sweden.

Author Profile

Vishal G. Shelke is a student of Master of Engineering in, Advance Manufacturing and Mechanical System Design Shri Sant Gajanan Maharaj College of Engineering, Shegaon, of (M.S.) India. He received a degree of Bachelor of Engineering in Mechanical Engineering from Sant Gadge Baba Amravati University, Amravati, (M.S.), India

Prof. Chinmay V. Patil is currently holding a position as Assistant Professor in Mechanical Engineering Department of S.S.G.M. College of Engineering, Shegaon (M.S.) India. He has 10 years of experience in academics. His research interests include Computer Aided Design and Manufacturing and Solar Energy Utilization.