Int. J. Chem. Sci.: 14(4), 2016, 2259-2264
ISSN 0972-768X
www.sadgurupublications.com

MODELLING AND VALIDATION OF SOLAR FLAT PLATE WATER HEATING SYSTEM SUBJECTED TO VARYING ABSORBER GEOMETRIES

M. DINESH BABU*, M. VENKATA RAMANAN and A. GANAPATHI

Institute for Energy Studies, College of Engineering Guindy Campus, Anna University, CHENNAI – 600025 (T.N.) INDIA

ABSTRACT

Increasing the surface area will increase the heat transfer capability of the collector. So an attempt was made to investigate the effects on collector efficiency of a solar water heater with fins attached internally in the riser tube. Then analyzing the solar collector using the Computational Fluid Dynamics (CFD) with and without internal fins so as to simulate the solar collector for better understanding of the heat transfer capabilities of the collector. The conjugate heat transfer phenomenon between collector and water is modelled using FLUENT CFD software. The solar radiation is considered as the heat flux while calculating boundary conditions for the collector area. The geometric model and fluid domain for CFD analysis is generated using ANSYS Design Modeller software, Grid generation is accomplished by ANSA meshing software. The results obtained from the CFD for tube with fin is validated with the CFD and experimental values for plain tube and found that there is an increase in temperature of about 3-4^{o}C for finned tube. Based on this, fabrication is done and experimental analysis is carried out with new set of collectors. The results obtained from experimental analysis shows that there is an increase in efficiency of about 3-5 % when we go for finned tube rather than plain tube.

Key words: CFD-Computational fluid dynamics, FPC-Flat plate collector, LPD-Litres per day.

INTRODUCTION

The quality of the solutions obtained from CFD simulations are largely within the acceptable range proving that CFD is an effective tool for predicting the behavior and performance of a solar air heater. One of the great challenges in the design of a solar air heater using CFD approach is the selection of appropriate turbulence model. The decision about a suitable turbulence model chosen in a CFD computation is not easy. In this article, a CFD investigation is also carried out to select best turbulence model for the design of a solar

*Author for correspondence; E-mail: dinesh198014@yahoo.com

air heater. A two-dimensional flow is assumed. The influences of the five different turbulence models on the quality of the obtained results are tested. However, for smooth solar air heater, effective efficiency is found to decrease for entire range of temperature rise parameter. The performance of solar collectors using different geometries and by using internal finning is studied[1-5] and result showed a better heat transfer enhancement which in turn improves the efficiency of the collector. Validation of CFD Simulation for Flat Plate Solar Energy Collector is done and the problem of flat plate solar energy collector with water flow is simulated and analyzed using computational fluid dynamics (CFD) software and experimental model was built and experiments were performed to validate the CFD model. It is found that there a good agreement between the experimental and simulated results for outlet water temperatures. The efficiency of the absorber tube can be improved by increasing the overall surface area, which increases the heat transfer to the working fluid. Analyses of the same have been done to see to what degree the insertions cause an improvement in heat transfer and as a result in increasing the outlet temperature of the working fluid. This has been carried out numerically by commercial CFD code Ansys CFX 12.0. The analysis has been carried out to study the effect of heat transfer in absorber tubes with and without insertions. The study also takes care in distributing different heat flux along the walls of the absorber tubes.

EXPERIMENTAL

Design of 25 LPD solar flat plate collector

A 25 LPD experimental collector set up is fabricated that includes riser tubes with Heat Exchanger as primary loop and hot water storage tank as secondary loop. The secondary circuit is fabricated with 25 litre insulated tank. The schematic diagram is shown in Fig. 1.

Fig. 1: 25 LPD collector with primary riser tube and secondary storage tank

The energy balance of the absorber plate yields the following equation for steady state q_u = (Ap S) – q_l = m LPD- litres per day, C_p ΔT q_u = Useful heat gain, i.e. the rate of heat transfer to the working fluid, S = Incident solar flux absorbed in the absorber plate/m, A_p = Area of absorber plate (m^2), q_l = Rate at which heat is lost by convection and re-radiation from the top and by convection from the bottom and sides.

Therefore, the area of plate $A_p = \left[\frac{m\,Cp\,\Delta T}{S\,\eta}\right] = \left[\frac{25 \times 4187 \times 35}{680 \times 60 \times 60 \times 7 \times 0.4}\right] = 0.534\ m^2$.

For this collector area, available length (L) & breadth (W) are calculated as the effective length of the collector is 0.945 m & Effective Width of the collector plate is 0.535 m with 4 riser tubes placed in 100 mm pitch each other.

Model of the riser tube by Pro-E

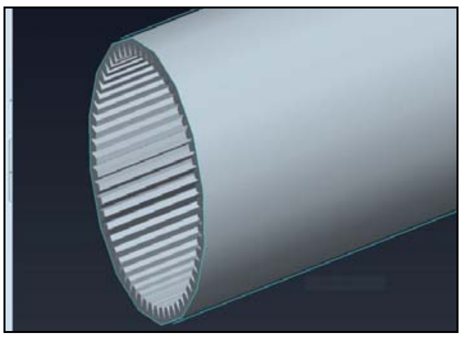

Fig. 2: Riser tube with fins

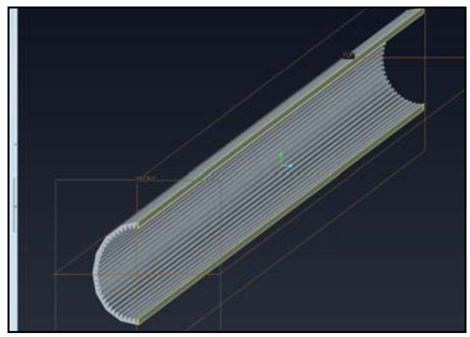

Fig. 3: Cross-sectional view riser tube

Meshing the model using ANSA for CFD

The meshes are built up of tetrahedral cells and are thus unstructured meshes. In order to better predict the internal flow field behavior, the optimized solution-adaptive mesh refinement is used.

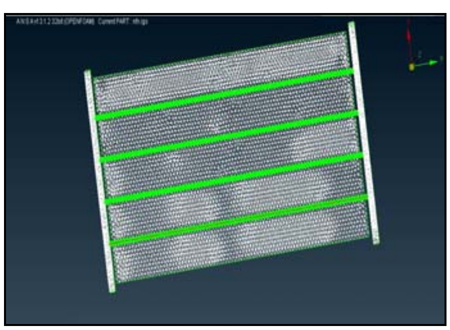

Fig. 4: Meshed model of the system

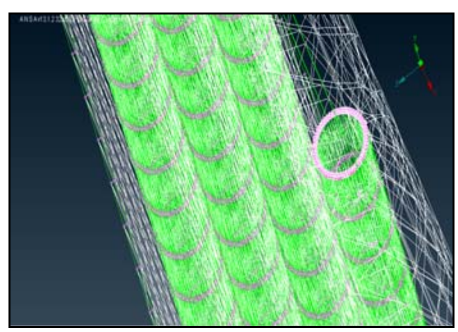

Fig. 5: Meshed view of the riser tubes

More cells are added at locations where significant flow changes are expected, for example near the inlet/outlet ports, along the length of the fin. The resulting mesh enabled the features of the flow field to be better represented. The symmetric solver selected here accounts for the three-dimensional effects, and the calculation domain was half of the physical body, based on symmetry considerations. The model which is in IGS format is imported to ANSA for meshing.

RESULTS AND DISCUSSION

Solution convergence and temperature distribution along the collector

The model, which is designed is simulated using ANSYS Fluent. A constant 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. The solution of the CFD analysis is converged. The temperature distribution along the Solar Flat Plate collector obtained using CFD analysis is shown below.

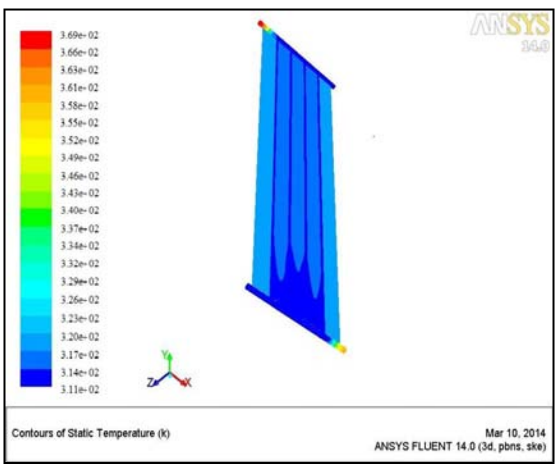

Fig. 6. Temperature distribution along riser tube of the collector

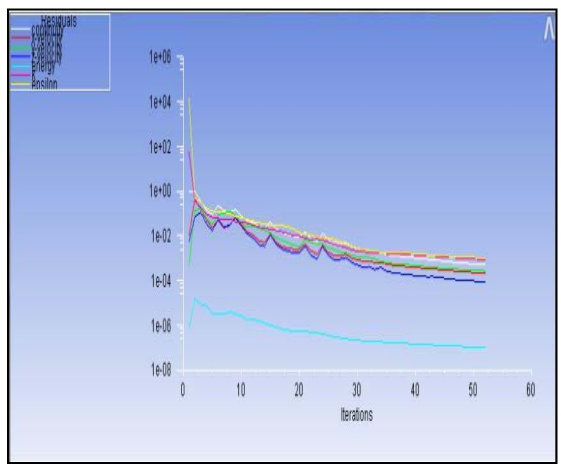

Fig. 7. Convergence Diagram of SFC by using ANSYS fluent

Validation of the experimental results with CFD

The outlet temperature obtained from the CFD simulation is validated with the experimental outlet temperature and error analysis is done.

Table 1: Comparison table for CFD outlet temperature for with and without fin

Time (hrs)	Inlet temp. (K)	Solar insolation (W/m^2)	Experimental outlet temp. (K)	CFD outlet temperature (K)	CFD outlet temperature with fin (K)
8	306	249	307.487	309.13629	309.85249
9	313.437	362	316.081	318.17087	318.92461
10	318.319	582	324.226	327.44142	328.39540
11	322.278	772	328.718	330.68714	331.28049
12	325.333	869	332.698	334.96724	336.53982
13	328.795	886	333.437	336.64287	338.50398
14	329.274	903	335.078	339.37584	341.5648
15	328.874	738.15	333.281	335.22073	336.59207

Outlet temperature of SFPC: Experimental vs Modelled results

The experimental outlet values of Solar Flat Plate Collector are compared with the CFD values. Graphs between time vs temperature and solar insolation vs temperature are plotted.

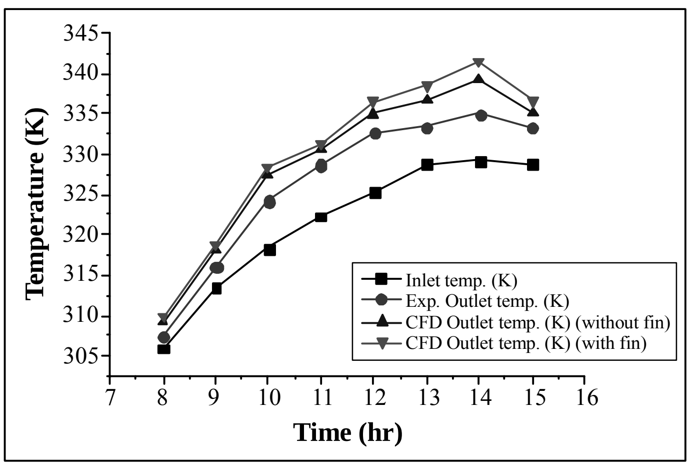

Fig. 8: Comparison of inlet & outlet temperature: Experimental vs CFD model

The CFD outlet is found to have same curve behaviour as that of the experimental value when it is plotted between temperature and solar insolation. The outlet of the SFPC system increases as the solar insolation increases and found to have a peak value at 14:00 hrs.

CONCLUSION

The results obtained from the CFD for tube with fin is validated with the CFD and experimental values for plain tube and found that there is an increase in temperature of about 3-4oC for finned tube. Based on this fabrication is done and experimental analysis is carried out with new set of collectors. The results obtained from experimental analysis shows that there is an increase in efficiency of about 3-5% when we go for finned tube rather than plain tube. This proves that there is a good agreement between the CFD analysis and the experimental values.

REFERENCES

1. A. Labed, N. Moummi, A. Benchabane, K. Aoues and A. Moummi, Performance Investigation of Singleand Double-Pass Solar Air Heaters Through the Use of Various Fin Geometries, J. Sustainable Energy, **31(6)**, 423-434 (2012).
2. C. D. Ho and T. C. Chen, The Recycle Effect on the Collector Efficiency Improvement of Double-Pass Sheet-and-Tube Solar Water Heaters with External Recycle, J. Renew Energy, **31**, 953-970 (2006).
3. H. C. Hottel and B. B. Woertz, Performance of Flat-Plate Solar-Heat Collectors, J. Trans ASME, **64**, 91-104 (1942).
4. M. Huq, A. M. A. Huq and M. M. Rahman, Experimental Measurements of Heat Transfer in an Internally Finned Tube, J. Heat Mass Transfer, **25**, 619-630 (1998).
5. K. Mohammadi and M. Sabzpooshani, Comprehensive Performance Evaluation and Parametric Studies of Single Pass Solar Air Heater with Fins and Baffles Attached Over the Absorber Plate, Energy, **57**, 741-750 (2013).

Revised : 05.08.2016 *Accepted : 07.08.2016*