Indian Journal of Science and Technology, *Vol 10(15), DOI: 10.17485/ijst/2017/v10i15/113827, April 2017*

ISSN (Print) : 0974-6846
ISSN (Online) : 0974-5645

Solar ETC Type Water Heaters – An Analysis Based on CFD Packages

Arun K. Raj[1*], K. Dileep[2] and S. Jayaraj[3]

[1,3]Department of Mechanical Engineering, NIT Calicut – 673601, Kerala, India; arunkraj03@gmail.com, sjayaraj@nitc.ac.in
[2]Department of Mechanical Engineering, VAST, Mumbai – 400059, Maharashtra, India; dileepk@vidyaacademy.ac.in

Abstract

Objectives: A comparative numerical analysis is conducted, to emphasize the capability of an open source CFD package (OpenFOAM) over the commercial CFD package of ANSYS. **Methods/Analysis:** Buoyancy driven flows occurring within the vacuum tubes of gravity-assisted solar water heaters are analyzed. A scaled model of the ETC type solar water heater has been chosen for computation. The geometry and meshing of the model was done by using the 'Design Modeler' and 'ICEM-CFD' packages available in ANSYS. The same configuration was also modeled in OpenFOAM to account for 3-D, transient, incompressible, laminar fluid flow analyses analogous to the available models in FLUENT. **Findings:** Boussinesq approximation is considered to be valid. Governing equations were discretized based on Finite Volume Method (FVM). 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 300 to 1000 W/m^2. For a uniform heat flux, with decrease in velocity of the fluid entering the storage tank, 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. The performance of ETC type solar water heaters is said to be more efficient at lower flow rates and the same is true at higher incident solar radiations. **Applications/Improvements:** Comparison of the results between OpenFOAM and FLUENT shows the wide reliability and urges the use of open source software like OpenFOAM for more complicated CFD modeling and computational studies.

Keywords: Finite Volume, Flow Rate, Heat Flux, OpenFOAM, Solar Water Heater

1. Introduction

Over the past century, there has been a dramatic increase in need for a much cleaner and an efficient source of energy, to account for the rising global energy demands. Solar energy proved to be a better substitute to cope with the non-renewable energy sources, due to its long term availability, longevity of solar equipment's in terms of harnessing the energy and also due to the ease of maintenance. An inclined surface subjected to a constant heat flux, has been experimentally studied by Vliet[1]. Based on the nature of flow and non-dimensional parameters such as Grashoff and Prandtl number, a set of empirical correlations was put forth to find the heat transfer coefficient. The analysis was further extended to study the turbulent characteristics, by using water flowing over a vertical plate subjected to a constant heat flux[2]. An experiment to analyze the thermosyphonic flow within flat plate solar collectors made from galvanized steel tubes was demonstrated by[3]. It was inferred that, the water flow

---

**Author for correspondence*

rate is dependent on the solar insolation falling over the collector surface. A 'Once-through' flow type thermosyphonic system - a variant of the thermosyphon system was experimentally studied by[4]. Based on the simulation results obtained, a semi-empirical correlation comprising of several key design parameters has been developed by[5]. On this basis, certain design rules were interpreted to improve the performance of a solar thermosyphon water heater. In[6] put forth a simple measuring device, to account for the flow rates within a thermosyphon solar water heater. The device tested to render moderate success, to account for the general characteristics, which were marked essential for a measuring device.

In[7] based on a transient simulation conducted using TRNSYS, tried to optimize the design parameters related with the thermosyphon solar water heater. It was concluded that, optimum design conditions could improve the system efficiency as well as transform the system to be more compact and much economical. An improved multivariable optimization technique was incorporated in TRNSYS, which was then used for analysis by[8]. In their analysis, the effect of weather conditions, previously analyzed data and design configuration of the collector were considered. A simple model validation by using TRNSYS approach was also carried out by[9]. Using an analytical approach, the expected energy gain from a solar domestic hot water system based on the design and constructional aspects were discussed by[10]. In turn, this approach could be used for energy optimization in the design phase as well as to test the existing systems to improve its performance. In[11] showed that the performance of a natural circulation solar water heater depends upon the flow rate as well as the incident solar radiation. To measure the flow rate within a water-in-glass type vacuum tube collector, the effect of tank temperature, aspect ratio, inclination of the collector surface as well as the input solar radiation was taken into consideration[12]. In[13] carried out a CFD based analysis on a modified evacuated tube type solar collector, using ANSYS-CFX. In[14] based on an analytical approach and by using TRNSYS, investigated the performance of a solar evacuated tube collector. By using regression analysis, linear curves were plotted relating the efficiency of the system with corresponding heat losses.

This paper has two key aims. Foremost, it highlights the physics behind the natural convection occurring within the tube enclosures, under the influence of buoyant forces. Moreover, it assesses the significance of flow rate and the input solar insolation falling over the collector surface, which in turn helps to raise the water temperature within the storage tank[15]. Secondly, it explores the way by which open source software like OpenFOAM could be used as a effective CFD tool in handling more complicated flow problems in conjunction with ANSYS-FLUENT.

2. Methodology

2.1 Geometry of Model

A three dimensional geometry of a 200 liter vacuum tube type solar collector, has been modeled by using the 'Design Modeler' package available in ANSYS. To save the computational time and cost, a scaled model has been adopted for carrying out the present analysis. The domain of the collector consists of an inlet, a cylindrical storage tank, an outlet and a certain number of inclined evacuated tubes. The outer diameter of the storage tank was 400 mm, while the inner diameter was 300 mm. The annular space was filled with an insulation material (polyurethane foam) to avoid heat leakages and to ensure its structural integrity. The inner section was made up of stainless steel while the outer section of the tank was made up of plastic. The storage tank had a length of 1000 mm. The inlet was provided at the bottom end of the tank with a diameter of 30 mm, while a 28 mm diameter outlet was provided at the top end of the tank. The scaled model comprised of three inclined tubes, which were made from borosilicate glass. The tubes had an inner and outer diameter of 42 mm and 50 mm, with an inclination of 30^0 to the horizontal. Rubber pads were provided at the tube ends, which acted as insulation and protected them from external breakages. The domain of interest is as illustrated in the Figure 1.

The main focus of the analysis was concentrated along the tubes and the outlet end of the storage tank. By using the 'ICEM-CFD' package provided by the ANSYS, the scaled geometry was equipped with a finer mesh for the inclined tube regions and the outlet end of the tank. Initially, surface meshing was carried out, based on which Delaunay scheme was used to render the volume meshing. An optimum number of mesh elements was chosen, based on a grid independence study. The scaled modeled comprised of 1,40,878 tetrahedral elements with an orthogonal quality of 0.45 and a skewness ratio of 0.44. Figure 2 depicts, the surface meshing as well as the cross sectional view of the volume mesh.

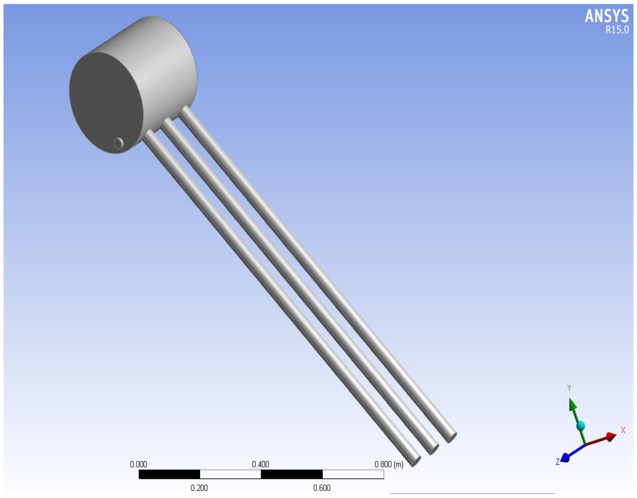

Figure 1. Domain of the scaled model used for computation.

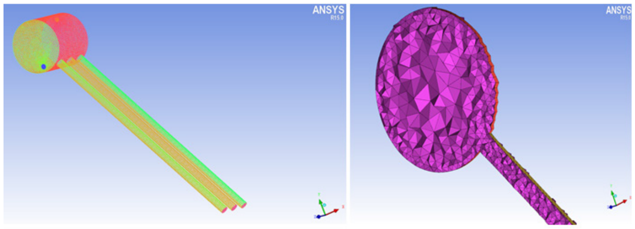

Figure 2. Surface and volume mesh of the scaled model.

2.2 Conservation Equations

In this present study, as a comparative analysis has been proposed, the conservation equations used to model the solver were similar in both of the CFD packages. The governing equations were modeled based on Boussinesq approximation and the effect of buoyant forces within a non-pressurized vacuum tube type solar collector is examined. A transient, three dimensional, incompressible laminar solver have been adopted for the analysis. In general, the conservation equations of mass, momentum and energy, incorporated for carrying out the present numerical analysis is as listed below:

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

$$\frac{\partial v}{\partial t}+v.(\nabla v)=-\frac{1}{\rho}\nabla p+\vartheta\nabla^{2}v-g\beta(T_{w}-T_{ref}) \qquad (2)$$

$$\frac{\partial T}{\partial t}+v.(\nabla T)=\alpha(\nabla^{2}T) \qquad (3)$$

Where α denotes the thermal diffusivity and β stands for the volumetric coefficient of thermal expansion. Based on the Boussinesq relation, β can be expressed as:

$$\frac{\rho_{0}-\rho}{\rho_{0}}=\beta(T_{w}-T_{ref}) \qquad (4)$$

Here ρ_0 denotes the reference density, T_w stands for the average temperature along the tube based on the average flux falling on the tube surface and T_{ref} signifies the reference temperature of the fluid entering the storage tank.

2.3 Boundary Conditions

Natural convection occurring within the tube enclosures facilitates in carrying the heat from the tube surfaces to the storage tank, thereby, raising the water temperature. The geometry as well as the mesh of the scaled model, built using various packages in ANSYS was imported into the OpenFOAM environment. Due to the adequate flexibility offered by the conversion utilities in-built in OpenFOAM, the mesh was successfully imported without any topological errors. At most care was taken to ensure, each of these boundaries were attributed with specific names so that, after importing into the OpenFoam framework these boundaries were attributed with similar patch names. Depending upon the nature of boundaries, specific conditions were imposed among all these patches.

The inlet was attributed as the velocity-inlet with a fixed value, corresponding to the desired Reynolds number. The cylinder tank and the tube ends were treated as adiabatic and no-slip condition was imposed. The inclined tubes were divided into two equal halves along the axial direction. It facilitated in assigning different values of heat fluxes, for the top and bottom sections of the inclined tubes. In OpenFoam, corresponding to the top and bottom sections of the tubes, a fixed Gradient condition along with the no-slip condition was used for carrying out the analysis. By default, it was assumed that only 25% of the radiations falling on the top side of the tube were set as the input to the bottom side. Thus, the top section of the tube facing the solar insolation received maximum radiations compared to the bottom section, which received radiations which are diffused and reflected from the ground. In this analysis, based on the variation of mass flow rate and solar insolation, the resulting outlet temperature of water from the storage tank is examined by using ANSYS-FLUENT and OpenFOAM. In general, a mass flow rate of 0.02 kg/s and solar insolation (Q) of 800 W/m^2 were chosen as the default boundary conditions for carrying out the present analysis. The reference temperature of the fluid was assigned to be 298 K. The thermophysical properties were chosen from standard data tables, based on the reference temperature.

2.4 Numerical Schemes

Conservation equations were discretized by using the Finite Volume Method (FVM), in both of the CFD packages. A transient analysis based on Pressure Implicit with Splitting of Operator (PISO) algorithm, has been carried out. The transient terms were discretized by using the first order, implicit schemes with fixed time stepping approach in FLUENT whereas first order, implicit Eulerian schemes has been used in OpenFOAM for discretization of transient terms. The run-time discretization schemes in OpenFOAM were used to be discretized using Gaussian integration and flux interpolation schemes. In both cases, the divergence terms are solved by using bounded, second order upwind differencing schemes. Gauss linear uncorrected schemes are bounded, non-conservative, first order differencing schemes used to solve the laplacian terms in OpenFOAM. The variation of pressure is numerically computed using the Preconditioned (bi)-Conjugate Gradient (PCG) schemes and the Diagonal Incomplete-Cholesky (DIC) scheme is used as a smoother. Preconditioned bi-Conjugate Gradient (PBiCG) were used to discretize velocity and temperature. Relaxation factors are set for velocity and pressure as 0.7 and 0.3, respectively.

3. Results and Discussion

In the present analysis, parametric studies have been carried out to inspect the role played by the gravitational forces, in assisting the natural convection occurring within the vacuum tube enclosures. Flow rates have been varied from 0.0002 to 0.03 kg/s by keeping the flux constant over the tube surface. Similarly, at a constant flow rate, the effect of variation in the collector performance at different solar intensities is also examined. The analysis was carried out using the same conservation equations, model geometry, mesh elements as well as similar boundary conditions in ANSYS-FLUENT as well as OpenFOAM. However, to ensure the accuracy of the results, it was necessary to establish that the solutions obtained during the numerical computations were independent of the mesh size. Therefore, a grid independence test was carried out with the mesh elements varying within the range of 97,936 to 1,98,299. It was inferred that the results obtained with the mesh elements within the range of 1,40,878 to 1,98,299 were similar. Hence, the scaled model with a mesh of 1,40,878 elements were chosen as the optimum mesh for carrying out further numerical computations as shown in Figure 3.

In order to study the effect of flow rate on the performance of an evacuated tube type solar collector, a range of flow rates were chosen for analysis. By varying the velocity and keeping all other geometrical dimensions and thermophysical properties as constants, the analysis was conducted. Figure 4 shows the response of the variation in mass flow rate on the outlet temperature of water obtained for duration of 600s. Axial distance (z-axis) along the storage tank outlet has been chosen as the reference, for plotting the graphs. It can be inferred that for a lower mass flow rate, the temperature of water obtained at the outlet was higher for the same intensity of radiation falling over the tube surfaces. As shown in Figure 4 for a mass flow rate of 0.0002 kg/s, the outlet temperature obtained was 300.24 K and for a flow rate of 0.03 kg/s, the water outlet temperature was found to be 299.88 K. This can be accounted due to the fact that a lower flow rate facilitates greater interaction between the fluid and the tube surfaces. This results in a higher heat gain from the tube surface to the fluid, which eventually rises up and gets collected at the top of the storage tank, due to buoyancy effects.

Similar analysis was performed by using FLUENT, yet another FVM based solver package like OpenFOAM. Figure 5 shows the comparison of the results obtained at 600 s, by using both CFD packages. It can be concluded that the trend of the outlet water temperature obtained by using either of the finite volume based solvers were identical. There is a slight variation in the values of temperature, which might have aroused due to the interpolation errors as well as due to the first order accuracy of the discretization schemes, used to solve the laplacian terms, in OpenFOAM.

The above results can be visualized with the help of contours and vector plots, which helps to give an insight onto the nature of flow occurring within the vacuum tubes of the collector. A cross sectional view of the center tube of the scaled model has been chosen for representation. Figure 6 illustrates the temperature contours plotted at different flow rates, for a time period of 600s. As shown in Figure 6 higher temperature gain is obtained corresponding to lower flow rates than at higher mass flow rate. Moreover, these contours indicate the recirculation of the fluid, from the bottom end of the tube towards the top end of the storage tank. In addition, the rise in temperature of water within the storage tank at different time instances corresponding to a mass flow rate of 0.03 kg/s is also depicted in Figure 7.

The effect of variation of the incident solar radiation over the tube surfaces has been investigated, over a wide range of heat flux (i.e. 300 to 1000 W/m^2). 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. Figure 8 depicts the variation in the outlet water temperature, at different values of solar insolation falling over the tube surfaces. It can be noted that, for a heat flux of 400 W/m^2 the water temperature rises up to 299.02 K, when subjected to a time period of 600 s. On the other hand, water gets heated to 300.81 K, when subjected to a solar insolation of 1000 W/m^2. Similarly, the variations in the temperature of water within the storage tank, at different time instances are also shown in Figure 9.

Figure 10 indicates comparison of the results obtained, by using FLUENT as well as OpenFOAM. Two cases of the solar insolation (i.e. 400 W/m^2 and 1000 W/m^2) at a fixed mass flow rate and for a time duration of 600 s have been considered for comparison. Similar to the analysis of variation in mass flow rate (i.e. Figure 5. both solvers yielded similar trend of results and it was found to be satisfactory.

Figure 11 illustrates the temperature contours along the cross section of the center tube, at different solar intensities and at a time instance of 600 s. 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. Moreover, the temperature contours at various time instances are also illustrated in Figure 12. Based on these contour plots, it can be well established that, as time proceeds the fluid flowing inside these vacuum tubes, recirculates under the effect of buoyant forces. Due to this phenomenon, hot fluid rises towards the top of the storage tank in-taking heat, while the cold fluid flows to the bottom of the tubes.

In addition to contour plots, vector plots helps to visualize the recirculation occurring within the vacuum tube enclosures of the non-pressurized evacuated tube type solar collector. Figure 13 depicts the cross sectional view of velocity vector plot along the center tube at a mass flow rate of 0.0002 kg/s and a solar insolation of 800 W/m^2. It can be clearly visualized that closer to the top side of the tube, water gains heat from tube surfaces and recirculate back towards the top end of the storage tank. Similarly, the cold water flows through the bottom side of the vacuum tubes towards the bottom end. Figure 14 illustrates a streamline plot of the storage tank, which indicates that the hot water gets collected at the top side of the tank and could be drawn off based on the requirement.

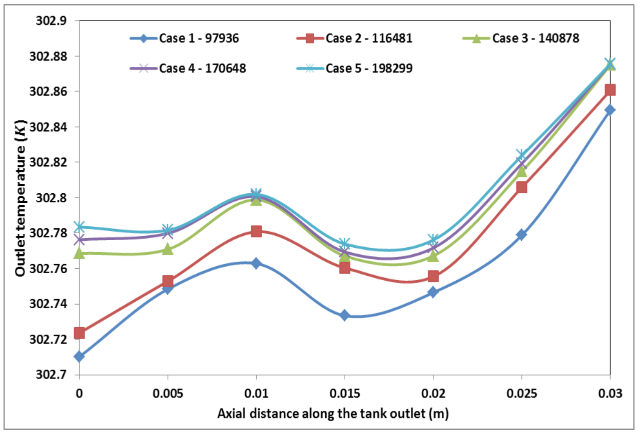

Figure 3. Grid independency test for the present analysis.

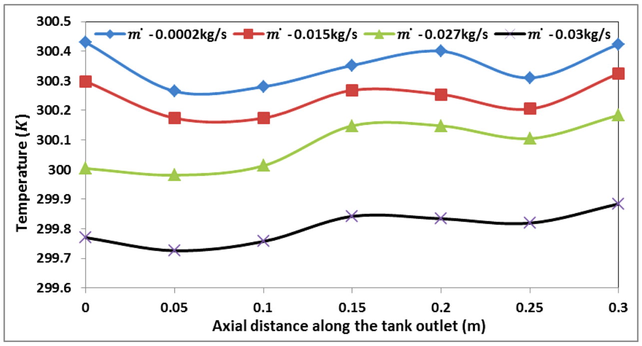

Figure 4. Flow rate variation at duration of 600 s.

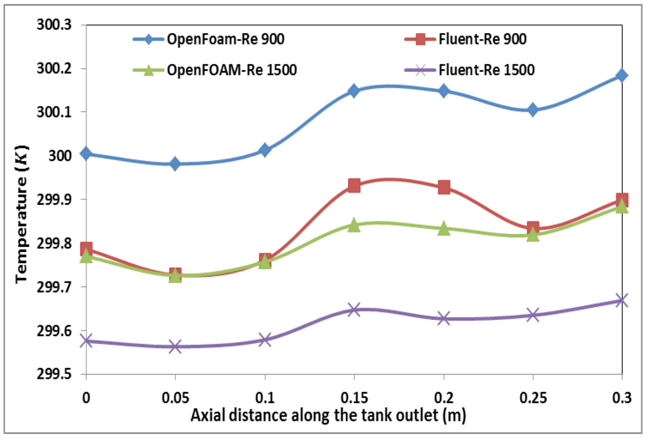

Figure 5. Comparison of results obtained at duration of 600 s.

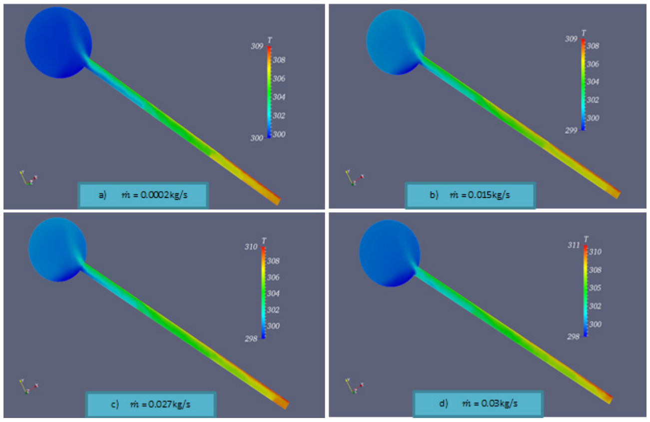

Figure 6. Temperature contours at different flow rates and at t – 600 s.

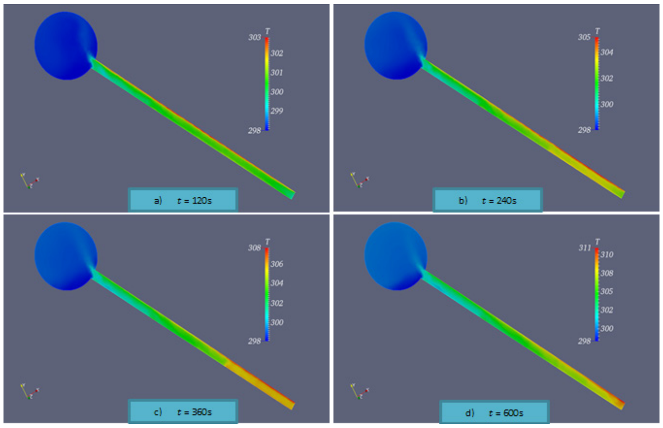

Figure 7. Temperature contours at different time instances and at 0.03 kg/s.

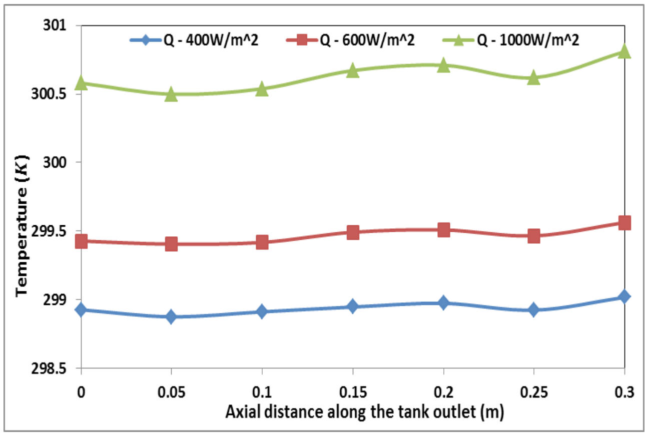

Figure 8. Heat flux variation at duration of 600 s.

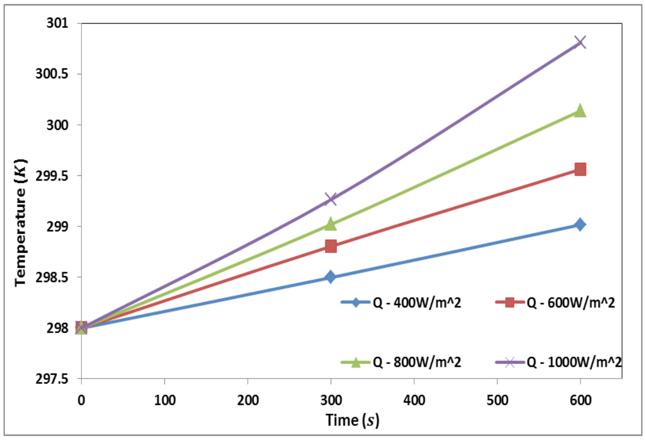

Figure 9. Effect of heat flux variation at different time intervals.

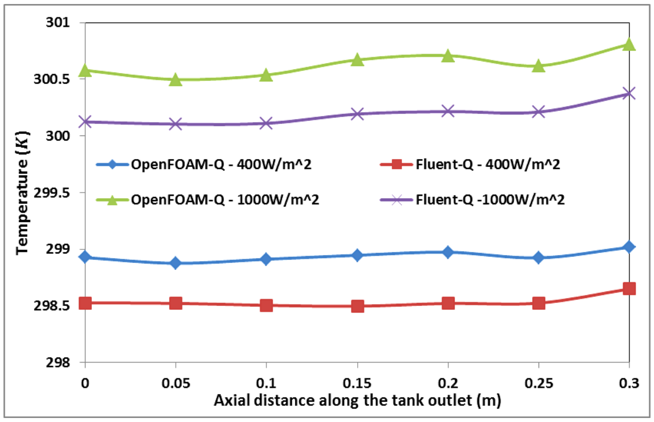

Figure 10. Comparison of results obtained at duration of 600 s

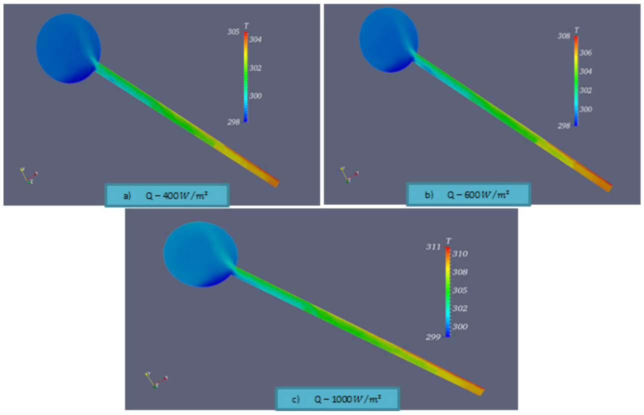

Figure 11. Temperature contours at different Q and at t – 600 s.

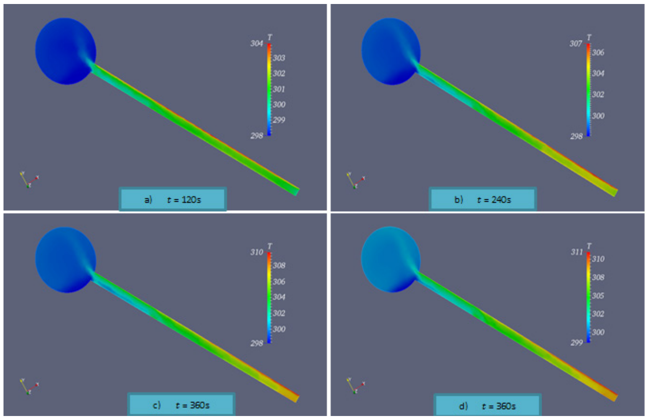

Figure 12. Contours at different time instances and at Q – 1000 W/m2.

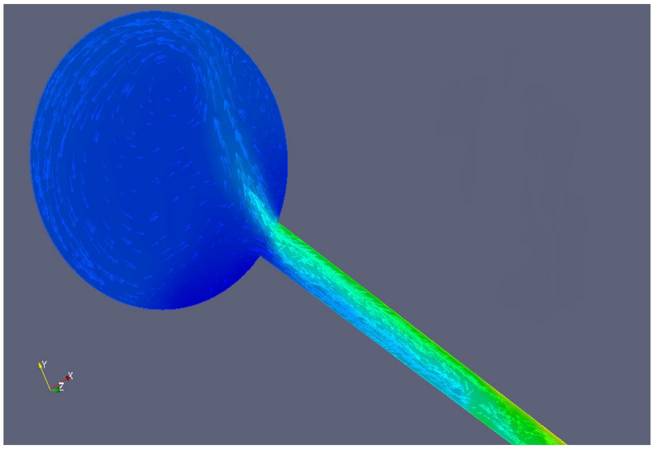

Figure 13. Velocity vector plots at Q – 800 W/m2 and 0.0002 kg/s.

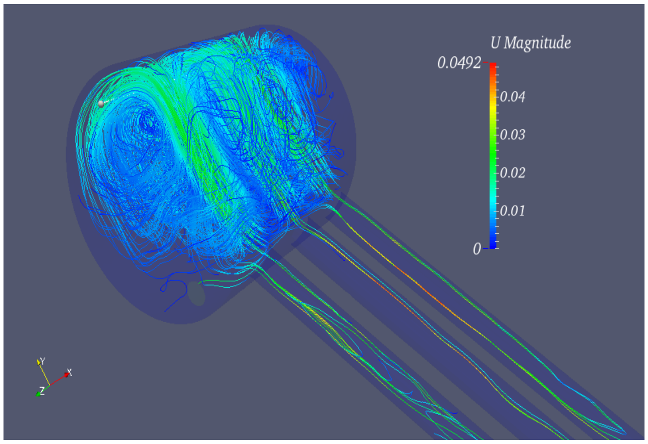

Figure 14. Streamlines inside the storage tank at Q - 1000 W/m2.

4. Conclusion

This study set out to better understand how the performance of a vacuum tube type solar collector is affected by flow rate as well as the incident solar insolation over the evacuated tube surfaces. It was evident from the analysis that natural convection phenomenon occurring within the tube enclosures driven by the buoyant forces helps to raise the temperature of water within the storage tank. 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 all other possible combinations. Contours and vector plots showed a visualized representation of the recirculation, which in turn helped to strengthen the understanding of inclined natural convection within tube enclosures. In turn, this paper has also argued that open source software like OpenFOAM could be used as a potential CFD tool for carrying out complex flow problems. The results obtained by using two CFD based, finite volume packages such as ANSYS-FLUENT and OpenFOAM has been compared. The results obtained were found to be satisfactory. Hence, the findings of this study suggest and urge the use of OpenFOAM as a potential tool for complicated CFD modeling and computational activities.

5. 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
7. Shariah A, Shalabi B. Optimal design for a thermosyphon solar water heater. Renewable Energy. 1997 Jul; 11(3):351–61. Crossref
8. Abdunnabi MJR, Loveday DL. Optimization of thermosyphon solar water heaters using TRNSYS. Part 2: Parametric study using a modified TRNSYS model. IPCBEE. 2012 Jun; 28:154–60.
9. Kalogirou SA, Papamarcou C. Modeling of a thermosyphon solar water heating system and simple model validation. Renewable Energy. 2000 Feb; 21(3-4):471–93. Crossref
10. Belessiotis V, Mathioulakis E. Analytical approach of thermosyphon solar domestic hot water system performance. Solar Energy. 2002 Apr; 72(4):307–15. Crossref
11. Bolaji BO. Flow design and collector performance of natural circulation solar water heater. Journal of Engineering and Applied Sciences. 2006 Jun; 1(1):7–13.
12. Budihardjo I, Morrison G L, Behnia M. Natural circulation flow through water-in-glass evacuated tube solar collectors. Solar Energy. 2007 Dec; 81(12):1460–72. Crossref
13. Sato AI, Scalon VL, Padilha A. Numerical analysis of a modified evacuated tubes solar collector. RE&PQJ. 2012 Apr; 1(10):384–9.
14. Mohammadkarim A, Kasaeian A, Kaabinejadian A. Performance investigation of solar evacuated tube collector using TRNSYS in Tehran. International Journal of Renewable Energy Research. 2014 May; 4(2):497–503.
15. Arun KR, Dileep K, Jayaraj S. Numerical investigation on buoyancy driven flows in a non-pressurized ETC type solar water heater used for domestic applications. 6th International National Conference on FMFP MNIT; Allahabad. 2016 Dec.