

Optimization Of The Flow Distribution In V-Groove Flat Plate Solar Collector using Cfd Simulation



Ali Omer, Hiram Ndiritu, Stephen Wanji

Abstract: Flat plate solar collector is the major component of a solar heating system that converts solar radiation to thermal energy. It provides clean energy at no operating cost, however, its poor performance constitutes a serious drawback to adopt it for small application. This inefficiency is the result of involved thermal losses and the lack of full exploitation of the available energy. To exploit the maximum potential, the working fluid flow should be uniformly distributed through the collector to extract the heat from the hot absorbing surface. This study addresses the uniformity of the flow distribution for v-groove flat plate solar collector for water heating to optimize the performance of the collector. The study investigated the effect of the manifold geometry and the number of the side riser channels on the flow distribution by using numerical computational fluid dynamics simulation on Ansys Fluent Software. The mass flow rate was optimized for maximum thermal performance and then the optimum point was used for investigating the flow distribution. The simulation was validated against experimental data from literature with 99% confidence. The study found that the circular manifold gives uniform flow distribution with a standard deviation of 5% at the optimum mass flow rate of 11.5g/s. the study concluded that the tapered circular manifold is the optimum geometry for uniform flow distribution as it provides the least pressure difference inside the manifold.

Keywords: flat plate collector; flow distribution; optimization; computational fluid dynamics;

I. INTRODUCTION

Uniform flow distribution plays a major role in determining the performance of a system that involves fluid dynamics [1]. Fluid manifolds can be categorized into two group bifurcation manifold which contains only two discharge per stage, and consecutive manifold which involves multiple discharge ports. A typical consecutive manifold tends to distribute the fluid in a random manner depending on the pressure drop inside the manifold [2]. The studies on the analysis of the collector flow distribution agreed in attributing the flow non-uniformity in the collector to the pressure difference that is occurring inside the manifold tube of the collector. The internal flow experiences viscous and friction forces as it flows between two points. The friction effects tend to produce a pressure drop in the flow direction while the momentum effect produces pressure to rise.

The proper balance between the two results in an optimum design that minimize the non-uniformity. The momentum of the flowing fluid into distributing manifold has a tendency to carry the fluid toward the manifold dead-end where the pressure increases [3].

In the Flat Plate Collector (FBC) the heat is transferred from the absorber plate to the working fluid as it circulates through the collector. This fluid circulation is distributed from the entrance manifold through the multiple riser tubes to the outlet manifold. To obtain maximum performance, the flow should be equally distributed among the riser tubes that is connecting the inlet and exit manifolds, however, a typical manifold will not provide even flow distribution due to the fluid friction and the sudden branching in the manifold as well as the change in the fluid momentum [2].

The literature is extensively proved that the performance of the FBC is affected by the flow distribution [4], [5] [6]. Quite many parameters are involved in the distribution of the fluid in the collector such as fluid mass flow rate, collector tilt angle, and flow direction. Juan et al [6] found that the high flow rate increase flow misdistribution due to increased inertia effect. Jianhua et al [7] found that the collector tilt angle negatively influences the flow distribution. Jafar et al [2] investigated the influence of the configuration on the flow distribution, they found that the collector configuration that has the same exit direction as the inlet is superior to its counterpart.

Juan et al [6] studied the flow misdistribution in flat plate solar collector. They used numerical CFD simulation to perform a comparison study for solar collectors with insert plate within the collector manifold to act as if the flow is coming from different flow jets. Their results showed that collectors with insert plates manifold have superior flow distributions than its plain counterpart however it involves more complexity and hence more cost.

Jimmy et al [8] identified strategies to study the flow distribution in manifold systems. These strategies are modifying the manifold designs like enlargement of the cross-sectional area and modifying the outflow channels.

The current study addresses the uniformity of the flow distribution for v-groove FBC to optimize the thermal performance of the solar collector. The study is based on modifying the inlet manifold geometry to provide a uniform flow distribution.

Manuscript published on 30 September 2019

* Correspondence Author

Ali Omer*, (PAUSTI). Nairobi, Kenya. Email: abenomer12@gmail.com

Dr. Hiram Ndiritu, Principal, CollTec, JKUAT, Nairobi, Kenya.

Dr. Stephen Wanji, department of mechanical engineering, JKUAT, Nairobi, Kenya.

© The Authors. Published by Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP). This is an open access article under the CC-BY-NC-ND license <http://creativecommons.org/licenses/by-nc-nd/4.0/>

II. METHODOLOGY

This study was conducted by using numerical CFD ANSYS FLUENT techniques. Ansys Fluent is one of the powerful general-purpose CFD codes. It solves the governing equations of flow, energy, and other scalars by using the control volume techniques. Any CFD simulation contains three main stages namely pre-processing, solver, and post-processing stages [9].

The pre-processing stage is concerned with defining the problem at hand in a simple way. This stage includes building the computational domain, discretize it, specifying reasonable assumptions, operating conditions, and preparing the problem for the solver. In the solver, the governing equations are solved by appropriate techniques. The results of the simulation are obtained in the post-processing. This steps of CFD simulation are summarized in Fig. 1. The solution procedure for solving flow and energy problem in FLUENT is shown in Fig. 2. To simplify the complexity of the simulation some necessary assumptions were made without obscuring the accuracy as follows:

1. The system is at the steady-state condition.
2. The flow is internal, incompressible and single-phase flow.
3. Thermos-physical properties of the material are independent of temperature.

A. The scope of the study

This study addresses the problem of flow distribution in the flat plate solar collector. Firstly the simulation procedures were validated against available experimental data from the literature. Secondly, the mass flow rate was optimized for maximum thermal performance and then the optimum point is used to study the flow distribution. The flow distribution problem was examined by investigating the effect of the manifold geometry and changing the number of the side channels.

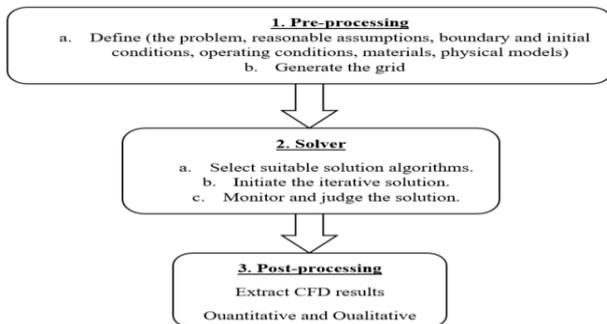


Fig. 1: The main elements of CFD simulation.

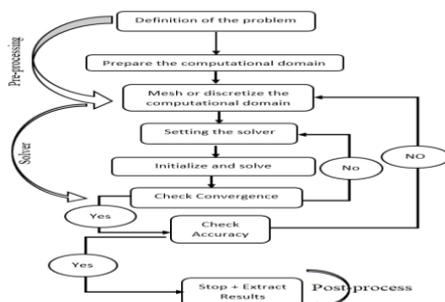


Fig. 2: The solution procedure in ANSYS FLUENT.

B. Computational domain of the study

The computational domain is a physical geometry in which the physical phenomena occur or the system equations are solved. This study was divided into two part, the first part included the validation and optimizing the mass flow rate, at this stage the computational domain contains the v-groove absorber plate with the triangular water channels and the insulation layer, only the housing box was excluded but still its effect was included in the simulation by the boundary condition. Fig. 3 shows the computational domain for this stage. The second part of the study included investigating the flow distribution uniformity, the computational domain was reduced by excluding the outlet header so as to facilitate the study. Fig. 4 shows the second computational domain.

C. The mathematical modeling

In this part, the interest is to describe the governing equations of the heat transfer and fluid dynamics phenomena that are solved in the numerical simulation of the FBC as well as its performance mathematical models. According to Versteeg et al [10]. The governing equations of the fluid dynamic and energy solved numerically are energy, continuity, and momentum equations. These equations expressed as follow.

$$\frac{\partial \rho T}{\partial t} + \frac{\partial \rho T u_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left[k \frac{\partial T}{\partial x_j} \right] \quad (1)$$

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \quad (2)$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = - \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] \quad (3)$$

Where: $\rho, \mu, P, T, t, c_p, k, u_i$ represent density, dynamic Viscosity, pressure, temperature, time, heat capacity, Thermal conductivity, and velocity in x, y, z respectively.

The general performance of the FBC is described by useful energy gain Q_u and efficiencies [11].

$$Q_u = m c_p (T_o - T_i) \quad (4)$$

Where m is mass flow rate, the thermal efficiency is given as:

$$\xi_{\text{thermal}} = \frac{Q_u}{A_a \cdot I} \quad (6)$$

Where: I is solar radiation, A_a is the area.

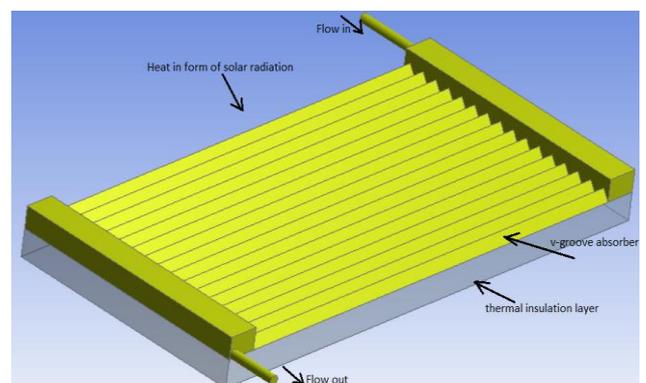


Fig. 3: Computational domain of the study at the first stage.

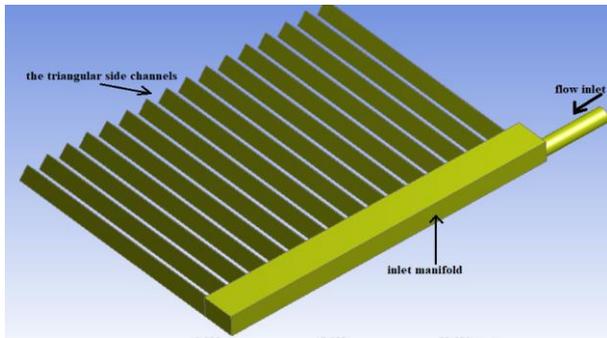


Fig. 4: Computational domain of the study at the second stage.

D. Validation of the simulation

To have meaningful simulation outcomes, it is necessary to obtain a converged, accurate, and valid simulation. In the current study, the solution convergence is judged based on mass and energy conservation, residual, and monitoring the stability of results.

The accuracy is obtained from the grid independence test Table-I shows the grid independence test which is conducted under the same conditions by refining the grid size gradually. Fig. 5 shows that from the mesh size of 3mm more refinement has a negligible effect and therefore, this size was used for the study.

To assess the validity and reliability of this simulation procedures and outcomes, the results were compared with an experimental data generated by Ng'ethe et al [12]. The confidence level of the experimental measuring equipment is 95% [12]. The simulation model parameters and boundary conditions are all informed by the experimental setup. Table- II summarizes the validation process. Fig. 6 shows the comparison between the simulation and experimental results, from which it was concluded that this simulation method is valid and can represent the real system with an acceptable average error of 0.66%. Therefore, it is possible to use these simulation models and procedures for the purpose of investigation study.

Table- I: Grid independence test for the simulation

Grid size [mm]	Element 10 ³	Nodes 10 ⁶	Mass [g/s]	Tout[K]
5.5	496	1.1	6.445	316.9
5	627	1.3	7.112	314.1
4	713	1.5	7.127	314.2
3	882	1.8	7.128	314.3
2	1233	2.7	7.128	314.3

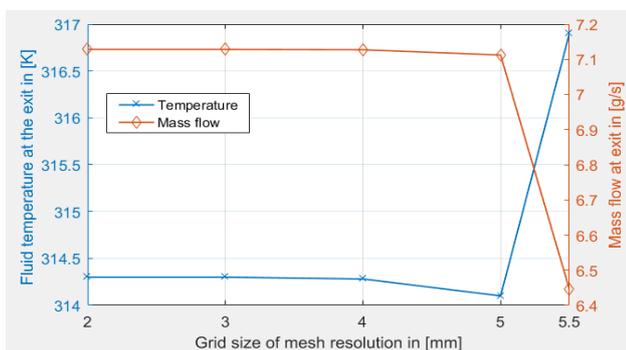


Fig. 5: Grid independence test.

Table- II: Comparison of the experiment with the simulation

Insolation [w/m ²]	T _i [K]	Exp. data	Sim. data	Error $\left(\frac{\text{Sim} - \text{Exp}}{\text{Exp}}\right) * 100$
		T _{out} [K]	T _{out} [K]	
817	291	316	318	0.63%
837	296	319.5	322	0.78%
286	297	306	307	0.33%
534	297	313	315	0.64%
357	297	308.1	309.6	0.48%
499	295	306.9	310	1.01%
259	297	302.6	305	0.79%

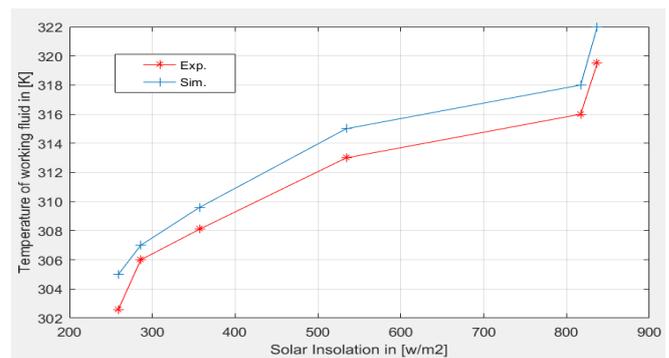


Fig. 6: Comparison of the experimental and simulation results.

III. RESULTS AND DISCUSSION

The flow distribution in the solar collector is affected by the design of the collector as well as the operating conditions. The present study addresses the flow distribution in the v-groove flat plate solar collector for water heating. The study started with optimizing the mass flow rate for maximum collector thermal performance, and then the optimum flow rate was used to study the effect of the manifold geometry and the effect of changing the number of the riser tube channels on the flow distribution.

A. Optimization of mass flow rate for maximum thermal performance

The mass flow rate of the solar collector that gives maximum thermal performance was obtained using the single-variable optimization method. The mass flow rate was varied from (zero to 24 g/s) at which the flow is laminar. The search for the optimum mass flow rate was restricted within the laminar flow zone since the scope of the study was confined with the laminar flow zone.

The optimum flow rate was found equal to (11.5 g/s) which gives the maximum collector thermal efficiency equal to (78.9%). Fig. 7 show the detailed steps of conducting the optimization by using the single-variable search optimization approach.

Optimization Of The Flow Distribution In V-Groove Flat Plate Solar Collector using Cfd Simulation

B. The effect of the manifold geometry on the flow distribution in flat plate collector

The effect of the manifold duct geometry on the flow distribution has been investigated, five geometries were considered namely triangular duct, circular duct, rectangular duct, a rectangular cross-section with one side tapered duct and rectangular cross-section with two sides tapered duct. These different geometries are shown in Fig. 8.

Simulations were run for five different geometries under identical condition. The pressure drop inside the manifold and the flow rates for individual branching channels were

computed. The different geometries were compared using the standard deviation of the flow which is the deviation of the observed flow rate at the branching channels from the optimum. Fig. 9 shows that the circular manifold geometry gives the best flow distribution to the branching channels with a minimum standard deviation of the flow equal to 6.8 %. This attributed to the small pressure drop in the circular manifold geometry as shown in Fig. 10.

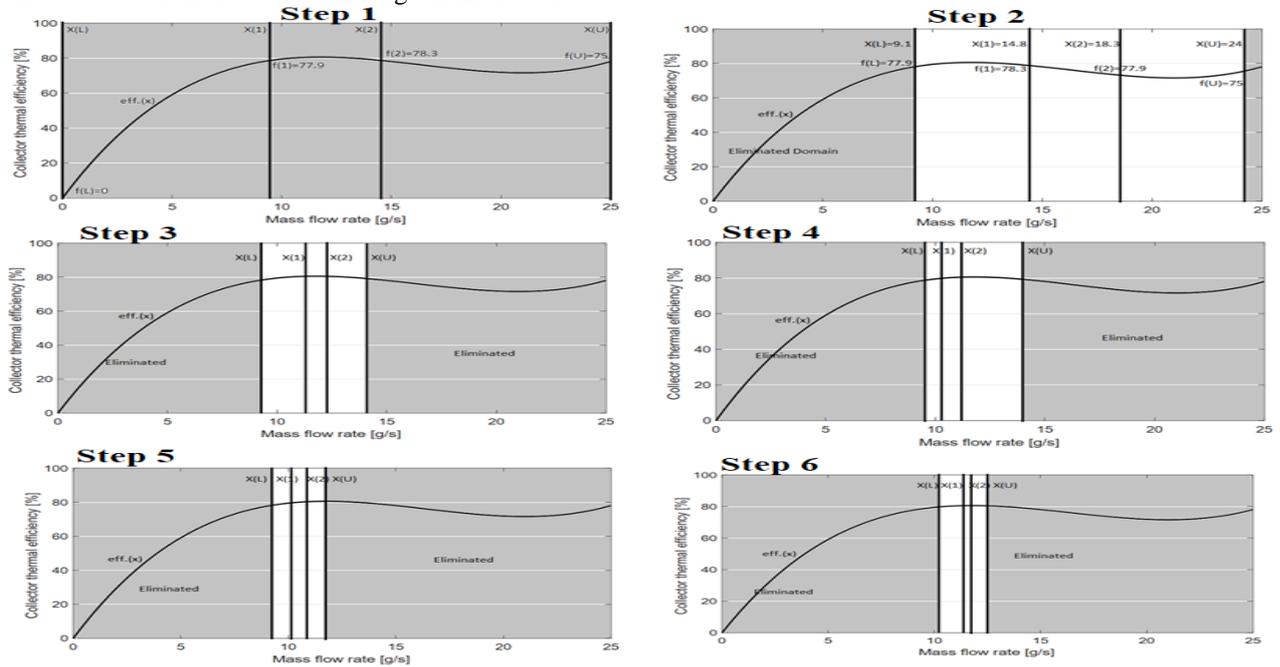


Fig. 7: Steps to conduct the single-variable golden section search optimization method.

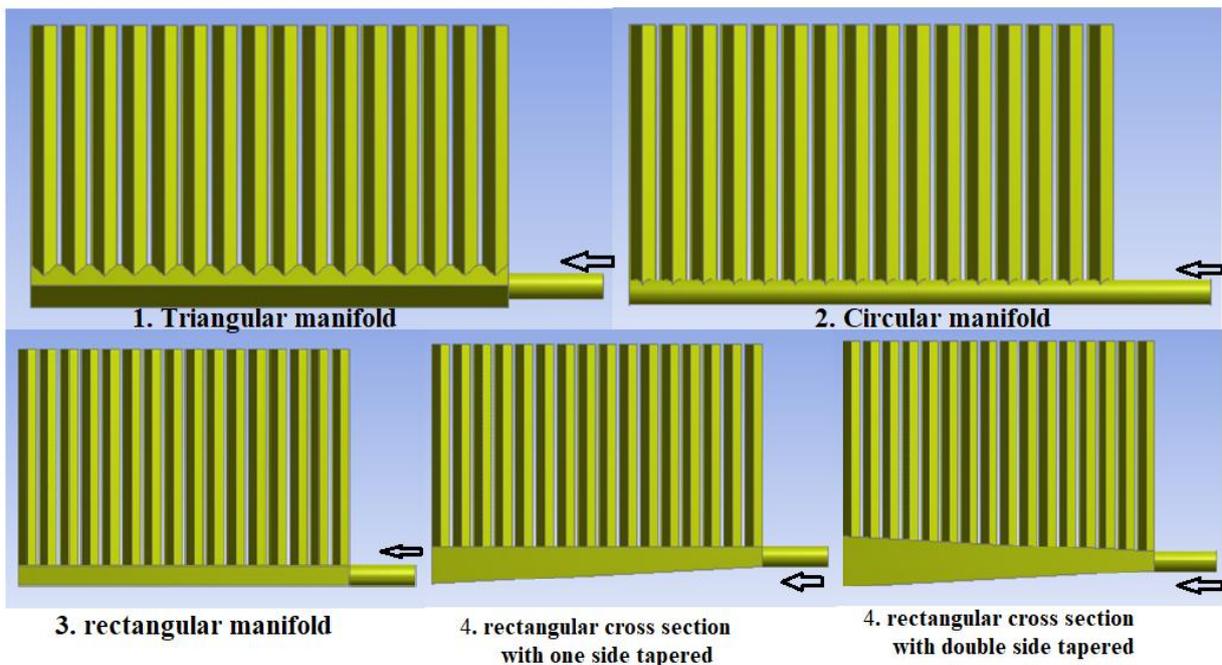


Fig. 8: The manifold geometries for investigating the flow distribution.

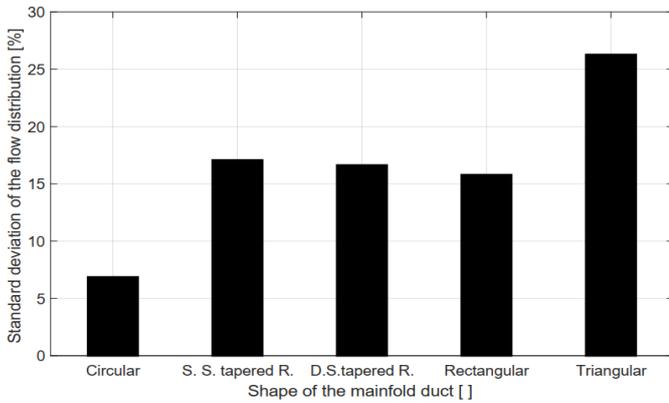


Fig. 9: manifold geometry vs. deviation of flow distribution.

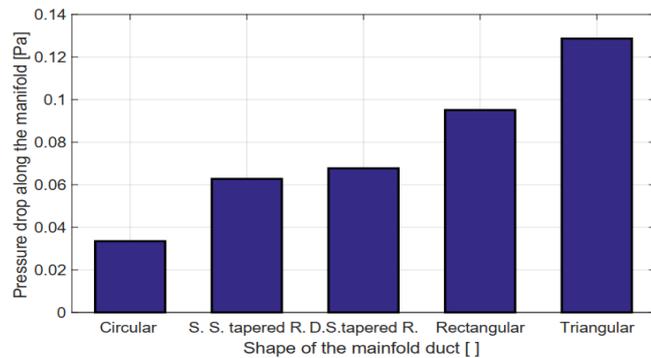


Fig. 10: manifold geometry vs. pressure drop.

C. The influence of tapering the circular manifold on the flow distribution in the flat solar collector

J. Wang & J.P.Chiou showed that the flow distribution in the manifold is governed by the pressure drop [3], [4]. In the internal flow, the pressure normally drops along the direction of the flow. However, in the manifold the pressure rise in the downstream. This phenomenon has been elaborated by wang et al [3].

The effect of tapering the circular manifold has been investigated to obtain the least pressure drop between upstream and downstream of the manifold and hence the best flow distribution.

The ratio of tapering was varied from 0.5 to 1.3. Fig. 11 depicts the variation of tapering ratio of the circular manifold against the pressure drop. It shows that at tapering ratio of 1.1 the pressure drop is the least at a value equal to 0.03 Pa. corresponding to this point the flow distribution was found also the best with standard deviation equal to 5% as shown in Fig. 12.

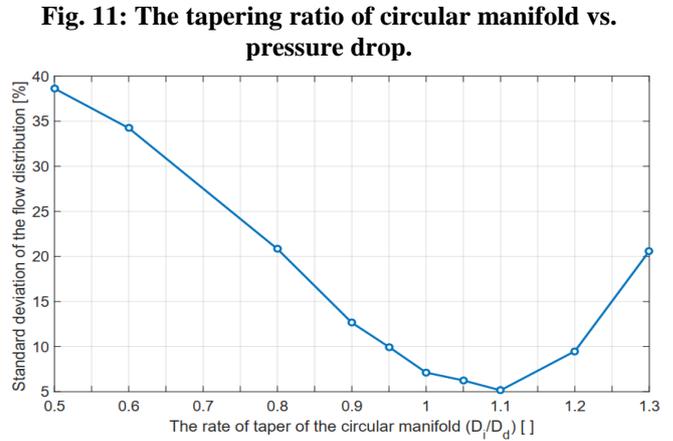
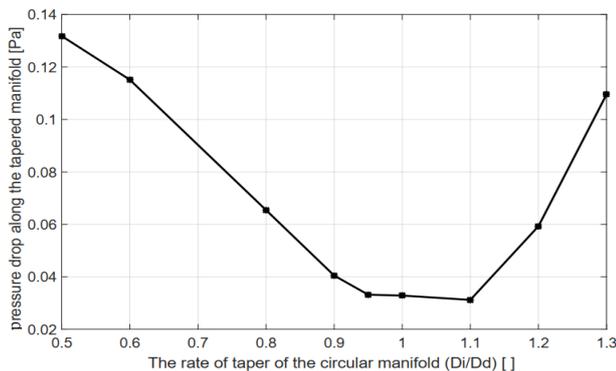


Fig. 12: The tapering ratio of circular manifold vs. deviation of flow distribution.

D. The effect of changing the number of side branching channels on the flow distribution

One of the strategies to study the flow distribution is to manipulate the sides branching channels as shown by Jimmy et al [8]. The influence of changing the number of side channels was investigated. The investigation covered changing the number of channels from 6 channels to 16 channels. Fig. 13 depicts the variation of the side channels number with the pressure drop, it shows that the value of the pressure drop is reduced as the side channels number is reduced due to absence of the pressure loss from the branching.

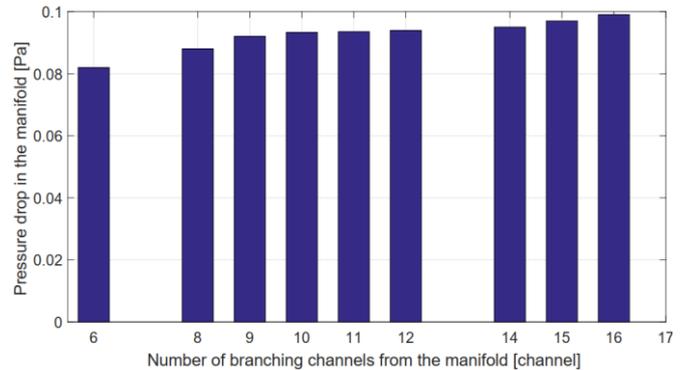


Fig. 13: The number of side channels vs. pressure drop.

IV. CONCLUSION

The flow distribution in the v-groove flat plate solar collector was studied by using numerical CFD simulation. The simulation was validated from literature and then used to study the flow distribution. The optimum mass flow rate was used to investigate the influence of the manifold geometry and the effect of the number of riser channels on the flow distribution. The following conclusions were derived:

1. The optimum mass flow rate was 11.5 g/s.
2. The circular manifold provides more uniform flow distribution than the other geometries that were considered in this study due to the pressure behavior inside the manifold.
3. The circular manifold with a tapering ratio of 1.1 was found to be the best in term of flow



distribution uniformity.

4. Increasing the number of the manifold riser tube causes a large pressure drop in the manifold, resulting in a higher deviation in flow distribution.

Acknowledgment

The authors acknowledge the Pan African University Institute for Basic Science, Technology and Innovation (PAUSTI).

REFERENCES

1. J. wang, "Theory of flow distribution in manifolds," *Chemical engineering journal*, vol. 16, no. 8, pp. 1331-1345, 2011.
2. Jafar M. Hassan, Thamer A. Mohamed, Wahid S. Mohammed, Wissan H. Alawee, "Modeling the uniformity of manifold with various configurations," *Journal of fluids*, vol. 2014, 2014.
3. J. Wang, "Pressure drop and flow distribution in parallel-channel configurations of fuel cells U-type arrangement," *International journal of hydrogen and energy*, vol. 33, pp. 6339-6350, 2008.
4. J. P. Chiou, "The effect of non-uniform fluid flow distribution on the thermal performance of the solar collector," *Solar energy*, vol. 29, no. 6, pp. 487-502, 1982.
5. X.A.WangL.G.Wu, "Analysis and performance of flat-plate solar collector arrays," *Solar energy*, vol. 45, no. 2, pp. 71-78, 1990.
6. Juan Manuel García-Guendulain, José Manuel Riesco-Avila, Francisco Elizalde-Blancas, Juan Manuel Belman-Flores, Juan Serrano-Arellano, "Numerical Study on the Effect of Distribution Plates in the Manifolds on the Flow Distribution and Thermal Performance of a Flat Plate Solar Collector," *energies*, 2018.
7. Jianhua Fan, Louise Jivan Shah, SimonFurbo, "Flow distribution in a solar collector panel with horizontally inclined absorber strips," *solar energy*, no. 12, pp. 1501-1511, 2007.
8. Jimmy C. K. Tang, Ephraim M. Sparrow, John P. Abraham, "Geometric strategies for attainment of identical outflows through all of the exit ports of a distribution manifold in a manifold system," *Applied Thermal Engineering*, vol. 29, pp. 3552-3560, 2009.
9. Mohammed Abdul Junaid, Mohammed Nazimuddin, Mohd Arifuddin, Mohammed Faisal, "Thermal Analysis of Solar Flat Plate Collector Using CFD," *International Journal of Engineering Research & Technology (IJERT)*, vol. 6, no. 4, pp. 659-662, 2017.
10. H.K Versteeg, W. Malalasekera, *An introduction to computational fluid dynamics the finite volume method*, Edinburgh: Longman Scientific and Technical, 2007.
11. G.N. Tiwari, Arvind T. Shyam, *Handbook of solar energy theory, analysis and application*, Media Singapore: Springer, 2016.
12. John Ng'ethe, Hiram Ndiritu, Stephen Wanjii, "Investigation of a partial air-evacuated v-grooved solar water heater in Kenya," *Journal of sustainable research in engineering*, vol. 4, no. 3, pp. 90-98, 2018

AUTHOR PROFILE



Ali Omer M.Sc. (Pursuing), research areas fluid dynamics, heat transfer, solar collectors, member of SEC.



Dr. Hiram Ndiritu, Ph.D., principle CoTec JKUAT, published over 31 articles and books, corporate member of EBK & IEK.



Dr. Stephen Wanjii, Ph.D. (thermo-fluids), a staff member of JKUAT & IEK, research areas fluid dynamics, heat and mass transfer.