Numerical Study of Photovoltaic Panel Thermal Efficiency Using Multi-Cooling Process

Atheer Raheem Abdullah, Bassim Mohammed Majel

Department of Refrigeration and Air Conditioning Engineering, Al-Rafidain University College, Baghdad 10001, Iraq

Corresponding Author Email: basim.mohammed@ruc.edu.iq

https://doi.org/10.18280/iijht.400429

Received: 10 July 2022
Accepted: 20 August 2022

Keywords:
photovoltaic cell (PV), solar energy, solar thermal collectors

ABSTRACT

Renewable energy sources are becoming more and more popular, regarding the pollution and non-sustainability of common energy sources. Photovoltaic is the most direct way to convert solar radiation into electricity using the photovoltaic effect. This technology generates direct current (DC) electrical power from semiconductors when they are illuminated by photons. This paper presents a mathematical model in CFD that simulates the thermal presentation of a solar thermal photovoltaic panel equipped with circler, square and elliptic pin fin cooling module. The effect of cooling water velocity on photovoltaic panel average temperature and average cooling water temperature has been studied and the effect of cooling water velocity on photovoltaic panel average temperature and average cooling water temperature. The results show that velocity contours for water flows in circular, elliptic and square pin fin cooling module with inlet water velocities (0.0002, 0.0004 and 0.001 m/s). It is noticed that for square pin fin, the velocity between the pins is higher than the other types of pins, which leads to more circulation of water and more cooling.

1. INTRODUCTION

The growing using fossil fuel in various transference sectors, produces important attention in the global as a result of the environs contamination and worldwide warming [1]. The sun is regarded as a main resource of energy for its cleanliness and constancy, different other types of energy for example oil, coal, and derivations of oil that contaminate the environs and air.

Most researchers emphasize the significance of solar energy due to the abundance of sunshine capable of satisfying our energy wants in the next years [2]. Solar energy is the maximum plentiful renewable energy source on earth, so be the key for the rising request of worldwide energy consuming. Generally, the using of solar energy is distributed into two ways: photovoltaic (PV) and photo thermal [3]. Modules of PV are sold commercially providing with efficiency, rated power output, and other electrical features at standard test conditions (STC). These conditions are definite as trying underneath solar irradiance of 103 W.m\(^{-2}\) which is equal to 1 sun, pressure of 1.5 AM and ambient temperature of 25°C. Nevertheless, for the duration of process in outdoor conditions the presentation of the PV experience differences which are related with conditions of climate and industry of solar cell [4].

Cells of PV are expensive, and the PV devices features are extremely reliant on ecological circumstances. As a result, to confirm the extreme using of the obtainable solar energy via a PV power scheme, it is significant to study its performance over modeling, beforehand applying it in reality [5]. The climate is permanently varying and as many researchers are hard to plan methods to improve the effectiveness of photovoltaic panel. So, solar simulators are very useful in investigation of a solar energy. Solar simulators are a light resource providing lighting estimated to the usual sunlight. They are used for manipulating indoor of the several components and devices to evaluate, but sometimes for the study Photovoltaic cells, the cell features and presentation validation of components is attained [6].

Photovoltaic cells are affected via the temperature in a reluctant method owing to the negative temperature coefficient of crystalline silicon. The negative temperature coefficient was found to be in the choice of (0.3 to 0.65) %/°C depending on the kinds and material utilized of the photovoltaic panel. This is directed to the production power and electrical effectiveness of the photovoltaic panel decreased [7]. A suitable control and organization of the flow rate of air circulating over the PV/T collector will be an active mean to attain this aim and to confirm a valuable use of the device for many usages along with of the seasons. Actually, usage of direct photovoltaic powering of a DC fan delivers the probability of getting a solar irradiance reliant on air flow rate [8].

As a result, PV/T-air and PV/T-water systems have been widely investigated and different kinds of configurations developed to test the overall performance of the combined system, but this research introduce new methods to enhance the heat transfer rate and reduce the panel temperature using various sections of pin fins in addition to the traditional method of cooling, which is water.

In the current work, the thermal model has been prolonged to a three-dimensional geometry and thermo-fluid dynamic equations have been resolved via a Computational Fluid Dynamic (CFD) program. The algorithm is applied in the profitable software tool ANSYS/FLUENT, which gives the benefit of studying complicated geometries, producing both unstructured and structured meshes, and of resolving thermo-fluid dynamic equations with modified source terms via a
User-Defined Function (UDF), in which models of thermal and electrical have been completely coupled with the solver.

2. PHOTOVOLTAIC COMPOSITION

PV schemes are categorized in relation to how the components of scheme are connected to other power sources for example schemes of utility-interactive (UI) and stand-alone (SA). In a stand-alone system showed in Figure 1, the scheme is designed to run independent of the electric utility grid, and is usually planned and sized to deliver definite DC- and/or AC electrical loads [9].

Figure 1. Schematic diagram of stand-alone photovoltaic system [9]

3. SOLAR CONCENTRATORS

The way Integrating concentrators with photovoltaic cells can improve the production of power and decrease the cost of photovoltaic schemes. Several solar concentrator designs were used in the previous years for different systems of photovoltaic to attain high optical effectiveness [10]. Taking into consideration the optics of the refractor and reflector the solar concentrators categorized with two kinds that is imaging and nonimaging [11, 12]. As the name defines solar concentrator with imaging improve an optical image of the sun on the objective area for example linear lenses and solar tower, whereas solar concentrators with nonimaging optics simply concentrate the sunlight on aim but don’t grow slightly presence of the sun for example compound parabolic [13].

The acceptance angle ($\theta_s$) defined where the angle of incidence with respect to 90% of the extreme optical effectiveness at normal incidence [14, 15]. The supreme concentration of concentrator ($CR_{\text{max}}$) is a function of acceptance angle ($\theta_s$) and the surrounded dielectric material (n) refractive index. Regularly, three dimensions concentrator are utilized in attaining great concentration via reducing the acceptance angle, nevertheless complicated apparatus for example sun tracking is essential so as to preserve the concentrator directed at the sun [16, 17]. Additionally, small acceptance angle might cause optical damages to the scheme from misalignment producing removal of the CPV scheme total presentation [18]. Figure 2 demonstrates different solar collector shapes.

Figure 2. Various designs of solar concentrators: (a) Parabolic concentrator (b) Fresnel lens (c) Central receiver scheme with reflectors. (d) Tubular receivers (e) Plane receiver (f) Asymmetric parabolic concentrator [16]

4. NUMERICAL WORK

The numerical simulations permit the analysis of complex phenomena devoid of resorting to expensive prototypes and problematic investigational measurements. For the duration of the latest years, the Computational Fluid Dynamics (CFD) becomes one of the utmost powerful and suitable programs for predicting the temperature distribution and water flow behavior in environments that conditioned. Numerical modeling of heat transfer, flow, and related phenomena is of pronounced significance. With the development of computer technology, mathematical method has been recognized as the third method in scientific world, especially in the area of fluid dynamics [19]. It is an approval of the first two methods in scientific world, i.e., theoretical and investigational method.

4.1 PV panel creation of geometry

The geometry considered in the present work is a photovoltaic panel equipped with pin fin cooling module, the fins is of three shapes (square, circler and elliptic) in order to find the maximum thermal performance of a solar thermal photovoltaic panel and which of the aforementioned sections gives the highest value for the heat transfer rate. Figure 3-5 shows basic dimensions of the CAD geometry model.

Figure 3. Basic dimensions in (mm) of the panel model with square pin fin cooling module
For the computational model, the geometry is produced via utilizing solid works software to build the photovoltaic panel equipped with pin fin cooling module three-dimensional solid model. The surfaces of the model were then assigned to the subsequent sections:

a. Collector-Boundary
b. Collector-Water-Interface
c. Collector-outter

The subsequent step was to identify the fields in the model. The models in this study consisted of two material fields:

d. Solid Domain
e. Water Domain

4.2 Mesh generation

The mesh was produced by utilizing the volume meshing procedure and the choice to extrude the prism layers instantly afterward the tetra meshing had ended was chose. Three different meshes, $3.5 \times 10^6$, $4 \times 10^6$, and $4.5 \times 10^6$ tested and compared in terms of panel temperature. It is found that mesh number of around $4 \times 10^6$ gives about 1% deviation compared to mesh size of $4.5 \times 10^6$; whereas the results from mesh number of $3.5 \times 10^6$ deviate by up to 8% compared to those from the finest one. Therefore, a mesh of around $4 \times 10^6$ elements was considered sufficient for the numerical investigation purposes. Standard Computational Fluid Dynamics methods need a mesh that turns the boundaries of

the computational field. The production of computational mesh that is appropriate for the discretized solution of three-dimensional Navier-Stokes, continuity and energy equations has continuously been the subject of intensive researches. This type of problematic covers an extensive choice of engineering implementations.

4.3 Three-dimensional mesh generation

Mesh generation of solid geometry and three-dimensional models are more durable to be divided into following steps containing two main issues for additional controlling of the mesh. This may include the followings: As far as panel surface has been meshed, now volume mesh can be created. Building the mesh needs fine cells in area close the surface of panel. Oppositely, utilizing this size of part in the entire field would lead to a massive number of elements. That is why it was decided to utilize a fine mesh in the area close to the surface of panel and utilize coarse meshes as the space from the surface produces as shown in Figure 6. Table 1 and 2 show the momentum and thermal boundary conditions, respectively.

---

Figure 6. Mesh panel model with pin fin cooling module

Table 1. Momentum boundary conditions

<table>
<thead>
<tr>
<th>Part</th>
<th>Type</th>
<th>Momentum Conditions</th>
<th>Shear Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Panel</td>
<td>Wall</td>
<td>Stationary</td>
<td>No Slipping</td>
</tr>
<tr>
<td>Side Walls</td>
<td>Wall</td>
<td>Stationary</td>
<td>No Slipping</td>
</tr>
<tr>
<td>Cooling module inlet</td>
<td>Velocity Inlet</td>
<td>- Velocity Magnitude = (0.0002, 0.0004 and 0.001 m/s), or (0.002, 0.004 and 0.01 kg/s)</td>
<td></td>
</tr>
<tr>
<td>Pressure Outlet</td>
<td>- Gauge Pressure=(0 pascal), [constant].</td>
<td>Method: (Normal to Boundary).</td>
<td></td>
</tr>
</tbody>
</table>

Table 2. Thermal boundary conditions

<table>
<thead>
<tr>
<th>Part</th>
<th>Type</th>
<th>Thermal Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Panel</td>
<td>Wall</td>
<td>- Heat flux Rate=900 (W/m²).</td>
</tr>
<tr>
<td>Side Walls</td>
<td>Wall</td>
<td>- Wall Thickness=0.01 (m).</td>
</tr>
<tr>
<td>Cooling module inlet</td>
<td>Velocity Inlet</td>
<td>- Heat flux Rate=0 (W/m²).</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Temperature=25°C.</td>
</tr>
</tbody>
</table>
4.4 Basic governing equations

The conservation equations for continuity, momentum, and energy equations and turbulent model can be written as follows [20]:

Conservation of Mass

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0
\]

Navier-Stokes Equations

\[
\frac{\partial (\rho u u)}{\partial x} + \frac{\partial (\rho v u)}{\partial y} + \frac{\partial (\rho w u)}{\partial z} = -\frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) + S_{\text{inj}}
\]

\[
\frac{\partial (\rho v v)}{\partial y} + \frac{\partial (\rho w v)}{\partial z} = -\frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) + S_{\text{inj}}
\]

\[
\frac{\partial (\rho w w)}{\partial z} = -\frac{\partial p}{\partial z} + \mu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) + S_{\text{inj}}
\]

Energy Equation

\[
\rho \frac{\partial}{\partial x} (uT) + \rho \frac{\partial}{\partial y} (vT) + \rho \frac{\partial}{\partial z} (wT) = \frac{\partial}{\partial x} \left( \Gamma_{\text{eff, h}} \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( \Gamma_{\text{eff, h}} \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( \Gamma_{\text{eff, h}} \frac{\partial T}{\partial z} \right) + S_{T}
\]

5. RESULTS AND DISCUSSION

A model of the PV panel with pin fin cooling module which has the following dimensions (L=1m, W=0.5m and H=0.03m), was taken in this study. Many computational runs were performed at various mean panel temperature, air inlet temperature and velocity. One method of offering data graphically is to display slices (plane) of the flow in which (y=0.02m) coordinate is held constant. Wholly graphic outcomes are schemed on the plane. All slices, sections, temperature contours and velocity vectors for the numerical work were presented.

5.1 Velocity contours

The plots and velocity contours are presented in this section for all the simulated cases with different inlet water velocities were considered, i.e. (0.0002, 0.0004 and 0.001 m/s). The water flow is designed to pass the pins which is used to cool the photovoltaic panel as a heat sink, this will lower the photovoltaic panel temperature to increase its efficiency. The velocity contours are shown in Figures 7-13 below for water flows in circular, elliptic and square pin fin cooling module respectively. It is noticed that for square pin fin, the velocity between the pins is higher than the other types of pins, this is because the sharp edges of the square shape work to separate the thermal boundary layer and thus increase the speed which leads to more circulation of water and more cooling.

Figure 14 shows the panel temperature versus inlet velocities, and the square pin fin cooling module is the best choice for design optimization due to lower panel temperature as compared to the other types, and as the velocity increases the panel surface temperature decreases, because the high velocities lead to disrupt the boundary layer, this leads to an increase in the rate of heat transfer which is considered a good indicator to increase the efficiency of the solar panel.

Figure 15 demonstrates the effect of working fluid velocity on cooling water average temperature, as a result the water temperature decreases with the increasing of inlet velocity due to the because of the increase in the heat transfer coefficient and radiation.
Figure 9. Temperatures & velocity contours for photovoltaic panel equipped with circular pin fin cooling module at $T_{in}=25^\circ$C, $V_{in}=0.001$ m/s

Figure 10. Temperatures & velocity contours for photovoltaic panel equipped with elliptic pin fin cooling module at $T_{in}=25^\circ$C, $V_{in}=0.0002$ m/s

Figure 11. Temperatures & velocity contours for photovoltaic panel equipped with elliptic pin fin cooling module at $T_{in}=25^\circ$C, $V_{in}=0.0004$ m/s

Figure 12. Temperatures & velocity contours for photovoltaic panel equipped with elliptic pin fin cooling module at $T_{in}=25^\circ$C, $V_{in}=0.001$ m/s
5.2 Temperature contours

In order to show the effect of water velocity on the temperature distribution shown in figures above, the temperature contours for \( T_{in}=25^\circ C \) as a cooling medium temperature. It is noticed that as the cooling water velocity increase, the photovoltaic panel temperature decreases which is the scope of the study for all types of pin fins used. And the square pin fin has a lower water temperature and photovoltaic panel temperature, the elliptic pin fins have higher water temperature and photovoltaic panel temperature, the circular one is between them. So, the square pin fins cooling module is better in achieving the lower temperature of photovoltaic panel and increase its effectiveness, and this fact is clear which displays the influence of cooling water velocity on photovoltaic panel average temperature and average cooling water temperature.

6. CONCLUSIONS

The present work studies the performance of cooling photovoltaic panel equipped with various pin fin cooling modules. A model of the PV panel with pin fin cooling module which has the following dimensions \((L=1\text{m}\times W=0.5\text{m}\times H=0.03\text{m})\), was taken in this study. According to the previous discussion of the obtained results, the following points can be concluded:

1. The plots of the velocity contours are presented in this section for all the simulated cases with inlet water velocities were considered, i.e. (0.0002, 0.0004 and 0.001 m/s).
2. It is noticed that for square pin fin, the velocity between the pins is higher than the other types of pins, which leads to more circulation of water and more cooling.
3. In order to show the effect of water velocity on the temperature distribution, the temperature contours for \( T_{in}=25^\circ C \) was considered as a cooling medium temperature and it is noticed that as the cooling water velocity increase, the photovoltaic panel temperature decreases which is the scope of the study for all types of pin fins used.
4. The square pin fins cooling module is batter in achieving the lower temperature of photovoltaic panel and increase its effectiveness.

REFERENCES


