PERFORMANCE ENHANCEMENT OF PHOTOVOLTAIC SOLAR CELLS BY INTRODUCING CIRCULAR RIBS: - A NUMERICAL ANALYSIS

Rajat Kumar Mehta, Mahendra Rai, Akash Deep Gupta and Yashi Gupta

Mahamaya College of Agricultural Engineering and Technology, Ambedkar Nagar (Acharya Narendra Deva University of Agriculture & Technology, Kumarganj, Ayodhya)

DOI: http://dx.doi.org/10.24327/ijsr.2020.1106.5391

INTRODUCTION

Demand of electric energy is increasing day by day. Conventional energy sources impact environment and running cost of production is also increasing significantly. Solar PV technologies is one of the prominent way to trim down the cost electricity generation. PV converters are semiconductor devices that convert the incident solar radiation into electricity while rest part is converted into heat. PV cells are used in a wide range of applications. Power generated by PV system can directly used to run wide range of application and can also store in batteries. PV energy is used also as the power supply source for most of launched satellites [1]. PV cells can be produced in a form of multi-junction solar cells [2], or single-junction AlGaAs/GaAs solar cells with a maximum efficiency of around 25% at high concentration ratios [3]. An inclusive evaluation of four different solar cells, including single crystalline, polycrystalline, gallium arsenide and super solar cells has been done by Li et al. [4]. A small fraction of absorbed sunlight is converted directly into electricity in PV cells. In most of researches, it is found that PV cells efficiency is less than 20% for silicon solar cells and around 40% for multi-junction solar cells [2]. The remaining 80% or 60% of the absorbed energy are converted into heat, which will cause a temperature rise of the PV cells. Most studies [5-7] claim that the maximum power produced varies almost linear with the operating temperature. It is reported that the voltage reduction, causing a drop of the maximum power generated [8]. The efficient use of solar energy in PV systems can be achieved not only by using electrical energy output but also by using the dissipated heat from the solar PV. PV thermal PV/T systems can be classified based on either water (PV/W), air (PV/A) or different types of nanofluid are used as coolant [9]. One can classify the PV cooling systems depending also on the cooling mechanism: passive or active cooling [10]. A large number of researches have been published in the field of cooling PV. Water is generally preferred compared to air in PV/T solar systems [11]. Different cooling design arrangements were studied numerically and experimentally. Some of these researches focused on cooling the PV cells from the back using different micro-scale [12-16] or macro-scale [17-19] back thermal absorbers. Others used both back and front film cooling techniques [20, 21]. The influence of the geometrical parameters on the performance of cooling PV cell for PV/T applications is not limited to the micro-scale only. Different configurations of heat sink were studied...
they theoretically by Ibrahim et al. [22]. They used policrystalline PV cells cooled by different thermal absorber configurations attached to the back of the cell. They concluded that under water flow rate of 0.01 kg/s spiral flow design proved to be the best design with the highest thermal and electrical cell efficiencies of 50.12% and 11.98% respectively. Chow et al. [23] numerically investigated the electric and thermal performances of a PV/T system using water thermo-syphon. They studied three different types of absorber. They achieved maximum thermal and electrical efficiencies of 66.8% and 12.1% respectively for the PV module with an absorber covering 50% of its surface. A hybrid PVT solar system was designed and studied by Teo et al. [24]. They actively cooled the PV cells by a parallel array of ducts with inlet/outlet manifold such that the airflow is uniform. They compared the efficiency of the uncooled PV module with the cooled one. The un-cooled PV efficiency ranged between 8 and 9%, while with active cooling, the solar cell efficiency ranged between 12 and 14%. The daily average thermal and electrical efficiencies were about 50.5% and 10.5% respectively. Rajeb et al. [25] studied numerically the influence of optical and meteorological parameters on the heat transfer in a PV/T water system for different geometric configurations of the cooling system. The results showed that the electrical efficiency increases when the wind speed increases but decreases when the space between the tubes is larger. Contrary to this, the thermal efficiency increases when the wind speed and the space between the tubes decrease. Baloch et al. [26] carried out an experimental and numerical study on the cooling of a PV panel by a convergent channel heat exchanger and compared to another un-cooled system. Slimani et al. [27] carried out a comparative study of the electrical and thermal performances of four PV/T configurations under climatic conditions of Algeria. Their experimental and numerical results are compared favorably to those found in the literature. Their results showed that the electrical efficiency is high when the wind speed is higher than that of the heat transfer fluid. It has also been found that the overall energy efficiency increases by adding the number of glazing above the panel and an absorber above the insulation.

In this work, the cooling of a solar panel is achieved by imposing a laminar/turbulent air flow through a channel located below the PV panel, as proposed by Teo et al. [24]. Various ribs configurations of circular ribs are considered as well as a wide range of meteorological conditions (irradiation, wind speed, ambient temperature). The results are discussed in terms of the electrical and thermal efficiencies and the PV cell and outlet air temperatures.

### Numerical Modeling

#### Geometrical Modeling

The PV/T collector system is composed of three different layers, namely a glass layer (whose thickness is e=3 mm), the PV cell (e=0.2 mm) and a Tedlar layer (e=0.1 mm). The thermo-physical properties of the materials are isotropic and independent of temperature. They are displayed in Table I. The thermal contact resistances between each layer constituting the solar cell are neglected.

<p>| Table I Optical and thermo physical properties of the layers constituting the pv module [28] |
|---------------------------------|-----|-----|-----|</p>
<table>
<thead>
<tr>
<th>Optical Properties</th>
<th>Material</th>
<th>Absorptivity</th>
<th>Transmittivity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Glass</td>
<td>0.04</td>
<td>0.92</td>
<td>0.85</td>
</tr>
<tr>
<td>PV cell</td>
<td>0.9</td>
<td>0.02</td>
<td>-</td>
</tr>
<tr>
<td>Tedlar</td>
<td>0.128</td>
<td>0.012</td>
<td>0.9</td>
</tr>
<tr>
<td>Material</td>
<td>ρ (kg.m⁻³), Cₚ (J.kg⁻¹.K⁻¹), k (W.m⁻¹.K⁻¹)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Optical Properties</td>
<td>Glass</td>
<td>2450</td>
<td>500</td>
</tr>
<tr>
<td>PV cell</td>
<td>2330</td>
<td>677</td>
<td>148</td>
</tr>
<tr>
<td>Tedlar</td>
<td>1200</td>
<td>1250</td>
<td>0.36</td>
</tr>
</tbody>
</table>

The PV cell is cooled from below by air flowing in a rectangular channel. Air is assumed to be Newtonian and incompressible. The heat transfer and fluid flow are simulated in 2D. The PV/T system dimensions are the same as reported in [28]. The channel dimensions are L=1.2 m and H=20 mm. Circular ribs (h = 1.5 mm, l = 20 mm) may be added in the channel to improve the cooling of the solar cell (Fig. 1). Pitch of ribs is taken 6 to 10 time of ribs height to take thermo-hydraulic advantage as reported by various investigators for solar air heaters [30-34].

![Fig 1 Circular ribs configuration underside of the PV](image)

**Thermo-fluid Model**

The multi-mode heat transfers within the solid (glass, PV cell, Tedlar) and fluid layers are solved by considering the following energy equations:

**Solid domain**

\[ \rho c_{pi} \frac{\partial T_i(x,y)}{\partial t} = \nabla q_i + Q_{ext,i} \]  

**Fluid domain**

\[ \rho_f c_{pf} \frac{\partial T_f(x,y)}{\partial t} + \rho_f c_{pf} \mu \nabla^2 T_f(x,y) = \nabla q_f + Q_{ext,f} \]  

Where the index I and f refer to the i-th solid layer and to the fluid, respectively, ρ is the density, C_p is the specific heat capacity, T(x,y) is the temperature field, u(x,y) is the velocity field and t is the time. q represents the heat transfer by conduction, Q_{ext} is the heat sources/sinks, and Q_{ext} is the viscous dissipation.

The velocity field is obtained by solving the conservative continuity and momentum equations. In the turbulent regime, the Reynolds-Averaged Navier-Stokes equations are closed using a standard k-ε model in its low-Reynolds formulation.

The term Q_{ext} is the sum of the thermal losses by convection towards the front and the rear faces of the panel, denoted Q_{1}, and of the heat gained by the incident solar radiation, denoted Q_{2}. Note that the thermal losses on the side surfaces are neglected and the bottom wall of the flow channel is perfectly insulated. The term Q_{1} is defined as:
Q_{c}=h_{c}A(T_{pv}-T_{a}) \tag{3}

where $h_{c}$ is the heat convection coefficient, $A$ the exchange area, $T_{pv}$ the temperature of the PV cell and $T_{a}$ the ambient temperature. The coefficient $h_{c}$ is given by [27]:

$h_{c}=5.82+4.07w \tag{4}$

with $w$ the wind speed

The incident solar radiation $Q_{s}$ on the PV module is calculated as follows [28]:

$Q_{s}=\alpha AG(1-\eta_{pv})/V \tag{5}$

where $\eta_{pv}$ is the PV cell electrical efficiency, which is equal to zero for the other layers, $G$ the solar irradiation, $\alpha$ the absorptivity and $A$ and $V$ the surface and the volume of the different layers respectively.

In order to calculate the internal energy generated in the solar cells, the iterative method proposed by Zhou et al. [28] is used to solve the dependence between the electrical efficiency, the internal energy and the cell temperature. The electrical efficiency of the solar cells is calculated as:

$\eta_{pv} = \eta_{ref}[1-\beta_{ref}(T_{pv}-T_{ref})] \tag{6}$

where $\eta_{ref}$ and $\beta_{ref}$ are the solar cell efficiency and temperature coefficient at a reference temperature of $T_{ref}=25 \degree C$, respectively. The reference solar radiation is assumed to be $G=1000 \ W.m^{-2}$.

**Numerical Method and Boundary Conditions**

The CFD tool ANSYS Fluent 2019 R3 based on the finite-volume method is used to solve the present conjugated heat transfer problem. The calculations are 2D steady-state. Second-order schemes are used to discretize the spatial derivatives. The algorithm SIMPLE is chosen to overcome the pressure-velocity coupling. The gradients are computed according to the Least Squares Cell-Based method. The turbulent Prandtl number is fixed to 0.744.

The mesh grid done using ANSYS Workbench 2019 R3 is composed of 172329 unstructured triangular elements (132329 elements for the fluid, 14 000 for the solar cell and the tedlar layer and 26000 for the glass layer) with a refinement in the fluid domain close to the walls to guarantee a wall coordinate lower than 1 for all calculations involving turbulent flows. It has been carefully checked that this mesh grid provides grid-independent solutions.

**Validation of the Numerical Model**

The present thermal model is validated by comparing the present results, first to the numerical results of Zhou et al.[28] obtained using a finite-element solver in the case of a PV module without cooling and then to the numerical and experimental results of Slimani et al. [26] for a PV/T system. Figure 2 (a) displays the effect of the ambient temperature on the heat dissipation of the PV module for different solar irradiances at a prescribed wind speed $w=1 \ m.s^{-1}$. $\Delta T$ represents the difference between the maximum cell temperature and the ambient temperature and then is a direct measure of the heat dissipation of the PV module. The temperature difference increases linearly when the solar irradiance increases. When the ambient temperature increases from -10 °C to 40 °C, the temperature difference decreases slightly, and the decrease is more obvious under high solar irradiance.

The influence of the wind speed on the heat dissipation is shown in Fig. 2 (b) for two ambient air temperature ($T_{a}=283$ K and 313 K) and $G=1000 \ W.m^{-2}$. The heat dissipation (or $\Delta T$) decreases exponentially when the wind speed increases from 0 to 8 m.s$^{-1}$. At fixed ambient temperature, the highest cell temperature decreases then in the same manner. At constant wind speed, the influence of the ambient temperature is weak and even vanishes for $w=8 \ m.s^{-1}$.

More interestingly, the present model compares fairly well with model developed by Zhou et al. [28]. It confirms the good implementation of the model within ANSYS Fluent and validates it for a PV cell without cooling.

![Fig. 2](a) Influence of the atmospheric temperature ($w=1 \ m.s^{-1}$) and (b) the wind speed ($G=1000 \ W.m^{-2}$) on the heat dissipation of the PV module. Comparisons with the numerical results of Zhou et al. [29]

**RESULTS AND DISCUSSION**

In this study, six configurations are considered: a simple channel without rib and a channel with circular cross section ribs having p/h 6, 7, 8, 9, 10. The channel height is fixed to $H=20 \ mm$ in all the simulations, some simulations are also performed for other channels heights. In the following part, the
solar flux when not specified is fixed to $G=1000 \text{ W.m}^{-2}$, and the ambient and inlet fluid temperatures are set to 30 °C and 27 °C, respectively. The working fluid is air.

**Thermal and Hydrodynamic Flow Fields**

One considers first a baseline case with regularly spaced circular ribs beside of PV cells for $h/H = 8$. The operating conditions are fixed to $w=1 \text{ m.s}^{-1}$ and $V_{in}=3.5 \text{ m.s}^{-1}$. The bulk Reynolds number based on the channel height is then equal to $Re_H =4666$ in the present case, such that the flow is turbulent. The objective is here to better understand how the ribs may affect the fluid flow and thermal field for a baseline case before quantifying their influence onto the electrical and thermal efficiencies of the module as a function of the operating parameters.

Figure 3 displays the temperature and velocity maps. The flow and boundary layers develop first between the inlet and the first rib. Then, a strong acceleration occurs closer of the rib to conserve the mass flow rate. In the wake of the rib, a large recirculation bubble forms behind the circular rib. Its length is 5 times the length of the rib for this particular value of the Reynolds number. The flow acceleration and the recirculation area are associated with particularly high levels of the turbulence kinetic energy. The maximum value of the eddy viscosity $\mu=5.68 \times 10^{-3} \text{ Pa.s}$ is observed at the center of the recirculation zone. Being two orders of magnitude higher than the dynamic viscosity of air, it confirms firstly that the $k-\varepsilon$ model acts more especially in this flow region and secondly that the flow is highly turbulent there. Turbulence is known to greatly enhance mixing and so the convective heat transfer. The flow then decelerates due to the enlarging of the effective area and the same phenomena repeat for each rib.

The fluid warms at the contact with the tedlar layer and by thermal diffusion, the temperature slightly increases in the fluid further away from the top wall. The recirculation zones mix the hot and cold fluids. After the third rib, three hot fluid regions form in the solid layers (not shown here). It leads to a well homogenized and higher fluid temperature in the wake of the last rib by comparison with the fluid temperature at the inlet. Such a configuration appears then to be efficient for extracting heat from the PV module and removing it by forced convection.

The same calculation has been done by imposing the flow from the right such that the flow faces the back of the circular rib. On one hand, it induces a higher pressure drop (1368.2 Pa) compared to the base case (853.6 Pa), but the maximum temperature within the PV cell is 7.1 K lower. For a thermal point of view, it is then more efficient to extract heat from the PV cell, but it requires a higher pumping power.

Other calculations have been performed for different blocking ratio $h/H$. They all lead to lower performances in terms of thermal and electrical efficiencies such that the present value $h/H=0.75$ appears to be optimal. However, from this calculation, it is clear that the inter-rib spacing could be optimized also as a function of the inlet fluid velocity to get even better performances. Typically, the inter-rib distance should be set to the length of the recirculation zone formed in the wake of the rib, which depends on the inlet velocity.

**Effect of the Inlet Fluid Velocity**

The effect of the inlet air velocity on the PV/T system performance is considered for various configurations of circular cross section ribs having $p/h = 6, 7, 8, 9, 10$ circular ribs. Their dimensions are fixed to $h=1.5 \text{ mm}$ ($h/H=0.075$) and $l=20 \text{ mm}$ ($l/L=1/60$). The wind speed and solar irradiance are also set to $1 \text{ m.s}^{-1}$ and 1000 W.m$^{-2}$, respectively.

Fig. 4 presents the influence of the inlet air velocity on the PV cell temperatures. For the case without rib, the present results confirm the former ones of Baloch et al. [26] and Radwan et al. [29]. Both temperatures decrease exponentially with increasing inlet air velocity. More interestingly, with application of circular ribs, the PV cell temperature drops by about 10 °C compared to the base case without rib at any given inlet air velocity between 1 and 3.5 m/s. The presence of the ribs induces large recirculation zones (Fig. 3) such that cooled air close to the bottom of the channel is partly re-injected to the top to extract heat from the PV cell. It increases the overall heat transfer within the channel and as a consequence the outlet air temperature increases. Increasing the inlet air velocity from 1 to 3.5 m.s$^{-1}$ leads to a transition from a laminar regime ($Re_H =1333$ at 1 m.s$^{-1}$) to a turbulent one ($Re_H =4666$ at 3.5 m.s$^{-1}$). This transition does not affect significantly the variation rate of the PV cell temperature with the inlet air velocity.
The variations of the electrical and thermal efficiencies with the inlet air velocity are illustrated in Fig. 6. The thermal efficiency $\eta_{th}$ is defined as:

$$\eta_{th} = \frac{\dot{m}c_p(\bar{T}_{out} - \bar{T}_{in})}{A \cdot G}$$  \hspace{1cm} (7)

where $\bar{T}_{in}$ and $\bar{T}_{out}$ are the average inlet and outlet air temperatures within the channel, respectively.

As the electrical efficiency depends on PV cell temperature ($\eta_{pv}$ is proportional to $-T_{pv}$, see (6)), it increases with increasing inlet air velocity. Though the rib configuration $P/h=8$ appears to provide better performances, the electrical efficiency remains in a narrow range [10.0 -11.5 %]. In the same way, the thermal efficiency is directly proportional to the outlet air temperature (7), such that $\eta_{th}$ increases for increasing values of the outlet air temperature due to higher inlet velocities. The variations of the thermal efficiency are much higher.

**Influence of the Solar Irradiance**

The influence of the solar irradiance on the cell and outlet air temperatures as well as on the electrical and thermal efficiencies is reported in Fig. 6 for the configuration with five ribs, an inlet velocity fixed to 2 m.s$^{-1}$ and three values of the wind speed. The solar irradiance represents the most important environmental parameter that affects the PV panel’s performance. Increasing the solar irradiation between 200 W/m$^2$ and 1000 W/m$^2$ leads to an increase of the PV cell and outlet air temperatures. The wind speed does not significantly affect the cell and outlet air temperatures. As expected, higher wind speeds induce a higher dissipation of heat by forced convection at the top of the PV panel. The PV cell temperature decreases, and air flow has less heat to extract such that the outlet air temperature decreases too.
wind speed decreases, but the influence of the wind speed gets lower at very low values of $G$ (around 200 W/m$^2$).

**CONCLUSION**

A two-dimensional numerical model has been developed using ANSYS Fluent and favorably compared to results available in the literature. It has been then extensively used to determine the thermal and electrical performances of a solar PV cell for different environmental conditions and geometrical configurations of the cooling duct including or not circular ribs. The following conclusions can be drawn:

The environmental conditions are uncontrollable parameters. As expected, low solar irradiation and high wind speed are favourable to get lower PV cell temperature and higher electrical efficiency.

For fixed meteorological conditions, increasing the number of circular ribs decreases the PV cell temperature and increases the outlet air temperature. As a consequence, both electrical and thermal efficiencies of the module are enhanced. A blocking ratio of $b/H=0.075$ seems to provide the best overall performance. The inter-rib spacing should be optimized for all values of the inlet air velocity. This parameter plays indeed an important role to cool down the PV module, turbulent air flows enhance the heat transfer by forced convection within the channel.

Further calculations are now required to investigate more complex geometries, including micro-channels of variable height. The use of more advanced fluids, such as nanorefrigerants, is also a possible way to increase the thermal efficiency of PV/T solar panels.

**References**


DOI: http://dx.doi.org/10.24327/ijrshr.2020.1106.5391

How to cite this article:

DOI: http://dx.doi.org/10.24327/ijrshr.2020.1106.5391


34. http://krishikosh.egranth.ac.in/handle/1/5810057953