DOI: https://doi.org/10.54966/jreen.v1i1.1238

Journal of Renewable Energies

Revue des Energies Renouvelables

journal home page: https://revue.cder.dz/index.php/rer

Conference paper

Numerical Study of a Heat Transfer in a Photovoltaic (PV) Panel Water Cooling Systems

Ridha Hannat *

Faculty of Technology, Setif 1 University -Ferhat Abbas, route de Bejaia, Setif, Algeria

ARTICLE INFO

Article history:

Received July 31, 2024
Accepted September 4, 2024

Keywords:

Photovoltaic,
Cooling,
CHT,
Heat transfer,
CFX.

ABSTRACT

The proposed work uses the CFD commercial flow solver Ansys CFX. This solver is a finite volume-based method. Ansys CFX is used to compute heat transfer at the fluid-solid interface using its CHT module to improve the efficiency of the cooling PV system. The cooling PV system proposed by Wilson is the best-chosen design since it uses a confined water flow and then no water is lost or evaporated also it uses the hydraulic head of water from a tank without the use of a circulating pump. According to the last studies of a confined flow, the $k - \omega$ turbulent model of Wilcox is selected to study heat transfer inside a 3D geometry proposed by Wilson. The recirculation area effect on heat transfer lowers the wall heat flux at the solid-fluid interface so this recirculation area must be avoided. To achieve the recirculation area, different cooling PV systems geometries can be investigated in future studies.

1. INTRODUCTION

It has been established that the PV cell's efficiency declines as the cell temperature is high (Thong et al., 2017), (Thong et al., 2016). Chaib and al. (Chaib et al., 2023) demonstrate that increasing temperature reduces the open-circuit voltage of the photovoltaic panel. There are different cooling PV systems but generally water spray jets have been used to cool the external surface PV panels to imitate rain or a water film (Lubon et al., 2020). The earlier cooling system proposed by Ramanan and al. (Ramanan et al., 2024) is the floating solar PV or FPV. The installation of solar PV panels on floating platforms over water bodies but this system is not suitable for arid regions. The best system to ensure a good heat transfer a fluid-solid direct contact is used to cool the internal surface (Wilson, 2009) (Bahaidarah et al., 2013) or indirect contact using a collector pipe (Arefin, 2019).

The fluid-solid direct contact is the best design, yet no experimental or numerical studies are realized. The proposed work uses the commercial flow solver Ansys CFX to compute conjugate heat transfer

* *Corresponding author, E-mail address: ridha.hannat@univ-setif.dz*

ISSN: 1112-2242 / EISSN: 2716-8247

CHT of CFX at the fluid-solid interface to estimate the wall heat flux by the use of the average Nusselt number.

2. MATHEMATICAL MODEL AND NUMERICAL METHOD

Ansys-CFX solves the R.A.N.S equations to obtain the heat exchanged at the fluid-solid interface. The Navier-Stokes equations for turbulent flow can be found in Wilcox (Wilcox, 2006). Ansys-CFX uses a collocated finite volume method to solve RANS equations. The gas law equation is used to relate density to pressure and temperature. From the three conservation equations, a coupled system of linearized equations is built and solved by Ansys-CFX. Using the velocity field and length scale obtained, the $k - \omega$ equations are then solved. Iteration between the RANS equation and turbulence equations continues until the residual falls below a level set by the user. In our case, the residual must be below 10^{-5}.

To avoid meaningless oscillations in the numerical solution, the blend factor advection scheme, with a value of 0.85, is used for the calculations. With this value, a near-second order is achieved.

3. TEST CASE SIMULATION

Wilson (Wilson, 2009) studied experimentally the gravity-fed cooling technique applied to a photovoltaic module exploiting the hydraulic head of water from a tank without the use of a circulating pump. The cooling PV system proposed by Wilson (Wilson, 2009) is chosen since it uses a confined water flow and hence no water is lost or evaporated. The PV panel is an eight-cell PV module, measuring 6 cm × 6 cm × 0.3 cm. The cooling PV system is a module measuring 6 cm × 6 cm × 1.5 cm. The water enters at the bottom of the module and leaves at the top. To have a fully developed velocity, the inlet, and the outlet are modeled as a duct with an inner diameter of 10mm and 15mm of length.

The geometry used is shown in Fig *1*. Both solid and fluid domains are merged in the same geometry to use the CHT at the fluid-solid interface in Ansys CFX (Fig *2*).

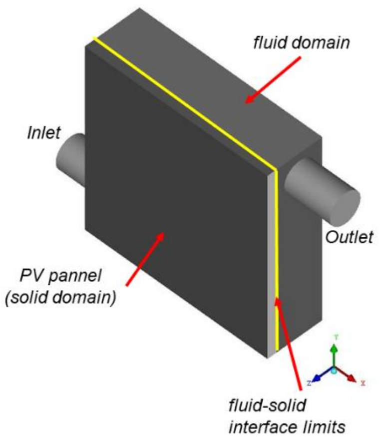

Fig 1 **.** Fluid and solid domains geometries

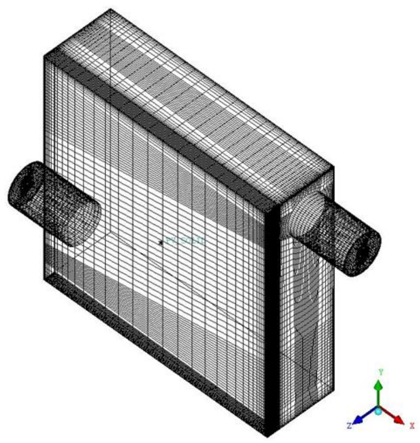

Fig 2 Fluid and solid merged meshes

In the present article, a volumetric debit of *02 l/min* is used. No need to use different debits since increasing debit will increase the heat transfer.

At the inlet, the water jet exits at a temperature $T_{in}=297$ K. The external PV panel wall is at ambient temperature $P_{PV}=333$ K. The Nusselt number is obtained along different line locations (the x direction or y direction).

The Nusselt number definition used is:

$$Nu = \frac{q''D}{(T_{PV} - T_{in})k} \tag{1}$$

For our study, the $k - \omega$ turbulent model in Ansys CFX is selected with a turbulent intensity of 1% at the inlet. the $k - \omega$ turbulent model in Ansys CFX is already validated in our previous studies (Hannat & Morency, 2012). However, investigation is needed before any conclusion about the turbulence model prediction in 3D cooling PV systems.

3.1 Mesh Study Headers

The Y^+ is estimated at all the walls in the fluid domain. Only two location lines are used here to show Y^+ values, on the middle horizontal and vertical lines at the fluid-solid interface. A hexahedral mesh generated with ANSYS-ICEM is generated such that $Y^+ < 1 \sim 5$, as shown in Fig 3 and Fig 4.

The node spacing grows following an exponential law with an initial ratio of 1.2 at all the inner walls. Mesh concentration is also imposed at the inlet and the outlet duct using O-grid technic in ICEM-CFD. Quantification of the numerical uncertainties is proposed by Boache (Boache, 1994) based on generalized Richardson extrapolation. For two different hexahedral meshes, the Grid Convergence Index (*GCI*) is expressed by:

$$GCI = \frac{3|\varepsilon|}{r^p - 1} \tag{2}$$

$$r^p = h_2 / h_1 \tag{3}$$

h_1 and h_2 : Node spacing for coarse and fine mesh.

1.6 10^6 nodes are used in the fine mesh and 0.2 10^6 nodes in the coarse mesh.

With a ratio of r=2 and a second-order method (p=2), the GCI is reduced to the error ε :

$$GCI = \varepsilon = \frac{\overline{Nu}_{fine} - \overline{Nu}_{coarse}}{\overline{Nu}_{fine}} \tag{4}$$

The area average Nusselt ($\overline{Nu_{coarse}}$) value obtained on the solid-fluid interface wall with a coarse mesh is compared to the area average Nusselt ($\overline{Nu_{fine}}$) value obtained with the finer grid. The *GCI* obtained for the present test case is 0.9 %. In the later studies, only the coarse mesh is used since the *GCI* is too small.

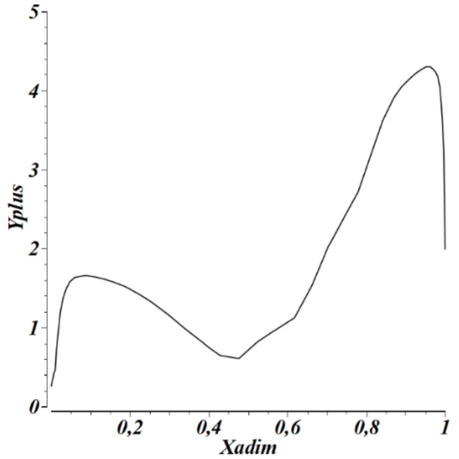

Fig 3. Y^+ along the horizontal middle line of the PV interface fluid-solid

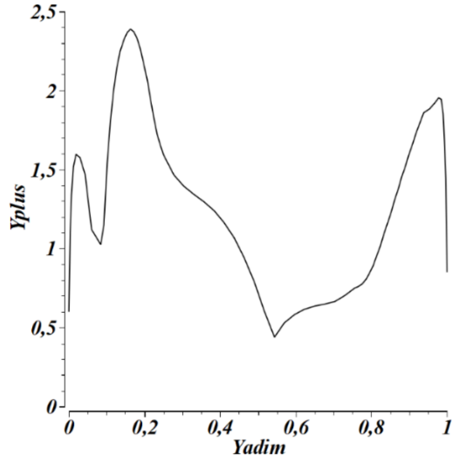

Fig 4. Y^+ along the middle vertical line of the PV interface fluid-solid

3.2 Mesh Study Headers

In Fig 5, the wall heat flux contours are plotted to compare results with the velocity streamlines in.Fig 6. The lower value of the wall heat flux occurs at the center of the fluid-solid interface as shown in Fig 5 corresponding to the large recirculation area in Fig 6. Heat transfer is higher at the jet impinging area and it gets lower in the recirculation area at the right-low corner. Low heat transfer at the center solid-fluid interface corresponds to the high total temperatures in Fig 7 and Fig 8. These results can be verified in Fig 9 where heat transfer is low at the center of the panel.

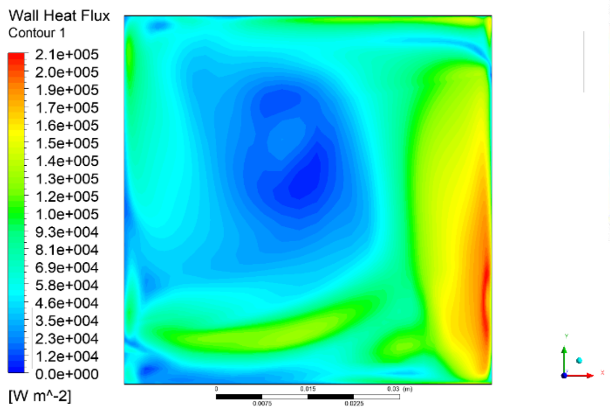

Fig 5. Wall heat flux at the fluid-solid interface

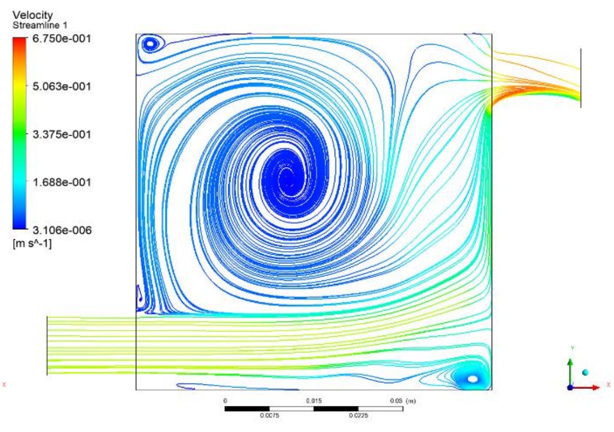

Fig 6. Surface velocity streamlines

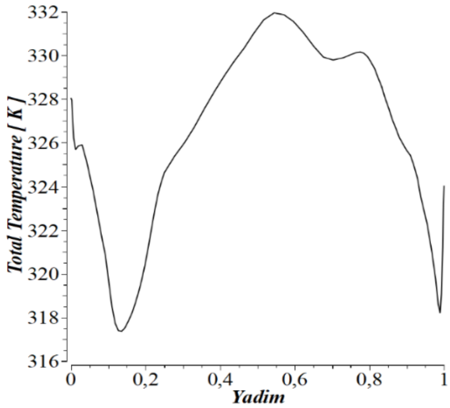

Fig 7. Total temperature at the perpendicular middle line of the fluid-solid interface

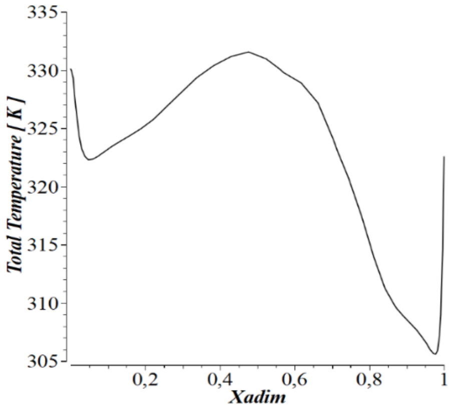

Fig 8. Total temperature at the horizontal middle line of the fluid-solid interface

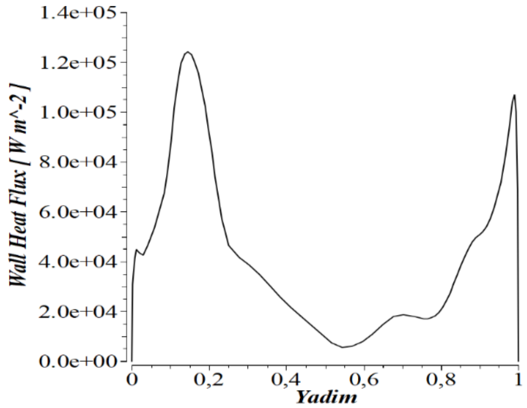

Fig 9. Wall heat flux at the perpendicular middle line of the fluid-solid interface

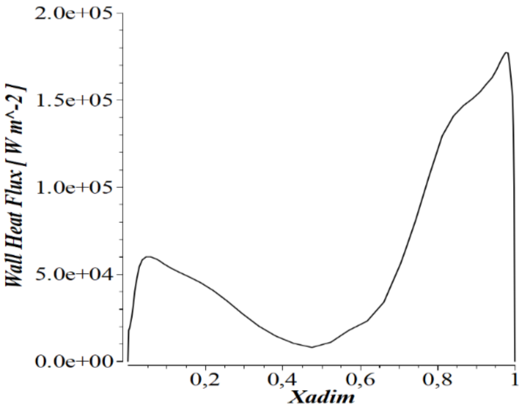

Fig 10. Wall heat flux at the horizontal middle line of the fluid-solid interface

The previous results can be summarized by using the Nusselt plot along the middle line at the solid-fluid interface as shown in Fig 11. According to the results from Fig 5, Fig 6, and Fig 11 increasing the wall heat flux needs to avoid recirculation area.

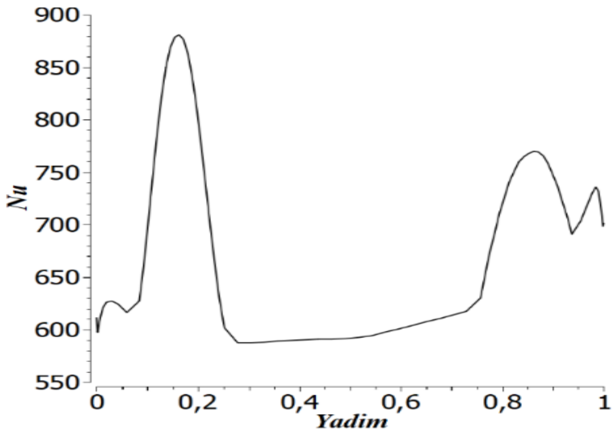

Fig 11**.** *Nu* at the perpendicular middle line of the fluid-solid interface

To avoid the recirculation area, the cooling PV system geometry must be modified by adding channels. This new geometry of the cooling PV system needs to be investigated in our future studies. The cooling PV system geometry can be divided into multichannel or it can be divided into countercurrent channels than a collector pipe used by Arfin (Arefin, 2019). Using an air piccolo system at the external PV panel can be investigated too.

4. CONCLUSION

In the present study, the cooling PV system proposed by Wilson (Wilson, 2009) is chosen since it uses a confined water flow, and hence no water is lost or evaporated. This study demonstrates that a large recirculation area can have large effects on heat transfer at the fluid-solid interface. To improve the heat transfer at the fluid-solid interface and hence avoid recirculation area, the cooling PV syste6m geometry must be modified by adding channels. Thus, Nusselt numbers predicted with Ansys-CFX at the center of the PV increase with the number of channels. There are no experimental results available to confirm our numerical results, but computation at low debit without channels predicts adequately the Nusselt number along the fluid-solid interface.

Agreement between experimental geometry and numerical geometry must be verified.

The numerical errors for the 3D results are conservatively estimated to be around 0.9%, which is very low from a point of engineering view. The small meshes enable us to easily run the numerical simulation on a personal computer.

NOMENCLATURE

CFD	Computational Fluid dynamics	ε	Error term
CHT	Conjugate Heat Transfer	h_1, h_2	Node spacing for coarse and fine mesh.
D	Inlet diameter (m)	q''	Wall heat flux *(W/m²)*
GCI	Grid Convergence Index	Y^+	Dimensionless *'y'* distance
k	Thermal conductivity *(W/m K)*		
Nu	Nusselt number		
PV	Photovoltaic		
p	Order method		
r	Ratio		
T	Temperature (*K*)		

REFERENCES

Arefin, M. A. (2019). Analysis of an Integrated Photovoltaic Thermal System by Top Surface Natural Circulation of Water. Frontiers in Energy Research, 7. https://doi.org/10.3389/fenrg.2019.00097

Bahaidarah, H., Subhan, A., Gandhidasan, P., & Rehman, S. (2013). Performance evaluation of a PV (photovoltaic) module by back surface water cooling for hot climatic conditions. Energy, 59, 445–453. https://doi.org/https://doi.org/10.1016/j.energy.2013.07.050

Boache, P. J. (1994). Perspective: A method for uniform reporting of grid refinement studies. Journal of Fluids Engineering, Transactions of the ASME, 116(3). https://doi.org/10.1115/1.2910291

Chaib, M., Abdeldjalil, D., Benatillah, A., Hachemi, N., Sakher, E., & Ben Abdelkarim, B. (2023). Modeling, simulation and analysis of the input climat parameter effect on the photovoltaic panel. 2nd

International Conference on Energy Transition and Security, ICETS 2023. https://doi.org/10.1109/ICETS60996.2023.10410824

Hannat, R., & Morency, F. (2012). Numerical validation of CHT3D/CFX in anti-/de-icing piccolo system. 4th AIAA Atmospheric and Space Environments Conference 2012. https://doi.org/10.2514/6.2012-2678

Lubon, W., Pełka, G., Janowski, M., Pajak, L., Stefaniuk, M., Kotyza, J., & Reczek, P. (2020). Assessing the impact of water cooling on PV modules efficiency. Energies, 13(10). https://doi.org/10.3390/en13102414

Ramanan, C. J., Lim, K. H., Kurnia, J. C., Roy, S., Bora, B. J., & Medhi, B. J. (2024). Design study on the parameters influencing the performance of floating solar PV. Renewable Energy, 223. https://doi.org/10.1016/j.renene.2024.120064

Thong, L. W., Murugan, S., Ng, P. K., & Sun, C. C. (2016). Analysis of photovoltaic panel temperature effects on its efficiency. 2nd International Conference on Electrical Engineering and Electronics Communication System 2016.

Thong, L. W., Murugan, S., Ng, P. K., & Sun, C. C. (2017). Energy efficiency analysis of photovoltaic panel on its operating temperature. Journal of Engineering and Applied Sciences, 12(14), 3692–3696. https://doi.org/10.3923/jeasci.2017.3692.3696

Wilcox, D. C. (2006). K-ω Model. In Turbulence Modeling for CFD. 3rd ed. DCW Industries, La Cañada, California

Wilson, E. (2009). Theoretical and operational thermal performance of a "wet" crystalline silicon PV module under Jamaican conditions. Renewable Energy, 34(6). https://doi.org/10.1016/j.renene.2008.10.024