Thermal stress analysis of photovoltaic modules using thermo-elastic modeling

Nasreddine El-Dehaibi
Energy Systems Engineering, University of Michigan, Ann Arbor, MI.

Introduction

Thermal stresses in photovoltaic (PV) modules can have a significant impact on their lifetime. The layered materials that constitute a PV panel are shown in figure 1. Each material possess a different coefficient of thermal expansion (CTE). This mismatch in the CTE can result in high thermal stresses within the PV module as ambient temperature changes throughout the day. Furthermore, the cyclic nature of ambient temperature can lead to fatigue failure in the PV module as the layers continuously expand and contract against each other.

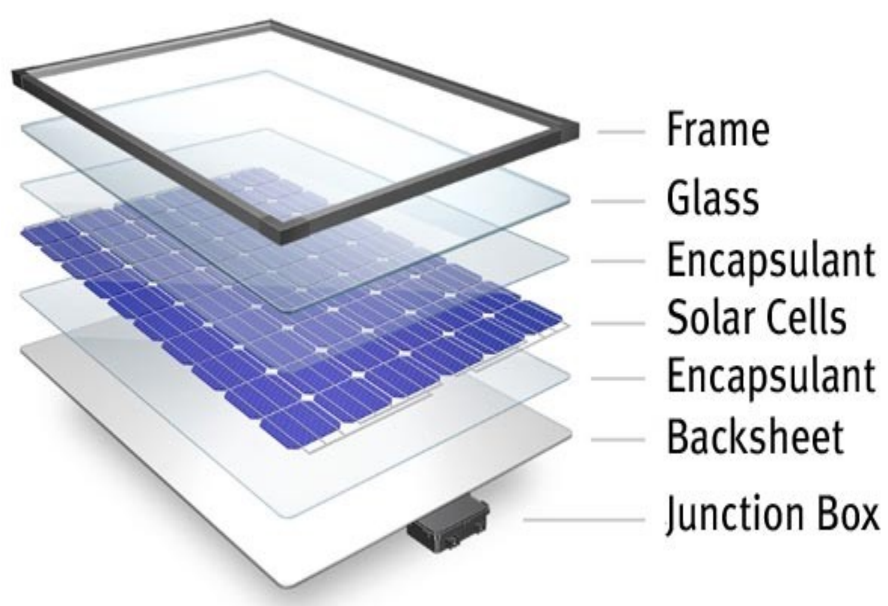

Figure 1: Layers of a typical PV Module.

Table 1: Text in Arial font size 20.

Objectives

Several works have gone into quantifying thermal stresses and understanding how they impact the lifetime of the modules. All mentioned works use finite-element-method (FEM) for developing the structural models and implement the IEC 61215 thermal cycle for determining thermal stresses [1,2,3,4]. The objective of this study is to:

1. Develop a simple analytical model that can quantify thermal stresses quickly and without the need of expensive software
2. Implement the analytical model using real temperature data and material properties from a commercial PV module
3. Validate the results from the analytical model using an FEM model

Methods

A thermo-elastic model is implemented in this study to calculate the thermal stresses in a PV module. The fundamental model used is

$$\{\sigma\} = \{D\}\{\varepsilon\}$$

The final representation of the model as used in this study is

$$\begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = \begin{bmatrix} E & \nu E & 0 \\ \nu E & E & 0 \\ 0 & 0 & G(1-\nu^2) \end{bmatrix} \begin{bmatrix} -\alpha\Delta T \\ -\alpha\Delta T \\ -2\alpha\Delta T \end{bmatrix}$$

The thermo-elastic model was implemented in MATLAB to compute thermal stresses in the PV module. Cell temperature data was obtained from a thermal model that uses ambient temperature and solar irradiance data from Riyadh in 2009.

The assumptions made for this model include that the PV module experiences elastic deformations only, each layer is an infinite plane, the junction box, frame, and metallic interconnects are neglected, the PV module undergoes normal stresses only, is isotropic, and undergoes plane stress, the stress-free state is at room temperature, and the temperature distribution in the PV module is uniform.

A 3D FEM was performed on Abaqus to validate the stress calculations from the thermo-elastic model. The idea was to model the PV module with as few simplifications as possible to see if the assumptions made in the thermo-elastic model are valid.

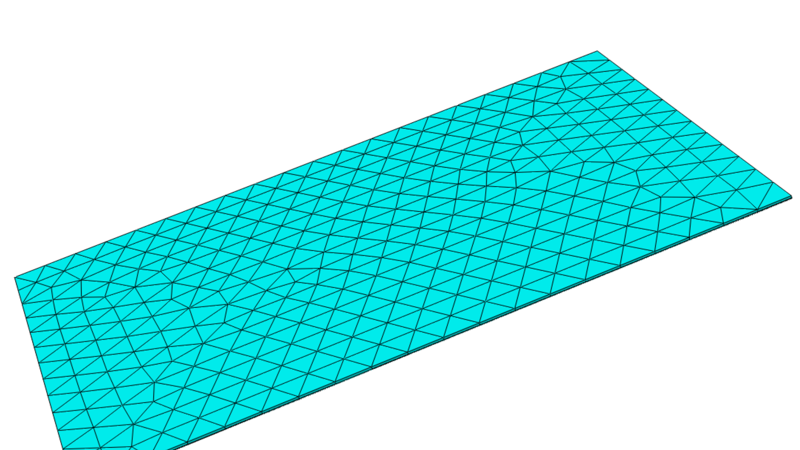

Figure 2: Mesh Model of PV Module

Figure 2 shows the mesh model created on Abaqus. Assumptions made in this model include that the junction box and metallic interconnects are neglected, the stress-free state is at room temperature, the temperature distribution in the PV module is uniform. All assumptions made are based on previous studies [1,2,3,4].

Results and Discussion

Figure 3 shows the average monthly stress value for each layer in the PV module. There is a clear correlation between the time of the year and the magnitude of the stress and strain; the hotter the month the greater the stress magnitude. This can be attributed to the larger temperature difference in a typical summer day compared to a winter day in Riyadh. The chart further suggests that the Hooke's Law model is accurate and applicable for this purpose.

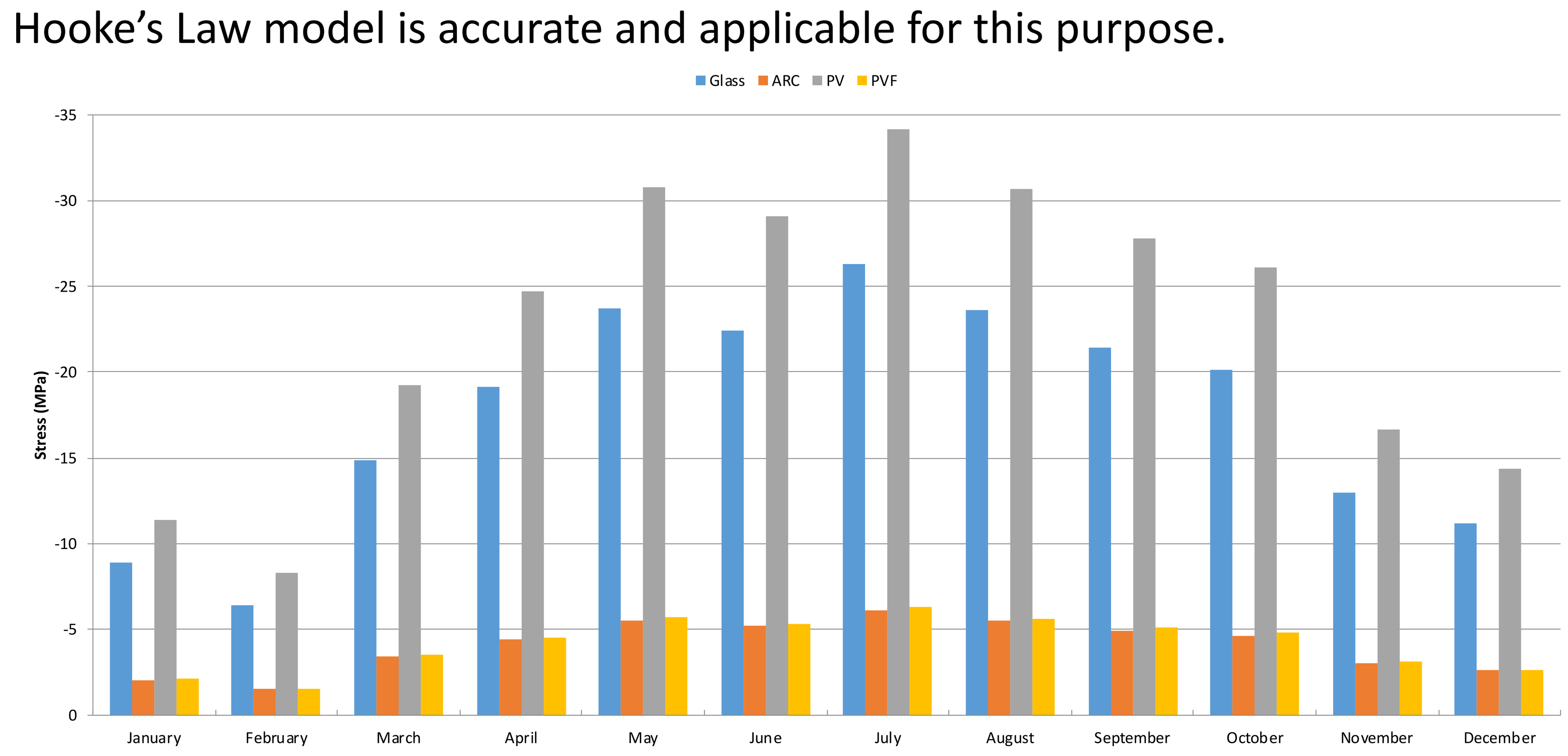

Figure 3: Average Monthly Stress Values

Comparing the FEM results with the results from thermo-elastic model in table 1, the numbers are almost identical. The largest percent difference was 0.89%, with the average being 0.36%.

	Thermo-Elastic Model (MPa)	FEM Model (MPa)	Percent Difference
Glass	21.5954	21.4986	0.45%
ARC	4.9993	5.0002	0.02%
PV	27.8124	27.7621	0.18%
PVF	5.1656	5.1569	0.17%

Conclusions

A simple thermo-elastic model is sufficient in performing a thermal stress analysis on PV modules. The assumptions of neglecting the encapsulant layers as well as external frames and fixtures have proved to be valid and have an insignificant impact on the thermal stress results. Without the encapsulant the PV module can be modeled as elastic and therefore makes thermo-elasticity applicable. This is a simple way of avoiding complex viscoelastic modeling as well as expensive CAE software for thermal stress analysis. Although the boundaries of the PV module cannot be modeled accurately with thermo-elasticity, it does not impact the results for the center of the module. Furthermore, it was demonstrated that an extensive thermal stress analysis spanning one year of temperature data can be simply executed with computational aid from MATLAB. This is contrary to FEM where a long-term analysis is far more complex and time consuming.

Acknowledgements

I would like to thank Dr. Nicolas Barth for his support with the thermal model and FEM models as well as Dr. Said Ahzi for his guidance throughout this study.

References

[1] M. Gonzalez, J. Govaerts, R. Labie, I. De Wolf, and K. Baert, "Thermo-Mechanical Challenges of Advanced Solar Cell Modules," in *IEEE*, 2011.
[2] S. Dietrich, M. Pander, M. Sander, S. H. Schulze, and M. Ebert, "Mechanical and Thermo-Mechanical Assessment of Encapsulated Solar Cells by Finite-Element-Simulation," in *SPIE*, 2010.
[3] U. Eitner, S. Kajari-Schr¨oder, M. K¨ontges, and H. Altenbach, "Thermal Stress and Strain of Solar Cells in Photovoltaic Modules," in *Advanced Structured Materials*, Berlin: Springer-Verlag, 2011, pp. 453–468.
[4] M. U. Siddiqui and A. F. M. Arif, "Effect of Changing Atmospheric and Operating Conditions on the Thermal Stresses in PV Modules," in *ASME*, 2012.