Jurnal Polimesin
Department of Mechanical Engineering
State Polytechnic of Lhokseumawe
http://e-jurnal.pnl.ac.id/polimesin
e-ISSN : 2549-1999 No. : 1 Month : February
p-ISSN : 1693-5462 Volume : 23 Year : 2025

Processing dates: received on 2024-10-18, reviewed on 2024-11-11, accepted on 2024-11-29 and online availability on 2025-02-28

Open-Source modeling and structural deformation analysis of a solar-powered charging station using FreeCAD™

Kasturi*, Regan Rahadian P, Novan Akhiriyanto, Muhammad Hafiz

Refinery Mechanical Engineering, PEM Akamigas, Cepu 58315, Indonesia
*Corresponding author: kastuiesdm@gmail.com

Abstract

Solar energy is a sustainable and widely available resource with minimal environmental impact and low maintenance requirements. Ensuring the structural integrity of photovoltaic (PV) module supports is crucial for safe and efficient operation. This study utilizes FreeCAD™, an open-source software with Finite Element Analysis (FEA) capabilities, to evaluate the deformation and stress distribution in a solar-powered charging station structure. Two thickness variations (3 mm and 1.5 mm) of hollow carbon steel supports were analyzed under identical loading conditions. Results indicate that reducing the thickness from 3 mm to 1.5 mm increased deformation by 69.81% (from 0.53 mm to 0.9 mm) and stress by 96.93%, both remaining within acceptable structural tolerance limits. These findings validate FreeCAD as a cost-effective alternative to commercial Finite Element Anaysis (FEA) software, leveraging open-source libraries like Python™, Calculix™, and Open Cascade Technology (OCCT) to perform complex simulations without expensive licenses or high-end hardware.

Keywords:
open-source, structure, deformation, finite element analysis, FreeCAD

1 Introduction

Solar energy is one of the most accessible forms of renewable energy, as it primarily requires sunlight and heat from the sun. It is characterized by near-zero pollution and minimal environmental impact, and it is relatively easy to maintain. When installing a solar Photo Voltaic (PV) module, it is crucial to adequately consider the structural safety of both the modules and their support systems. The supports for PV systems are commonly subjected to mechanical pressure loading due to the static weight of the PV panels, which can significantly affect the overall stability of the PV structures.

To ensure that the modules and their supporting structures meet qualification requirements, various mechanical loading tests can be conducted. These tests can help determine the fatigue degradation of the structure through cyclic deformation analysis. Understanding the load capacity of the structure is essential for ensuring its durability and reliability.

While there are multiple software tools available for analyzing structural loads on 3D parametric Com[puter Aided Design (CAD) objects, such as ANSYS™, FreeCAD™ is recognized for its smaller file size and powerful capabilities for conducting specific deformation and stress analyses. Additionally, its open-source nature encourages design and analysis without the financial constraints associated with proprietary software.

FreeCAD™ is a general-purpose Open-Source Software (OSS) directly aimed at mechanical engineering and product design and can be used for other purposes such as architecture, finite element analysis, 3D printing, and other tasks. FreeCAD™ makes heavy use of open-source libraries that exist in scientific computing such as Python™, Calculix™, and Open Cascade Technology (OCCT); the workflow in Fig. 1. [9][10][11]

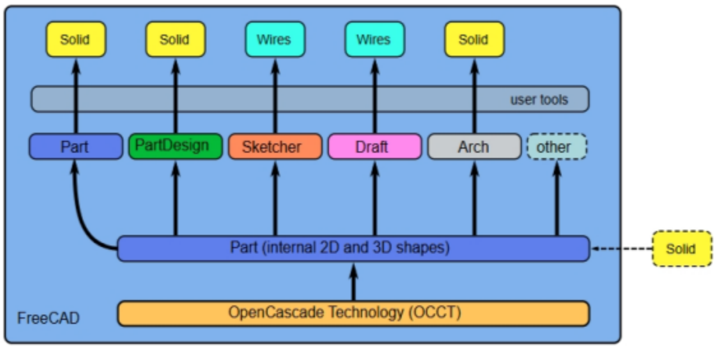

Fig. 1. Freecad Workbenches Workflow

Finite Element Analysis (FEA) is a computational method employed to solve complex mathematical Equations for simulating real-world engineering problems under specific boundary conditions. This analysis yields parameters that can inform the modeling of structural objects. FEA relies heavily on subdividing the model into numerous nodes and elements, with the approximate solution being constructed through weighted residual methods and mathematical techniques.[12][13][14] The FreeCAD™ FEM Workbench is utilized to conduct the FEA based on the provided model. The workflow of this analysis is illustrated in Fig. 2.[15][16]

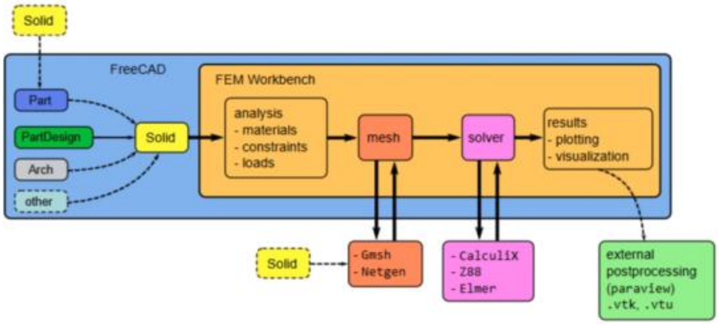

Fig. 2. FEM workbench workflow

FreeCAD™ uses Calculix as the calculation model, which is more accurate in solving complex parametrical FEA calculation than ANSYS and ABAQUS™ on a few applications, this is caused by the material properties is further analyzed inside the calculation.[19]-[29]

A comparative method is used to compare the material specification in different thicknesses, this is done to determine the amount of deformation that can occur in different thicknesses to find the most ideal specification that can withstand the load given by the PV.

Thickness comparison is a method of Least Annual Cost (LAC) to determine the safety of the chosen item while preserving its economic value. Deformation analysis is the study of the safety of a component made by a material based on the amount of deformation caused by a specific load given to the component. By comparing the thickness based on the used material's physical parameters and the load given to the model, the most cost-efficient material thickness can be determined based on the deformation conducted using FEA analysis.

2 Method

Modelling and deformation evaluation. The modelling phase was crucial for defining the dimensions, thickness, and material properties of the structure, as detailed in Subtopic A. Subsequently, the CAD model was analyzed using Calculix™ to assess its safety based on the observed deformations, as discussed in Subtopic B.

2.1 Solar powered charging station modeling

Two different thicknesses were picked to compare the amount of deformation that can occur in the structure. The structure technical drawing in Fig. 3, with a design specification (Table 1).

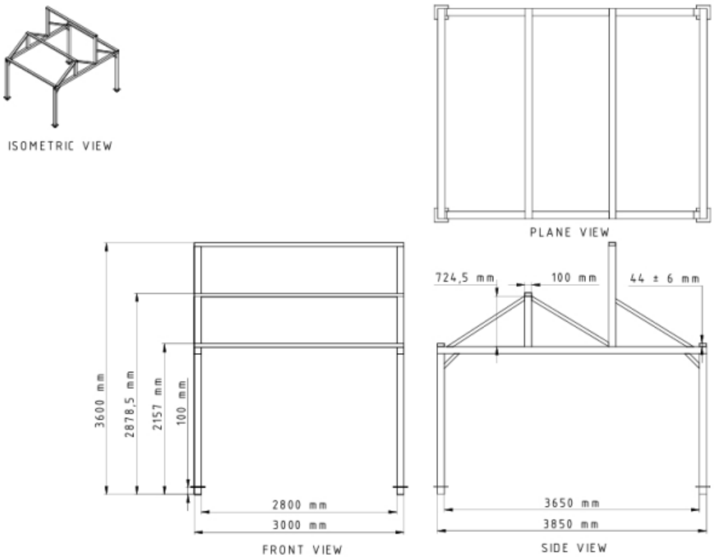

Fig. 3. Charging Station Technical Drawing

Table 1. PV Charging station specification

Material	Dimension
Hollow Steel (CS)	100 × 100 (mm)
Hollow Steel (CS)	100 × 50 (mm)
Thickness (A)	3 (mm)
Thickness (B)	1.5 (mm)

The charging station structure is modelized into a computer-aided design (CAD) model using FreeCAD™ PartDesign Workbench and Draft Workbench based on the technical drawing dimension above. The model in Fig. 4 and Fig. 5.

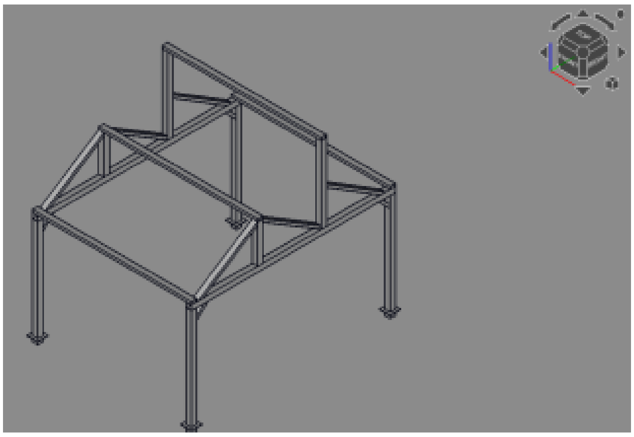
Fig. 4. Charging station structure CAD model

CalculiX™-Steel is used for the material, the material data are based on the standard steel material for CalculiX™ sample calculations 1998 in Table 2. [16]. CalculiX™-Steel is used for the material, the material data are based on the standard steel material for CalculiX™ sample calculations 1998 in Table 2 [16]. Based Table 2. the allowable stress can be determined Eq. (1). [17]

Table 2. CalculiX-Steel Physical parameters

Material Properties	Values (units)
Elastic Modulus	210 (GPa)
Yield Strenght	500 MPa
Density	7900 (kg/m3)
Poisson's ratio	0.3
Thermal Conductivity	43 (W/m/K)
Expansion Coefficient	12 (µm/m/K)
Safety Factor	1.75

pTable 2. CalculiX-Steel physical parameters

Material Properties	Values (units)
Elastic Modulus	210 (GPa)
Yield Strenght	500 MPa
Density	7900 (kg/m3)
Poisson's ratio	0.3
Thermal Conductivity	43 (W/m/K)
Expansion Coefficient	12 (µm/m/K)
Safety Factor	1.75

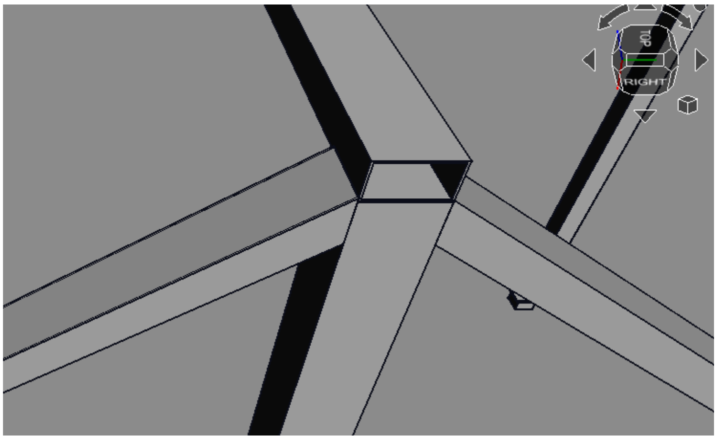
Fig. 5. Hollow steel thickness design

$$Allowable\ Stress = \frac{Yield\ Strenght}{Safety\ Factor} \quad (1)$$

Which resulted in an allowable stress of 285.71 MPa. This result is then compared with the von misses stress result from the FEA to determine if the model is acceptable or not based on the allowable stress that the material can withstand.

2.2 Structure Deformation Analysis

To determine its deformation and displacement, the nodes (initial shape coordinate) must be set as a boundary in which the model can deform, The FEMMeshNetgen™ tool is used as the meshing option of the structure. The numbers of elements in Fig. 6, and Table 3.

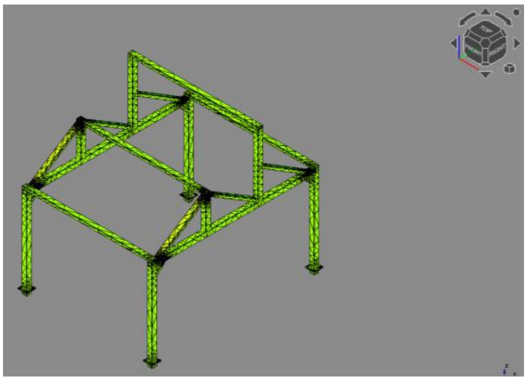
Fig. 6. Charging station structure mesh

The boundary condition is set based on Table 4 and the load condition is set based on Table 5. Both boundary and load conditions in Fig. 7. The boundary condition is taken from the structural fixed foundation which in Fig. 8 and Fig. 9. The load given to the structure is the weight of the PV which in Fig. 10.

Table 3. Netgen Element Result

Fem Mesh	Values (units)
Nodes	146731
Edges	7517
Faces	41812
Polygons	0
Volumes	78821

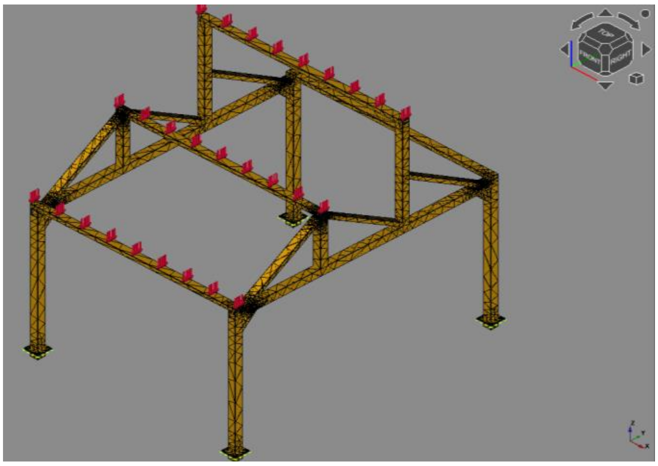

Fig. 7. Boundary and load condition

The boundary condition is set by the given parameter.

1. Fixed support of the Charging Station Structure. The fixed support is set in geometry faces in which the geometry is static (foundation).

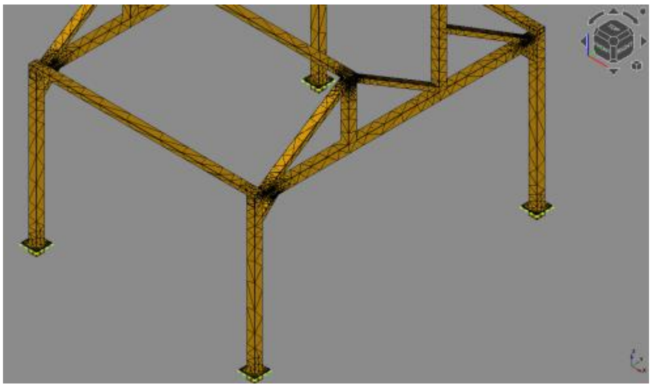

Fig. 8. Structure fixed support

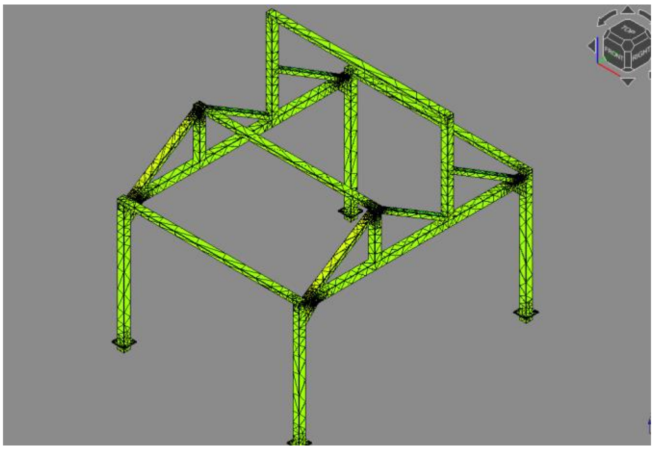

Fig. 9. Structure thermal condition

2. Charging Station Structure Thermal Condition. The structure will be put outside with an ambient temperature of 40° C, this is why the thermal condition is set 40° C. While the load condition is taken from this Eq.
3. Force load given to the structure (N). The force load given to the structure is obtained from the weight of the PV using the Eq. (2).[17], [18]

$$F = ma$$
$$kgf\ (N) = kg\ x\ F \tag{2}$$

The load is given in this face to simulate the load given by the weight of the PV and the railing Equipment (95 kg)

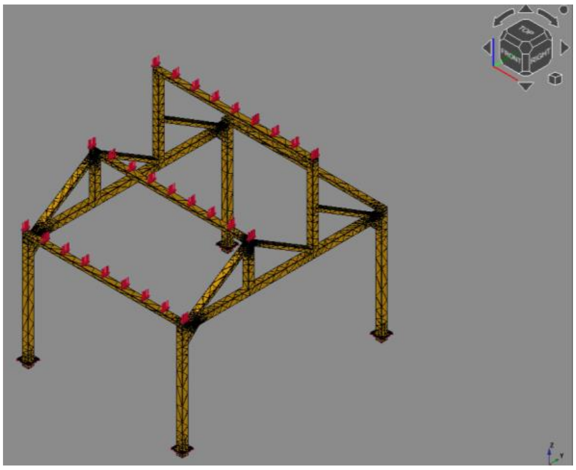

Fig. 10. Structure load given

The boundary condition value and structure load are needed to determine the amount of deformation in the model which value in Table 4. and Table 5.

Table 4. Structure Boundary Condition Given

Boundary Condition	Values (units)
Fixed support	4 (Faces)
Thermal condition	40° (°C)

Table 5. Structure Load Condition Given

Load Condition	Values (units)
Force load (-Y Axis)	931.95 (N)

The boundary and load condition is applied to both structures with different thicknesses, one with the thickness of 3 mm and one with the thickness of 1,5 mm which in Table 6. The deformation of the model based on its thickness can be used as a LAC model to make the model as cost-efficient as possible.

Table 6. Structure Thickness Condition Given

Material	Values (units)
Thickness (A)	3 (mm)
Thickness (B)	1.5 (mm)

The displacement increases can be analyzed using Eq. (3) to represents the percentage difference in deformation relative to the material thickness. Deformation percentage difference Eq. 3,

Fig. 11. CalculiX™ CcX Solver Tab

$$\frac{(Max\ Condition\ High - Max\ Condition\ Low)}{|Max\ Condition\ Low|} \times 100 \tag{3}$$

Based on the boundary condition and load condition given on the structure, the simulation then can be done by choosing the desired analysis settings within the FEM Workbench.

The method chosen in this analysis was the CalculiX™ CcX™ Solver tool, with choosing the elasticity Eq. and the deformation Equation. Fig. 11. Shows the solution tab chosen on the software.

3 Results and discussion

3.1 FEM Workbench simulation result

When running the simulation using FreeCAD™ versus other simulation software, the histogram can be seen which indicate the number of nodes affected by the stress or the deformation on the given magnitude.

After running the FreeCAD™ simulation based on the analysis chosen within the boundary condition and load condition given on the charging station structure, the result in Figs. 12-17.

3.1.1 Total deformation

The total deformation of the charging station structure based on the given parameters in Table 7 and Table 8. With the deformation magnitude in Fig. 13 and Fig. 15. The structure deformation heatmap in Fig. 12 and Fig. 14.

Table 7. Thickness A Total Deformation Result

Deformation	Values (units)
Maximum	0.53 mm
Minimum	0 mm

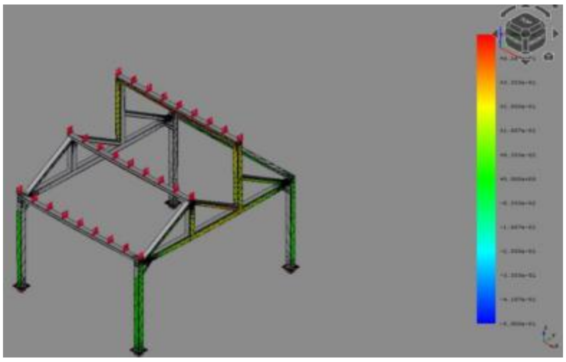

Fig. 12. Structure (A) total deformation

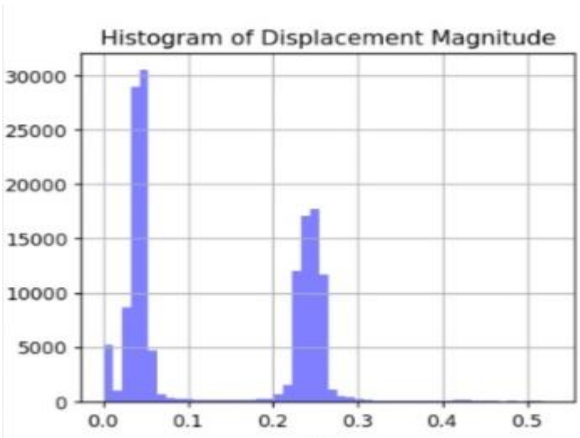

Fig. 13. Deformation Magnitude Histogram

Table 8. Thickness B total deformation Result

Deformation	Values (units)
Maximum	0.9 mm
Minimum	0 mm

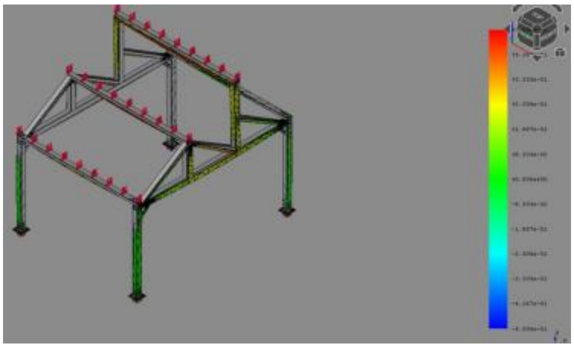

Fig. 14. Structure (B) total deformation

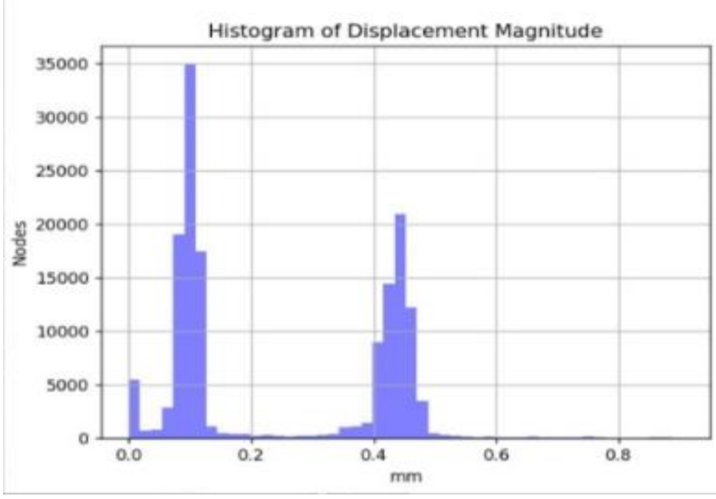

Fig. 15. Deformation magnitude histogram

3.1.2 Equivalent stress (Von Mises stress)

The maximum stress taken by the charging station structure based on the given parameters in Table 9 and Table 10. Green indicator on the structure shows that the structure is safe to use, in Fig. 16 and 17 with a von Mises stress result in Table 9 and Table 10.

Table 9. Structure (A) equivalent stress result

von Mises stress	Values (units)
Maximum	77.31 (MPa)
Minimum	0 (MPa)

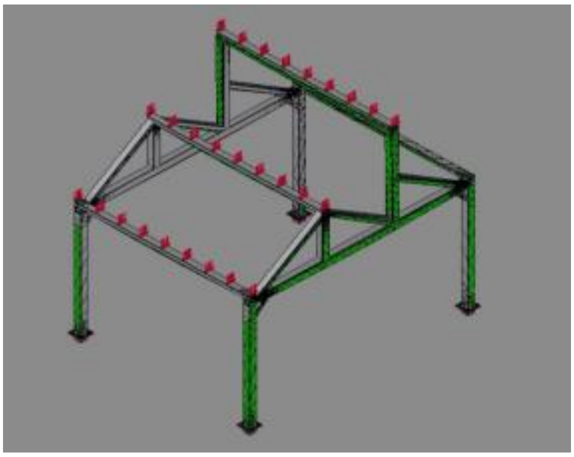

Fig. 16**.** Structure (A) Equivalent Stress

Table 10. Structure (B) equivalent stress result

Von Mises stress	Values (units)
Maximum	152.68 (MPa)
Minimum	0 (MPa)

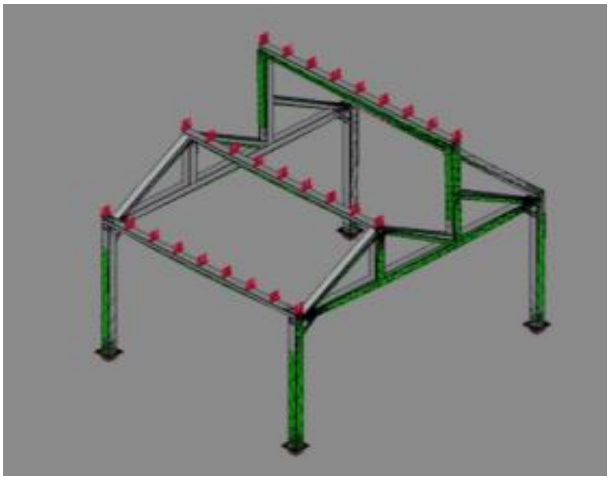

Fig. 17. Structure (B) **e**quivalent stress

3.2 Displacement Analyzation Result

Based on the result above, the maximum deformation in each thickness of the charging station structure with the given load and boundary condition on Table 11 and the Equivalent stress (Von Mises stress) in Table 12.

Table 11. Structure Total Deformation

Max Deformation	Values (units)
Structure (A)	0.53 mm
Structure (B)	0.9 mm

Table 12. Structure Equivalent Stress

Max von Mises stress	Values (units)	Maximum allowable stress
Structure (A)	77.31 (MPa)	285.71 (MPa)
Structure (B)	152.68 (MPa)	

The increase in deformation can be calculated using Eq. (3), which represents the percentage difference in deformation relative to the material thickness.

Based on the equations. above, reducing the thickness from Structure A, which has a thickness of 3 mm, to Structure B, which has a thickness of 1.5 mm, resulted in a 69.81% increase in total deformation. Additionally, there was a 96.93% increase in the stress experienced by the structure.

Both structures are deemed safe for use. This conclusion is based on the maximum allowable stress that steel can withstand before sustaining damage. Given a yield strength of 500 MPa and a safety factor of 1.75, the allowable stress is calculated to be 285.71 MPa. This value is lower than the maximum von Mises stress for Structure A (77.31 MPa) and Structure B (152.68 MPa). Therefore, both structures are within the allowable limits, indicating sufficient structural integrity under the applied loading conditions.

While the stress results indicate that the deformed structures are suitable for further use, this analysis does not account for other factors such as wear and deformation due to corrosion.

Based on the result above, Structure B is deemed fit as the model thickness in a real-world application, this result indicates the material can achieve optimal performance under the given load and boundary conditions while also staying cost-efficient, helping to determine the LAC, ensuring both cost-effective while also reliable in real-world application.

4 Conclusion

This study modelled and analysed the structural deformation of a solar-powered charging station using FreeCAD's™ FEA tools. Simulation showed that reducing material thickness from 3 mm to 1.5 mm resulted in a 69.81% increase in deformation and a 96.93% increase in stress. Despite these changes, both configurations remained within acceptable stress tolerance limits, confirming the feasibility of using the thinner material under controlled conditions. Commercial software like ANSYS offers similar structural analysis capabilities, meanwhile, FreeCAD™ presents a viable open-source alternative, providing powerful FEA functionality through Python, Calculix™, and OCCT. Its cost-effectiveness, accessibility, and low hardware requirements make it a practical solution for structural analysis and deformation modelling.

References

[1] R. A. Rachmanto, F. J. Regannanta, Ubaidillah, Z. Arifin, D. Widhiyanuriyawan, E. Yohana, and S. D. Prasetyo, "Analysis development of public electric vehicle charging stations using on-grid solar power plants in Indonesia," *Int. J. Transp. Dev. Integr.*, vol. 7, no. 3, pp. 215–222, 2023. doi: 10.18280/ijtdi.070305.

[2] S. D. Prasetyo, A. R. Prabowo, and Z. Arifin, "The use of a hybrid photovoltaic/thermal (PV/T) collector system as a sustainable energy-harvest instrument in urban technology," Heliyon, vol. 9, no. 2, p. e13390, 2023. doi: 10.1016/j.heliyon.2023.e13390.

[3] L. Suman, K. Sadhu, P. Kumar, R. S. Dhar, R. Dey, S. B. Naskar, A. Ganguly, and A. Kumar, "Analysis of mechanical stress and structural deformation on a solar photovoltaic panel through various wind loads," Microsystem Technologies, vol. 27, 2021.

[4] Y. Adedĳi, "Review of Analysis of Structural Deformation of Solar Photovoltaic System under Wind-Wave Load," 2022. doi: 10.31224/2273.

[5] R. Arndt and R. Puto, "Basic Understanding of IEC Standard Testing For Photovoltaic Panels," TUV SUD Product Service, Peabody, MA, 2010.

[6] N. Bosco, "Evaluation of Dynamic Mechanical Loading as an accelerated test method for ribbon fatigue," IEEE 39th Photovoltaic Specialists Conference (PVSC), pp. 3173–3178, 2013.

[7] R. G. Budynas, J. K. Nisbett, and McGraw-Hill, Shigley's Mechanical Engineering Design, 8th ed. New York, 2008.

[8] R. C. Juvinall and K. M. Marshek, Fundamentals of Machine Component Design, 4th ed., John Wiley & Sons, Inc., New York, NY, 2006.

[9] I.-C. Jong and W. Springer, "Teaching von Mises stress: From principal axes to non-principal axes," American Society for Engineering Education, 2009.

[10] J. Riegel, W. Mayer, and Y. van Havre, Freecad, version 0.21. [Online]. Available: https://www.freecad.org/

[11] D. Di Donato and M. Abita, "Low-cost 4D BIM modeling: A comparison between FreeCAD and commercial software," *ISPRS - International Archives of the Photogrammetry, Remote Sensing and Spatial Information Sciences*, vol. XLII-2/W17, pp. 107–114, 2019. doi: 10.5194/isprs-archives-XLII-2-W17-107-2019.

[12] F. Machado, N. Malpica, and S. Borromeo, "Parametric CAD modeling for open-source scientific hardware: Comparing OpenSCAD and FreeCAD Python scripts," PLoS ONE, vol. 14, p. e0225795, Dec. 2019.

[13] S. N. Ain, R. Nadler, R. A. Ilyas, A. M. Radzi, D. Sivakumar, "The Effect of Fiber Length on Mechanical and Thermal Properties of Roselle Fiber-Reinforced Polylactic Acid Composites via ANSYS Software Analysis," in Roselle, S. M. Sapuan, R. Nadler, A. M. Radzi, and R. A. Ilyas, Eds., Academic Press, 2021, pp. 215-235, ISBN: 9780323852135, doi: 10.1016/B978-0-323-85213-5.00012-3.

[14] F. Sarghini, F. Erdogdu, and R. Accorsi, "Designing advanced food packaging systems and technologies through modeling and virtualization," in Sustainable Food Supply Chains: Planning, Design, and Control through Interdisciplinary Methodologies, R. Accorsi and R. Manzini, Eds. Academic Press, 2019, pp. 83-104. doi: 10.1016/B978-0-12-813411-5.00006-5.

[15] V. Jagota, A. Sethi, and K. Dr.-Khushmeet Kumar, "Finite Element Method: An Overview," Walailak Journal of Science & Technology, vol. 10, pp. 1–8, 2013. doi: 10.2004/wjst.v10i1.499.

[16] J. Riegel, W. Mayer, and Y. van Havre, "FEM Workbench - FreeCAD Documentation." [Online]. Available: https://wiki.freecad.org/FEM_Workbench

[17] Y. A. Çengel and A. J. Ghajar, Heat and Mass Transfer: Fundamentals and Applications, 5th ed. New York, NY, USA: McGraw-Hill Professional, 2014, pp. 1–10.

[18] A. Thompson and B. Taylor, "Guide for the Use of the International System of Units (SI), Special Publication (NIST SP)," National Institute of Standards and Technology, Gaithersburg, MD, 2008. [Online]. Available: https://doi.org/10.6028/NIST.SP.811e2008.

[19] T.-Q. K. Lam, T.-M. D. Do, and B. Trương Văn, "Structural analysis of continuous beam using FEM and ANSYS," Journal of Mechanical Construction, vol. 11, pp. 60-65, 2021. doi: 10.54772/jomc.v11i02.291.

[20] Y. Liu, "ANSYS and LS-DYNA used for structural analysis," International Journal of Computer Aided Engineering and Technology, vol. 1, 2008. doi: 10.1504/IJCAET.2008.021254.

[21] K. Kono, M. Hayashi, T. Matsunaga, and H. Okuda, "Extension of analysis support with FreeCAD workbench FEM_FrontISTR," The Proceedings of The Computational Mechanics Conference, 2023, pp. OS-0602. doi: 10.1299/jsmecmd.2023.36.OS-0602.

[22] D. Di Donato and M. Abita, "Low-cost 4D BIM modeling: A comparison between FreeCAD and commercial software," ISPRS - International Archives of the Photogrammetry, Remote Sensing and Spatial Information *Sciences*, vol. XLII-2/W17, pp. 107–114, 2019. doi: 10.5194/isprs-archives-XLII-2-W17-107-2019.

[23] D. Winarski, K. Nygren, and T. Winarski, "Finite Element Analysis versus Empirical Modal Analysis of a Basketball Rim and Backboard," Vibration, vol. 7, no. 2, pp. 582–594, 2024. doi: 10.3390/vibration7020030.

[24] I. Sh. Nasibullayev, "Application of free software FreeFem++/Gmsh and FreeCAD/CalculiX for simulation of static elasticity problems," Multiphase Systems, vol. 15, pp. 183–200, 2020. doi: 10.21662/mfs2020.3.129.

[25] J.-Y. Li, L. Gu, H.-S. Xu, Y. Dai, Y. Zhang, R. Yu, L. Zhang, D. Wang, and W. Jiang, "FreeCAD based Monte Carlo modeling approach for fusion reactor facilities," Fusion Engineering and Design, vol. 155, p. 111711, 2020. doi: 10.1016/j.fusengdes.2020.111711.

[26] J. Jorgensen, M. Hodkiewicz, E. Cripps, and G. M. Hassan, "An open-source analysis workflow for geometrically imperfect bolted ring-flanges in wind turbine support structures," Journal of Physics: Conference Series, vol. 2767, 2024. doi: 10.1088/1742-6596/2767/6/062004.

[27] H. Wang, W. Lu, R. Shi, S. Yu, and F. Wu, "Research and Development of Railway Alignment Design System Using FreeCAD Software," IOP Conference Series: Earth and Environmental Science, vol. 719, p. 032009, 2021. doi: 10.1088/1755-1315/719/3/032009.

[28] F. Diara and F. Rinaudo, "Towards FreeCAD experimentation and validation as a FOS HBIM platform for building archaeology purposes," 2021.

[29] W. Finnegan, Y. Jiang, N. Dumergue, P. Davies, and J. Goggins, "Investigation and Validation of Numerical Models for Composite Wind Turbine Blades," J. Mar. Sci. Eng. vol. 9, p. 525, 2021. doi: 10.3390/jmse9050525.