The International Journal of Engineering and Science (IJES)
|| Volume || 12 || Issue || 9 || Pages || PP 35-43 || 2023 ||
ISSN (e): 2319 – 1813 ISSN (p): 2319 – 1805

Optimization design study on a prototype Simple Solar Panel Bracket

Shui-Sheng Xu[1,*], Ke-Xin Xie[2], Shi-Peng Li[2]
1 TONKING NEW ENERGY TECHNOLOGY (JIANGSHAN) CO., LTD., Quzhou, 324100, China;
2 College of Mechanical Engineering, Quzhou University, Quzhou, 324000, China;

Abstract: *In order to improve the overall performance of solar panel brackets, this article designs a simple solar panel bracket and conducts research on it. This article uses Ansys Workbench software to conduct finite element analysis on the bracket, and uses response surface method to optimize the design of the angle iron structure that makes up the bracket. The overall model of the bracket before and after optimization is analyzed and compared. The optimized angle iron section adopts the section height of 32mm, the section width of 21.6mm, and the section thickness of 2mm. Compared with the original stent, the weight of the optimized stent was reduced by 0.4365kg, and the weight loss rate reached 11.02%. At the same time, the maximum displacement of the optimized bracket is reduced by 0.0531mm and the maximum stress is also reduced by 1.587MPa. This indicates that the solar panel bracket enhances the overall performance of the bracket while achieving lightweight.*

Keywords: *Solar panel bracket; Ansys workbench; Finite element analysis; Response surface; Multi-objective optimization*

Date of Submission: 25-09-2023 Date of acceptance: 12-10-2023

I. Introduction

The widespread use of solar panels provides clean and renewable energy for social development, becoming a key component in promoting sustainable development[1, 2]. The bracket needs to bear the weight of the solar panel and ensure its stability. If the bracket structure is not strong enough, the solar panel may deform or break, which not only affects power generation efficiency but also may cause equipment damage. Therefore, studying the strength of solar panel bracket structures is crucial for improving the reliability and safety of solar systems.

Jiang et al. conducted analysis and research on the structural design of photovoltaic bracket foundations built on landfill sites, analyzing the advantages and disadvantages of different foundation forms[3]. Yin took a certain buckle type full hall support as the research object, and used the finite element method to analyze the stiffness and strength of various parts of the support system. He also analyzed the wind resistance stability of the support, and the calculation results met the requirements[4]. Liu et al. conducted structural static simulation analysis on the sangxia solar bracket structure and conducted modal analysis based on the actual working conditions of the bracket, so as to analyze the overall performance of the solar bracket[5]. Zhu et al. used Abaqus software to conduct research and analysis on solar panel brackets, and through mechanical simulation and structural optimization, they improved the strength and stiffness of the solar panel bracket while reducing its weight[6].

As a clean and renewable energy source, the application range of solar panels is continuously expanding[7, 8]. With the continuous development of solar panels, solar panel bracket are also facing increasingly high requirements. The solar panel bracket needs to bear the weight of the solar panel, and its strength structure needs to ensure that the solar panel will not deform or damage[9, 10]. Based on this, this article conducts research on solar panel bracket, and the analysis results can provide reference basis for the design of subsequent solar panel bracket.

II. Bracket model and calculation method

2.1 Bracket model

The newly designed solar panel bracket in this article has a length of 508mm, a width of 574mm, and a height of 418mm. All parts of the solar panel bracket are connected by angle iron. Simplify the process holes and small rounded corners on the solar panel bracket, and the simplified three-dimensional model of the solar panel bracket is shown in Fig. 1

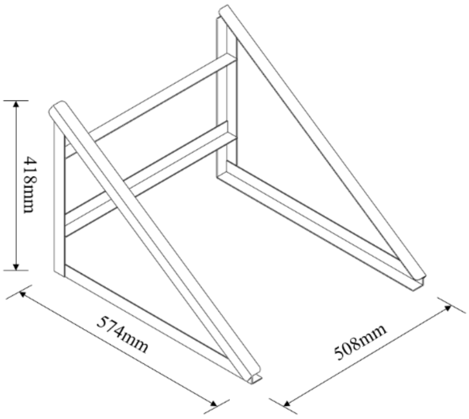


Fig. 1 3D model of solar panel bracket

2.2 Boundary conditions

This article uses Ansys Workbench for simulation analysis of solar panel bracket. In order to obtain more accurate simulation data, this article uses a combination of triangular and quadrilateral elements to mesh the bracket, and sets the size of the mesh element to 1mm, dividing it into a total of 616887 elements and 1615166 nodes. The solar panel bracket is made of Q235 carbon structural steel, whose elastic modulus is 210GPa, poisson ratio is 0.3, and mass density is 7850kg/m^3. In order to simplify the calculation, the solar panel is applied to the corresponding part of the bracket in the form of gravity load, and a fixed constraint is set at the bottom of the bracket. The weight of the solar panel is 152N. The boundary conditions of the solar panel bracket are shown in Fig. 2.

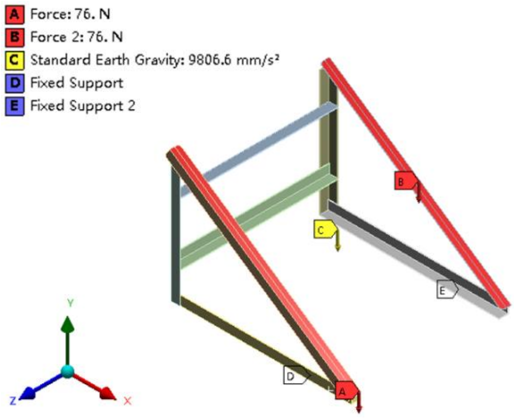


Fig. 2 Boundary conditions of the solar panel bracket

In order to more intuitively reflect the deformation of the support beam, this article adds monitoring paths (1: starting point, 2: ending point) at the centerline of the upper surface of the two support beams. The schematic diagram of the monitoring path is shown in Fig. 3.

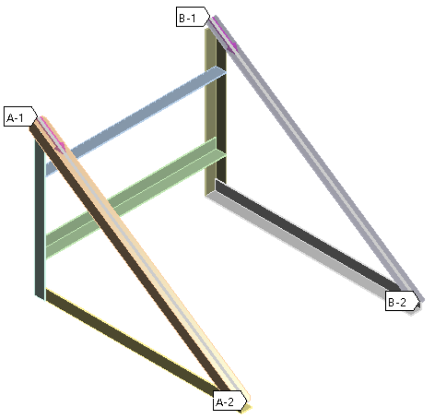


Fig. 3 Monitoring path of support beam

2.3 Static analysis results

The displacement calculation results of the solar panel bracket are shown in Fig. 4. The maximum displacement of the bracket occurs in the middle of the two support beams, with a maximum displacement of 0.2227mm. Meanwhile, the displacement change at the middle of the right support beam is slightly greater than that at the middle of the left support beam. The maximum displacement of the support beam of the solar panel bracket is less than 0.25mm, and the overall displacement of other components is less than 0.1mm, which can meet the strength design requirements of the bracket.

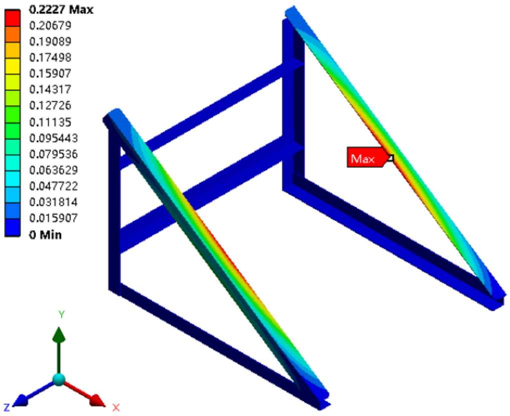


Fig. 4 Displacement diagram of the bracket

In Fig. 5, starting from the upper end of the support beams on both sides (A-1 and B-1), the displacement of the left and right support beams gradually increases from 0.0164mm and 0.0166mm at the upper end, respectively. The maximum displacement reached at 50% of the middle of the support beam is 0.1434mm and 0.1436mm respectively. Subsequently, the displacement continued to decrease to 0.0041mm and 0.0042mm at the lower end of the support beam. This is because the fixed constraints of the left and right support beams are at the upper and lower ends, so under the gravity load of the solar panel, the maximum displacement position appears in the middle of the support beam, which is consistent with the actual situation. In addition, the displacement trend of the left and right support beams is roughly the same, but the overall displacement of the right support beam is slightly greater than that of the left support beam. The displacement of the support beams on both sides is very small, which can meet the strength requirements of the solar panel bracket.

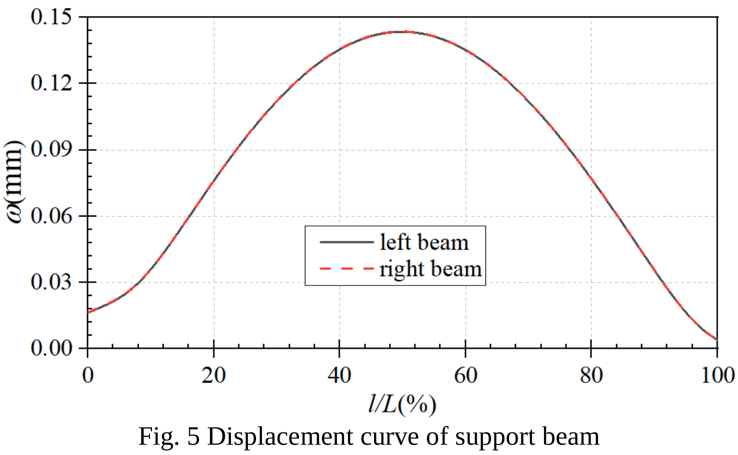


Fig. 5 Displacement curve of support beam

The stress calculation results of the solar panel bracket are shown in Fig. 6. The maximum stress of the bracket occurs at the position where the upper end of the left support beam contacts the fixed beam, with a maximum stress value of 38.519MPa. The local stress of the bracket is shown in Fig. 7. At the same time, based on the mechanical performance parameters of the support material, it can be obtained that the maximum stress value of the support is far less than the yield strength limit of the material used.

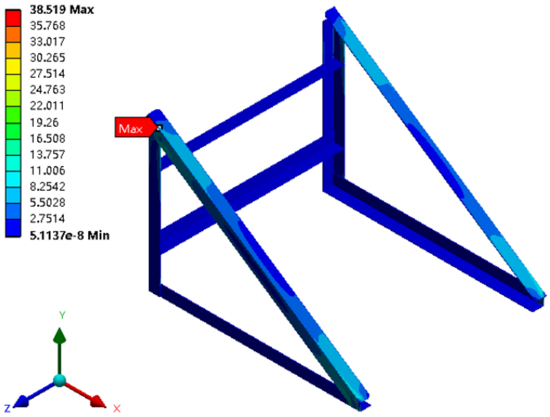


Fig. 6 Stress diagram of the bracket

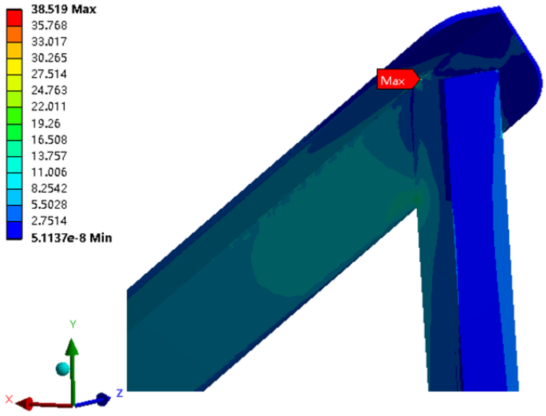


Fig. 7 Local stress diagram of the bracket

In Fig. 8, starting from the upper ends of the support beams on both sides (A-1 and B-1), the stress values of the support beams on both sides gradually increase from 1.0206MPa and 1.0249MPa at the upper end to 7.130MPa and 7.135MPa at a support beam ratio of 14.93%, respectively. As the distance increases, the stress values of the left and right support beams gradually decrease to 1.6417MPa and 1.6387MPa at 51.74%, and then increase again to 7.506MPa and 7.503MPa at 90.05%. The stress variation trend of the support beams on both sides remains consistent, and the difference in stress variation between the two is very small. From this, it can be seen that the stress values of the support beams on both sides of the bracket are relatively small, and there will be no strength damage, which can meet the strength design requirements of the solar panel bracket.

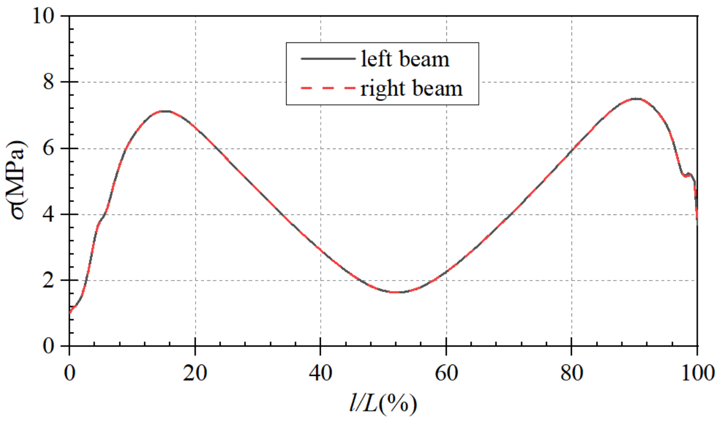


Fig. 8 Stress curve of support beam

III. Optimization Design Based on Response Surface Methodology

In order to further improve the performance of the bracket, the bracket is lightweight while ensuring the strength of the solar panel bracket. Considering that the cross-sectional shape of the angle iron used for making the bracket is the same, this article uses Ansys Workbench's Response Surface Optimization to optimize the cross-sectional shape of all components of the bracket.

3.1 Optimizing mathematical models

The aim of this study is to optimize the cross-sectional shape of the angle iron in the solar panel bracket, so as to achieve optimal performance while meeting strength requirements. This optimization aims to optimize the quality, maximum displacement, and maximum stress of the support. The optimized mathematical model is shown below.

$$\min F(x) = \{F_1(x), F_2(x), F_3(x)\}^T \tag{1}$$

$$\begin{cases} F_1(x) = \min(m_{\max}) \\ F_2(x) = \min(\omega_{\max}) \\ F_3(x) = \min(\sigma_{\max}) \end{cases} \tag{2}$$

Where: $m_{\max}$ is the mass of the bracket, kg; $\omega_{\max}$ is the maximum displacement of the bracket, mm; $\sigma_{\max}$ is the maximum stress of the bracket, MPa.

Using response surface analysis method for multi-objective optimization[11, 12], three parameters of angle iron section height *h*, section width *b*, and section thickness *t* were used as design variables. The range of design variables is shown in Tab. 1, and the shape of angle iron section is shown in Fig. 9.

Tab. 1 Value range of design variables

Design variable	Original value	Value range
Section height *h*/mm	30	[20, 40]
Section width *b*/mm	30	[20, 40]
section thickness *t*/mm	2	[1, 3]

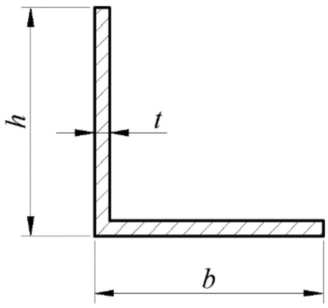


Fig. 9 Cross section shape of angle iron

In summary, the optimization model for establishing multi-objective optimization problems of angle iron cross-section is as follows:

$$\begin{cases} \min F(x) = \left\{F_1(x), F_2(x), F_3(x)\right\}^T \\ s.t. \begin{cases} 100 \le h \le 120 \\ 35 \le b \le 45 \\ 1.5 \le t \le 2.5 \end{cases} \\ X = \left[h, b, t\right]^T \end{cases} \quad (3)$$

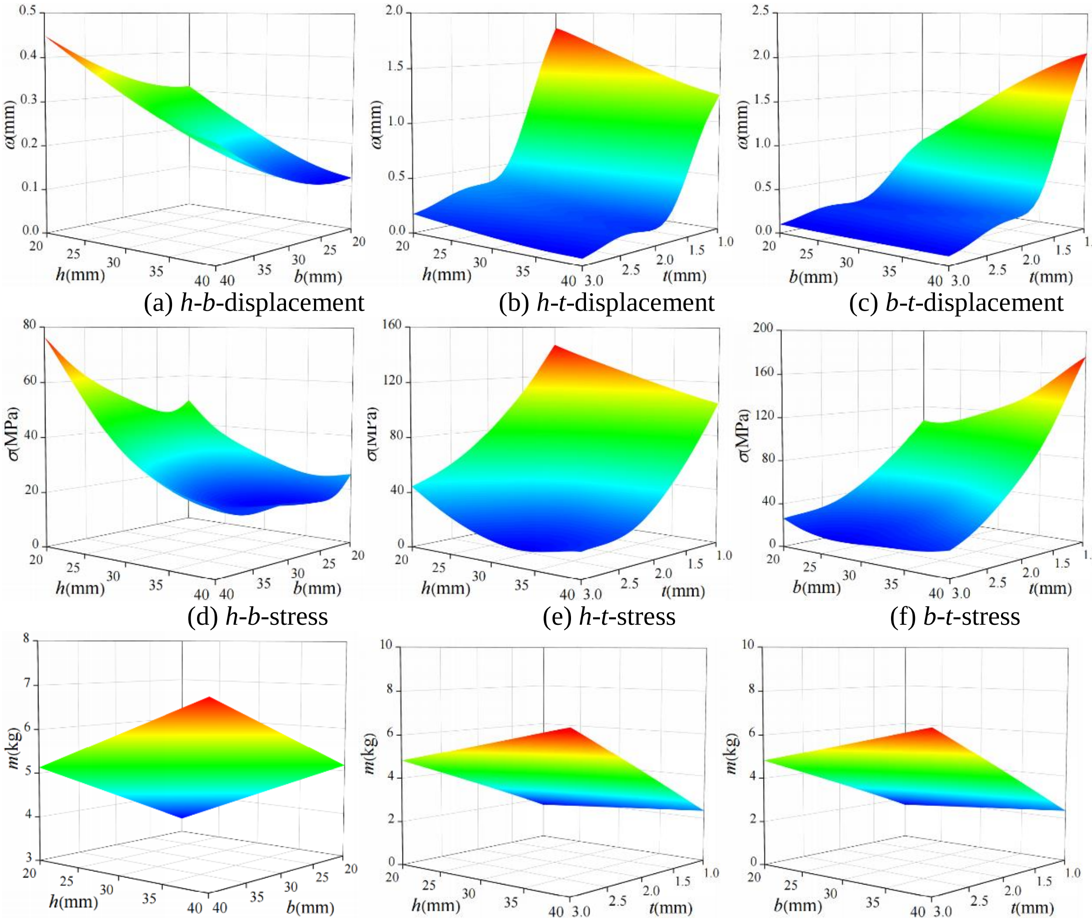
3.2 Optimization methods

The Box-Behnken method was selected for this optimization, and 13 sets of design points were generated based on the optimization method. The calculation was carried out with constraints. The completed data is shown in Tab. 2.

Tab. 2 Calculation results of optimization design points

Order	Height *h*/mm	Width *b*/mm	Thickness *t*/mm	Mass m_{max}/kg	Maximum displacement ω_{max}/mm	Maximum stress σ_{max}/MPa
1	30	30	2	3.9615	0.2227	38.519
2	20	20	2	2.5939	0.3201	67.915
3	40	20	2	3.9579	0.1302	41.646
4	20	40	2	3.9579	0.4415	115.860
5	40	40	2	5.3255	0.2373	42.716
6	20	30	1	1.6721	1.8578	142.750
7	40	30	1	2.3558	1.2511	103.430
8	20	30	3	4.8093	0.1747	43.704
9	40	30	3	6.8629	0.0680	22.711
10	30	20	1	1.6721	0.8219	106.160
11	30	40	1	2.3541	2.0659	180.980
12	30	20	3	4.8116	0.0982	26.520
13	30	40	3	6.8629	0.1139	21.254

This study uses the Genetic Aggregation algorithm to fit the response surface, and uses the MOGA (multi-objective genetic) optimization algorithm to solve the optimal parameters of the angle iron cross-section. During the execution of genetic algorithms, the maximum allowed Pareto percentage is set to 70%. The initial total number of samples is 3000, and three optimal design points are selected from the optimization results. Fig. 10 shows the optimized response surface of the angle iron cross-section, while Tab. 3 lists the parameters of the candidate optimal design points.



(a) *h*-*b*-displacement (b) *h*-*t*-displacement (c) *b*-*t*-displacement
(d) *h*-*b*-stress (e) *h*-*t*-stress (f) *b*-*t*-stress
(g) *h*-*b*-mass (h) *h*-*t*-mass (i) *b*-*t*-mass

Fig. 10 Response surface

Tab. 3 Candidate Optimal Design Points

Oeder	Height *h*/mm	Width *b*/mm	Thickness *t*/mm	Mass m_{max}/kg	Maximum displacement ω_{max}/mm	Maximum stress σ_{max}/MPa
1	32.180	21.542	2.0229	3.5704	0.16737	37.780
2	31.993	21.520	2.0352	3.5768	0.16904	37.478
3	31.806	21.692	2.0335	3.5729	0.17077	37.477

By comparing the three candidate points mentioned above, the optimized dimensions are rounded and the optimized parameters are obtained as follows: the height *h* of the angle iron section is 32mm, the width *b* is 21.6mm, and the thickness *t* is 2mm. The optimized bracket mass is reduced by 35.8kg compared to before optimization. The parameter change table is shown in Tab. 4.

Tab. 4 Parameter changes before and after optimization

Name	Height *h*/mm	Width *b*/mm	Thickness *t*/mm	Mass *m*/kg
Original	30	30	2	3.9615
Optimized	32	21.6	2	3.5250
Difference value	+2	-8.4	0	0.4365

3.3 Optimization analysis results

Modify the cross-sectional structure of the angle iron according to the rounded dimensions in Tab. 4, and perform numerical simulations again in Ansys Workbench software. The numerical simulation results of the optimized bracket are shown in Fig. 11 and Fig. 12. The weight of the optimized solar panel bracket is 3.525kg, which is 0.4365kg less than before, and the weight reduction rate is 11.02%. Meanwhile, the maximum displacement of the optimized bracket is 0.1696mm, which is 0.0531mm less than the original bracket. The maximum stress of the optimized bracket is 36.932MPa, which is 1.5870MPa less than the original bracket. While achieving lightweight of the bracket, it further improves the overall performance of the bracket.

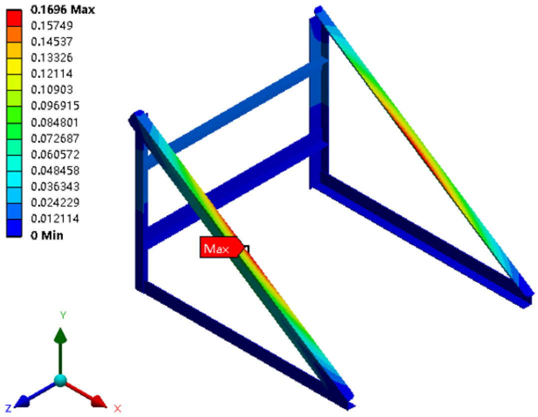


Fig. 11 Displacement diagram of the optimized bracket

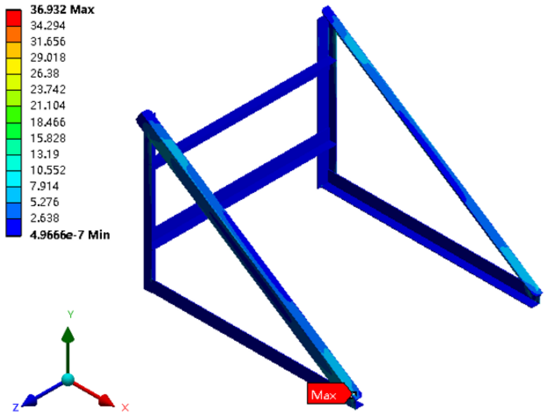


Fig. 12 Stress diagram of the optimized bracket

From Fig. 13, it can be seen that the maximum displacement of the optimized support beam is smaller than the maximum displacement before optimization, and the trend of displacement change of the support beam before and after optimization is roughly the same. The maximum displacement of the left support beam decreased from 0.1434mm before optimization to 0.1324mm after optimization, with a displacement change of 0.0110mm. The maximum displacement of the right support beam decreased from 0.1436mm before optimization to 0.1323mm after optimization, with a displacement change of 0.0113mm. This means that the maximum displacement of the front and rear crossbeams after optimization decreased by 7.67% and 7.87%, respectively.

From Fig. 14, it can be seen that the overall stress of the optimized support beam is smaller than that before optimization, and the trend of stress changes in the support beam before and after optimization is roughly the same. The maximum stress of the left support beam decreased from 7.506MPa before optimization to 6.279MPa after optimization, with a stress change of 1.227MPa. The maximum stress of the right support beam decreased from 7.503MPa before optimization to 6.273MPa after optimization, with a stress change of 1.23MPa.

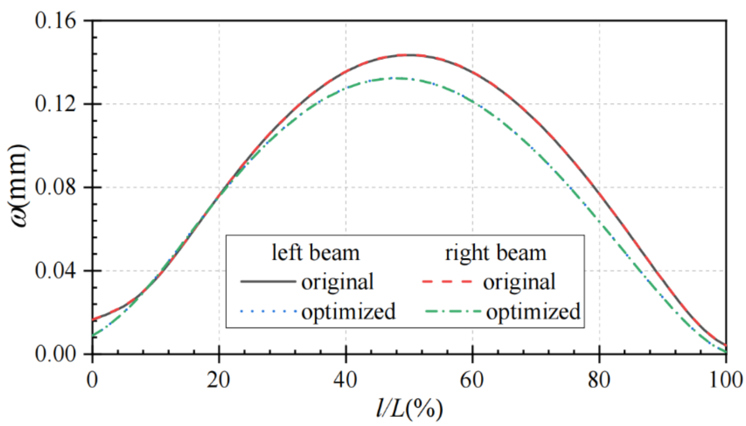


Fig. 13 Displacement comparison curve of support beam

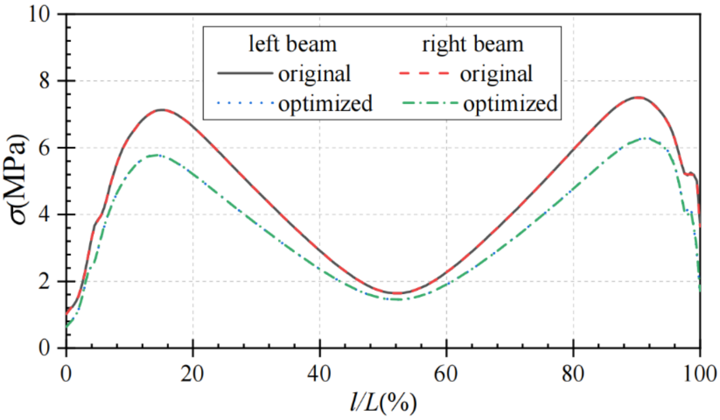


Fig. 14 Stress comparison curve of support beam

Based on the above content, it can be concluded that the overall displacement and stress changes of the optimized bracket are very small. While achieving lightweight bracket, the overall performance of the bracket has been improved. The comparison of performance parameters of the bracket before and after optimization is shown in Tab. 5.

Tab. 5 Performance comparison of bracket before and after optimization

Name	Original bracket	Simplified bracket	Optimized bracket
m_{max}/kg	3.9615	3.5250	-0.4365
ω_{max}/mm	0.2227	0.1696	-0.0531
σ_{max}/MPa	38.519	36.932	-1.587

IV. Conclusion

As a key load-bearing component of solar panels, the strength research of brackets is crucial for ensuring the safe and stable operation of solar equipment. This article conducts numerical simulation on the solar panel bracket and optimizes the design of the angle iron structure that forms the bracket based on the simulation analysis results. The main conclusions are as follows:

(1) From the displacement and stress cloud maps of the finite element analysis results, it can be seen that the maximum displacement of the bracket under stress does not exceed 0.25mm, and the maximum stress does not exceed 40.0MPa. From this, it can be concluded that the solar panel bracket can meet the strength design requirements and has the possibility of further lightweight. This article can provide lightweight design for the bracket while ensuring its overall performance and structural strength.

(2) This article uses multi-objective genetic algorithm to optimize the design of the bracket angle iron structure. The optimized angle iron has a cross-sectional height of 32mm, a cross-sectional thickness of 2mm, and a cross-sectional width of 21.6mm. After optimization, the overall mass of the stent was 3.5250kg, which decreased by 0.4365kg compared to before optimization, with a weight reduction rate of 11.02%. At the same time, the maximum displacement and stress of the newly designed bracket have been reduced. The maximum displacement of the optimized bracket decreased by 0.0531mm compared to before optimization, and the maximum stress decreased by 1.587MPa compared to before optimization. While reducing the weight of the bracket, it improves the overall performance of the bracket.

(3) This article monitors the displacement and stress of the two support beams of the bracket, which can clearly reflect the structural performance of the support beams. The analysis results will provide valuable reference for the design of future solar panel brackets.

References:

[1]. Zhong Yajuan, Zhao Dale, Zhao Ying. Research on Quantity Proportion of Different PV Bracket Forms in PV Power Station[J]. Solar Energy, 2022, (09): 90-96.
[2]. Chen Chen, Fang Hai, Chen Xiangxi, et al. Research Rrogress and Application of Solar Photovoltaic Support Structure[J]. Journal of Nanjing Tech University(Natural Science Edition), 2022, 44(04): 365-372.
[3]. Jiang Saixiong, Zhao Tingting, Yan Xu. Research on Design of PV Support Foundation in Landfill Site[J]. Electric Power Survey & Design, 2022, (04): 65-70, 88.
[4]. Yin Dezhi. Layout Optimization and Structural Calculation Analysis of Full Support Used in Cast-in-Place Box Girder Construction[J]. Low Temperature Architecture Technology, 2021, 43(07): 100-103.
[5]. Liu Yingyu, Xiao Hong, Zheng Fanghui, et al. Condenser Bracket Structure Analysis of Trough Type Solar Energy Based on ANSYS[J]. Machinery & Electronics, 2013, (09):37-40.

[6]. Zhu Xiangnan, Lu Fengxia, Song Ziling. Mechanics Analysis and Optimization Research of Solar Water Heater Bracket[J]. Mechanical Engineer, 2012, (05): 71-73.

[7]. Xian Yuxuan, Li Penghui, Yang Kailin, et al. Analysis of Influence Parameters of Purlin Bearing Capacity of Solar Photovoltaic Fixed Bracket[J]. Low Temperature Architecture Technology, 2023, 45(03): 63-66.

[8]. Zou Han, Shao Ligang, Qin Honggang, et al. Power Generation Performance of Efficient PERC Components with Different Bracket Forms[J]. Transactions of the Chinese Society of Agricultural Engineering, 2020, 36(05): 220-225.

[9]. Luo Yuefei, Wu Qi. Analysis of the Support Structure of Parabolic Trough Solar System Using Fluid Structure Interaction[J]. Sci-tech Innovation and Productivity, 2019, (08): 56-59.

[10]. Bao Hongxing, Wang Xudong, Wei Ziling, et al. Research and Application of Structure Design of the New Photovoltaic Square Bracket[J]. Journal of Baotou Vocational & Technical College, 2020, 21(04): 1-5, 13.

[11]. Zhan Yulin, Xu Jianghui, Xu Jun, et al. Optimization of Bridge High Temporary Lifting Support Based on Response Surface Method and Particle Swarm Algorithm[J]. China Railway Science, 2022, 43(06): 39-46.

[12]. Wang Yugang, Xiu Shichao. Multi-Objective Optimization Design of Motor Bracket Using Response Surface Method[J]. Machinery Design & Manufacture, 2021, (10): 42-44.