Optimization design of tension machine frame based on Solidworks and ANYSY Workbench

Abstract: The parameterized model of the tension machine frame is established through Solidworks software, and then the optimization function of ANSYS Workbench is used to optimize the design of the frame. Under the condition of meeting the strength, the weight reduction design of the frame is completed. Compared with the original design scheme, the maximum equivalent stress and maximum overall shape of the optimized scheme will be lower, the weight will be reduced by 7.11%, and the weight reduction effect will be significant, thus reducing the design cost and having great economic benefits.

Key words: Solidworks; Tension machine frame; ANSYS Workbench. The optimization design

1 Introduction

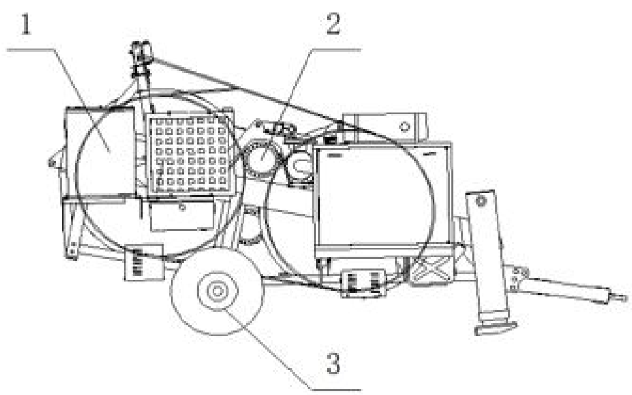
In recent years, the demand for electricity is increasing day by day with the rapid development of China's economic construction. Power construction and supply have become the pillar of national economic construction. The stringing construction is one of the most critical processes in the construction of transmission lines. The use of tension stringing not only greatly improves the efficiency, accelerates the project progress, but also significantly improves the project quality [1].

The current traction and tensioning equipment is mostly developed for ultra-high voltage construction, which is large in tonnage and heavy in weight, and is not suitable for power transmission and transformation construction. At the same time, over the years, the appearance and structure of the traction and tensioning machine has not been upgraded, and domestic peers have seriously copied. The traction and tensioning machine developed by domestic companies has no characteristics and advantages in appearance and structure, and its competitiveness is not obvious. The structure design and research of domestic traction machine and tension machine has gradually developed from relying solely on experience to using modern design methods such as finite element method. In order to adapt to the development needs of contemporary industry, reduce the quality of the tension machine and meet the transition operation, lightweight design of the tension machine is required. Liu Zhuli et al. [2] used ANSYS to perform static analysis on the QT280 trailer tractor developed by Henan Electric Power Boda Technology Co., Ltd., and the maximum displacement was greatly reduced after optimization. Wang Lihe et al. [3] used ANSYS Workbench to carry out static analysis on the subsoiler frame and completed the lightweight design. Hua Guangjun et al. [4] used ANSYS to carry out static analysis on the heavy packing steel frame box, and completed optimization by changing the size of the steel frame to achieve lightweight. But through the research, it is found that: the research on the structure optimization and finite element analysis of the tension machine is still blank in China, and there is still a lot of room for excavation in this field.

In actual work, practitioners rely on experience to judge that the design of the tension machine is often overstaffed, which greatly exceeds the use requirements, resulting in material waste and high cost. To solve the problem of insufficient research on the structure optimization of the tension machine, this paper uses Solidworks to draw the three-dimensional model of the tension machine frame, and uses ANSYS Workbench finite element analysis software to carry out static analysis on the tension machine frame, so as to obtain the deformation and stress of the tension under extreme working conditions, and uses multi-objective genetic algorithm to optimize.

2.Working principle of SA-ZY-2×40 tension machine

Tensioner is a kind of mechanical equipment used to provide resistance torque through double drums during tension stringing construction of transmission lines, so that conductors (ground wires, optical cables) can be stretched under certain tension through double drums. Tensioner is used to tension one or more wires (ground wire, optical cable) to obtain a good tension state. Tensioner is basically composed of tension generation and control device, wire spreading mechanism, mechanical transmission assembly, brake, frame and auxiliary device, as shown in the figure.



1-Tension wheel；2- Reducer；3- Walk round

Figure 1 SA-ZY-2×40 tension machine

The tension generation and control device is the most critical part of the tension machine. At present, the devices that cause the resistance torque in the paying off construction of the tensioner include overflow valve throttling, mechanical friction, electromagnetic eddy current and air compression discharge. Due to the different forms and structures of tension generating devices, the heat dissipation methods are also different, so they have their own characteristics. The hydraulic system of the tension machine is a closed system with deformation. The wire rope is pulled by the tractor, and the wire rope is connected with the wire. The wire is wound on the tensioner. The tension of the wire increases or decreases the torque through the tension wheel, the final gear set and the planetary reducer to drive the inclined shaft plunger motor to rotate. At this time, the inclined shaft plunger motor uses the rotating pump oil as the hydraulic oil pump. The oil outlet of the hydraulic pump (hydraulic motor) is connected with an overflow valve in series. The oil is throttled by the overflow valve to generate back pressure, which changes the hydrostatic pressure of the liquid and the input torque of the hydraulic pump, thus changing the rotation resistance of the tension wheel, namely, the wire tension.

3.Parametric modeling of the frame

(1) Connection between Solidworks and ANSYS Workbench

Considering the complexity of drawing 3D models with the ANSYS Workbench software, Solidworks is used to draw the 3D model of the tension machine frame. Solidworks is associated with the ANSYS Workbench software, and the 3D model is imported into the ANSYS Workbench for finite element analysis. At the same time, the relevant parameters of the model are also transferred to the ANSYS Workbench for optimization analysis [5].

(2) Parametric modeling and mechanical model simplification

The overall structure of the tension machine body is relatively complex. Before discretizing the model, simplify it. The specific method [6] is to omit the non bearing parts and skin of the frame; Widgets that have little impact on the whole can be ignored; Remove unnecessary holes and fillets.

When using Solidworks to model the frame, set the main beam width D1, axle length D2, and support thickness D3 as parameters, and the simplified model is shown in Figure 1.

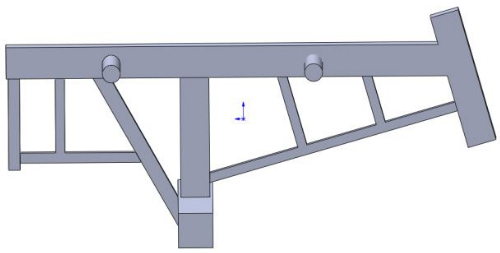


FIG. 1 Three-dimensional model of the rack

4.ANSYS Workbench finite element analysis of 2 frame

(1) Material parameter setting

When analyzing in ANSYS Workbench, you need to set material properties first. This paper studies the tension machine SA-ZY-2 * 40 of Henan Electric Power Boda Company. The frame material is Q345 steel, and its material parameters are: elastic modulus 206GPa, Poisson's ratio 0.28, density 7850kg/m^3， The yield limit is 345MPa, and the strength limit is 490~620MPa. After the material definition, the original mass of the rack is 5795.5kg.

(2) Grid division

In the finite element analysis, the grid division is the key, which not only ensures the accuracy of load loading and calculation, but also considers the workload of calculation. In general, the smaller the grid, the higher the accuracy of calculation and the greater the amount of calculation. Therefore, the calculation accuracy and efficiency should be balanced and the grid should not be too refined [7]. For this model, the size is large and the number of structures is large, taking into account the computer hardware environment, internal storage space and accuracy of calculation results. In this paper, the automatic mesh generation method is selected, and the mesh quality is controlled by Sizing size. Set the grid cell size to 20mm, click the Generate Grid button, the number of generated grids is 62872, and the number of nodes is 144524. The rack grid is shown in Figure 2.

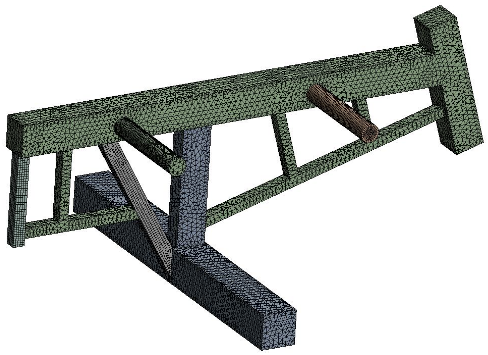


FIG. 2 Rack meshing diagram

(3) Impose boundary conditions

The tension machine is passive. The traction machine pulls the wire rope. The wire rope is connected with the wire through the connector, pulling the wire. The wire is wound on the tension wheel to drive the tension wheel to rotate. The rated power of the matching tractor of this type of tensioner is 80KN. Therefore, the maximum tension is 80KN. The gravity acceleration is 9.8m/s^2, The direction is downward. As this analysis is about the rack, the specific boundary conditions are applied as shown in Figure 3.

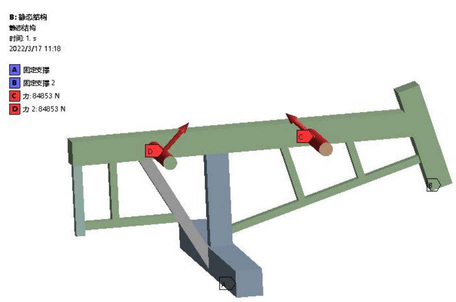
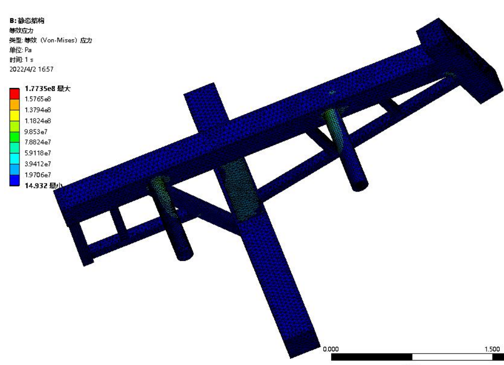


FIG. 3 Boundary conditions applied

(4) Result calculation and analysis

After calculation by ANSYS Workbench software, the equivalent force cloud diagram of the rack is obtained. It can be seen from Figure 4 and Figure 5 that the minimum equivalent stress is 14.932Pa and the maximum equivalent stress is 177.35MPa, which are mainly concentrated on the circular support bearing the tension wheel. The force on the rest parts is generally low, so these parts with low force can be optimized.



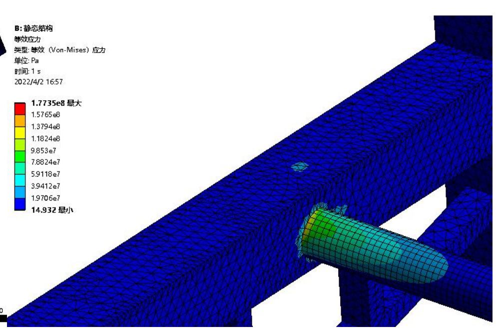


FIG. 4 Overall stress cloud diagram of the frame FIG. 5 Stress cloud diagram of the frame

part

5. Optimization scheme and response surface optimization

5.1 Optimization Scheme

In order to improve the static performance of the rack and effectively reduce the quality of the rack, it is necessary to optimize the topology of the original rack. According to the cloud figure 6 of the total deformation displacement of the frame, the main beam, axle and bracket with less deformation are the main beam, axle and bracket. Therefore, the dimensions of the three are selected as optimization parameters. The main beam D1, axle D2 and bracket D3 correspond to the input parameters P1, P2 and P3 in ANSYS Workbench respectively. Frame mass P4, maximum total deformation P5 and maximum equivalent stress P6 are set as output parameters. The material value shall be saved to the maximum extent to ensure that the whole structure is not greatly affected. The setting here changes to 20%.

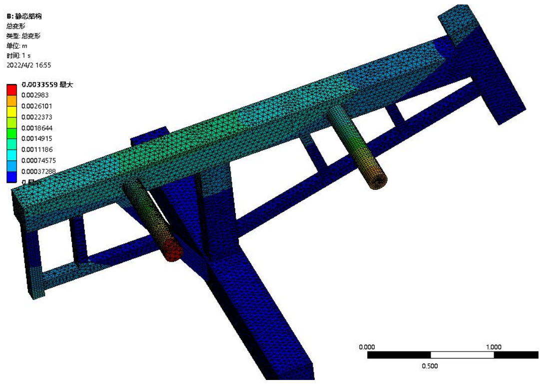


FIG. 6 Total deformation cloud

5.2 Response surface optimization

The response surface method is an optimization method that combines experimental design with mathematical statistics to establish an empirical model. Based on experimental design, empirical formula or numerical analysis, it conducts continuous experimental evaluation on the set of design points in the design space, and constructs the global approximation of objectives and constraints [8-10]. The experiment type design in this paper adopts Latin cube sampling design, which is called a "design of filled space". The number of sample points required for sampling optimization variable search space is small [10-12]. Select user-defined sample type, and take 15 sample points. According to the value range set in advance in this paper, 15 groups of input parameters will be automatically generated, and the corresponding output parameters will be generated according to the previously set materials, boundary conditions and constraints, contact conditions and loads.

Table 1 Design point table

Name	P1 （mm）	P2 (mm)	P3（mm）	P4（kg）	P5（mm）	P6（MPa）
1	245	3030	103	5781	0.33934	174.78
2	248.33	2490	83	5399.3	0.34269	166.66
3	238.33	2850	99	5595.6	0.3476	174.74
4	235	2430	107	5283.1	0.34792	169.92
5	241.67	3270	81	5894.8	0.35012	168.16
6	231.67	3210	105	5805.1	0.35223	172.98
7	228.33	3090	97	5683.2	0.35962	172.91

8	225	2790	93	5439.5	0.36536	172.43
9	218.33	2670	101	5312.5	0.37016	177.11
10	221.67	2730	91	5368	0.3702	172.65
11	211.67	2970	109	5482.2	0.37499	183.05
12	215	2610	95	5235.7	0.37725	177.86
13	208.33	3150	89	5556.3	0.38978	176.64
14	205	2550	87	5103.3	0.39579	175.64
15	201.67	2910	85	5328.5	0.40228	180.48

Where P1 is the width of the main beam; P2 is the axle length; P3 is the support thickness; P4 is the geometry mass; P5 is the maximum total deformation; P6 is the maximum equivalent stress.

It can be seen from the design point that the minimum mass m=5103.3 kg and the maximum mass m=5894.8 kg; Minimum maximum equivalent maximum stress eq σ =166.66MPa, maximum equivalent maximum stress eq σ =183.05MPa ; The minimum deformation displacement is 0.33934mm, and the maximum deformation displacement is 0.40228mm.

The multi-objective genetic algorithm MOGA method is used for optimization. The initial number of samples is 3000, the number of samples per iteration is 600, the maximum allowable Pareto percentage is 70%, and the maximum number of iterations is 20 to generate 3 candidate points[13-16]. The solution target is set to minimize the geometry mass and the total deformation.

Table 2 Candidate point table

Name	Candidate 1	Candidate 2	Candidate 3
P1（mm）	249.94	249.94	249.95
P2（mm）	2403	2402.9	2408.9
P3（mm）	109.8	107.64	109.65
P4（kg）	5381.9	5378.9	5385.9
P5（mm）	0.34066	0.34104	0.34067

Where P1 is the width of the main beam; P2 is the axle length; P3 is the support thickness; P4 is the geometry mass; P5 is the maximum total deformation.

The mass of the tension machine frame is reduced from 5795.5kg to 5383.4kg, with a weight reduction of 7.11% after optimization. Moreover, the maximum total deformation is reduced from 0.34255mm to 0.33978mm, and the equivalent maximum stress is reduced from 179.3MPa to 170.24MPa, which is far less than the yield limit (345MPa), meeting the construction requirements.

6. Conclusion

The author establishes the three-dimensional model of the tension machine frame through Solidworks and imports it into the ANSYS Workbench to establish the finite element analysis model. Under the extreme working conditions, the static analysis is carried out. It is found that the force on the frame is small, the structure is redundant, and there is a lot of optimization space.

After optimization, the mass of the tension machine frame is reduced from 5795.5kg to 5383.4kg, with a weight reduction of 7.11%. And the maximum total deformation and equivalent maximum stress are reduced, which improves the stability of the construction of the tension machine and meets the rigidity and strength required for construction. The optimization effect is obvious and lightweight is realized.

Reference

[1] Jiang Pinghai. Mechanical Equipment and Application of Tension Wire Laying [M]. Beijing: China Electric Power Press, 2004: · 33-36.

[2] Liu Zhuli, Zhou Hao, Hou Donghong. Structure analysis and optimization of traction wheel of QT280 tractor-tractor [J]. Journal of zhengzhou university (engineering science),2016,37(01):79-83.

[3] Wang Lihe, Zhao Yonglai, Cui Hongmei, Li Ying, Hu Shuping. Statics Analysis and Lightweight Design of Deep Loose Machine Frame Based on ANSYS Workbench [J]. Journal of chongqing university of technology (natural science),2019,33(02):87-93.

[4] Hua Guangjun, Yi Yingyin, Xiao Jian, Zhang Kaikai, Wu Jiping. Based on Ansys lightweight design of heavy steel box packaging engineering [J]. Journal of packaging engineering, 2022 lancet (03) : 183-188. The DOI: 10.19554 / j.carol carroll nki. 1001-3563.2022.03.023.

[5] Han Junhua. Optimization Design of Hydraulic Support Base Based on Pro/E and ANSYS Workbench [D]. Hebei University of Engineering,2016.

[6] Xu Chang. Electric bus body frame lightweight research [D]. Jilin university, 2020. The DOI: 10.27162 /, dc nki. Gjlin. 2020.002150.

[7] LI Zhuang. Finite Element Analysis and Lightweight of a Body Frame Based on ANSYS [D]. Hebei university of science and technology, 2021. DOI: 10.27107 /, dc nki. Ghbku. 2021.000203.

[8] Gao Yuhua. Structure Analysis and Optimization Design of Car Body in White with Hardware in the Ring [D]. Hefei University of Technology,2009.

[9] Liu Guiping. Research on Multi-objective Optimization Method Based on Micro Genetic Algorithm and its Application [D]. Hunan University,2008.

[10] Gao Yuhua. Structure Analysis and Optimization Design of Car Body in White with Hardware in the Ring [D]. Hefei University of Technology,2009.

Gao Yuhua Structural Analysis and Optimization Design of Car body in White based on hardware in the loop Hefei [D] University of Technology, 2009.)

[11] MO Yimin, Liu Qingchun, Gao Shuo, et al. Optimal design of key structure of a minibus rollover simplified model [J]. Machinery design & manufacture,2022,372(2):265-268,276.

[12] ZENG Yang, Zhou Jun, Shen Zhiyuan, et al. Optimization design of composite bulkhead structure based on response surface method [J]. Journal of chongqing university,2020,43(6):82-89.

[13] Chang 'e Guan, Renhua Xie. Optimization design of tractor transmission system components based on ANSYS [J]. Journal of agricultural mechanization research,2021,43(4):239-242. (in Chinese) DOI:10.3969/ j.issn.1003-188x.2021.04.045.

[14] Bu Shen-Shen, TIAN Huai-wen, ZHOU Jie. Optimal Design of Parking Roof Inspection Vehicle Underframe Based on ANSYS Workbench [J]. Computer and digital engineering,2021,49(4):822-827. (in Chinese) DOI:10.3969/j.issn.1672-9722.2021.04.040.

[15] Optimization design of chimney support steel frame using ANSYS [J]. China special equipment safety,2021,37(10):22-24. DOI:10.3969/ j.issn.1673-257x.2021.10.005.

[16] Fang Jiu Jian, Li Tian Jian. Optimization design of machine tool body size based on

ANSYS [J]. Agricultural equipment & vehicle engineering,2021,59(6):1-4.DOI:10.3969/j.issn.16733142.2021.06.001.