Original Research Article

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

Abstract:

The parameterized model of tension machine frame was established by Solidworks software, and the optimization function of ANSYS Workbench was used to optimize the frame design. Under the condition of satisfying the strength, the weight reduction design of the frame was completed. Compared with the original design scheme, the maximum equivalent stress and the maximum total shape of the optimized scheme are lower, the weight is reduced by 7.11%, and the weight reduction effect is significant, so as to reduce the design cost and have great economic benefits.

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

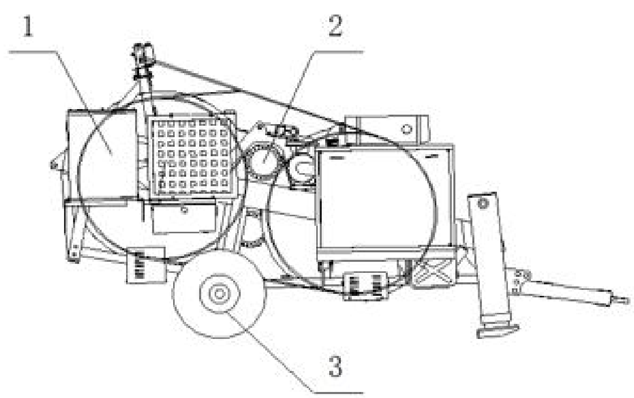
1 Introduction

In recent years, along with the rapid development of our economic construction, the power demand increases day by day, the power construction and supply become the pillar of national economic construction. Wire laying is one of the most critical processes in the construction of transmission lines. The use of tension wire laying not only greatly improves the efficiency and the progress of the project, but also significantly improves the quality of the project [1].

The tension device is developed for uhv construction more, large tonnage, heavy weight, do not apply to the power transmission and transformation construction, at the same time, over the years, stretching machine appearance structure upgrade, domestic peers plagiarize imitation is serious, stretching machine developed by domestic company on the appearance, structure characteristic and advantage, competitiveness is not obvious. The structural design and research of traction machine and tension machine in China has been gradually developed from simple empirical design to the use of modern design methods such as finite element. In order to adapt to the development of modern industry, reduce the quality of tension machine, to meet the transfer operation, it is necessary to carry out lightweight design of tension machine. Liu Zhuli [2] used ANSYS to conduct statics analysis on the QT280 towed tractor developed by Henan Electric Power Boda Technology Co., LTD., and greatly reduced the maximum displacement after optimization. Wang Lihe [3] used ANSYS Workbench to carry out statics analysis on the frame of deep loosening machine and completed lightweight design. Chuan Guangjun [4] used ANSYS to carry out statics analysis on the steel frame box for heavy packaging, and optimized it by changing the size of the steel frame to realize lightweight. However, it is found that the research on structural optimization and finite element analysis of tension machine in China is still blank, and there is still a lot of room for exploration in this field. In the actual work, practitioners rely on experience to judge that the design of the tension machine is often more than the use of the requirements, resulting in material waste and high cost. Aiming at the problem of insufficient research on structural optimization of tension machine, this paper uses Solidworks to draw a three-dimensional model of tension machine frame, uses ANSYS Workbench finite element analysis software to carry out statics analysis of tension machine frame, obtains the tension deformation under the limit condition and the stress situation, and uses multi-objective genetic algorithm to optimize.

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

Tension machine is a kind of mechanical equipment that provides resistance moment through double reels in the construction of transmission line tension, so that the wire (ground wire, optical cable) is extended under a certain tension through the double reels. The tension machine is used for tensioning one or more wires (ground wire, optical cable) to obtain a good tensioning state. The tension machine is basically composed of tension generation and control device, wire expansion mechanism, mechanical transmission assembly, brake and frame and auxiliary devices, as shown in the figure.



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

Figure 1 SA-ZY-2×40 tension machine

The hydraulic system of tension machine is a deformable closed system. The tractor pulls the wire rope, and the wire rope connects the wire, and the wire is wound around the tension machine. The tension of the wire drives the oblique shaft type plunger motor to rotate through the tension wheel, the final gear set and the planetary reducer to increase or decrease the moment. At this time, the oblique shaft type plunger motor uses the rotating pump oil as the hydraulic oil pump. The oil outlet of the hydraulic pump (hydraulic motor) is connected with the relief valve in series, and the oil is throttled through the relief valve to produce back pressure, change the liquid static pressure, change the input torque of the hydraulic pump, so as to change the rotation resistance of the tension wheel, namely: wire tension.

3.Parametric modeling of the frame

(1) Connection between Solidworks and ANSYS Workbench

Considering the complicated operation of drawing 3D model with ANSYS Workbench software, Solidworks was used to draw 3D model of tension machine frame, and Solidworks was associated with ANSYS Workbench. The 3D model was imported into ANSYS Workbench for finite element analysis, and the related parameters of the model were also transferred to ANSYS Workbench for optimization analysis [5].

(2) Parametric modeling and mechanical model simplification

The overall structure of the tensioning machine body is relatively complex, so simplification should be carried out before discretization of the model. The specific methods [6] are as follows: leaving out the non-load-bearing parts and skin of the frame; Pendants that have little impact on the whole can be ignored; Remove unnecessary holes and rounded corners.

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 established is shown in Figure 1.

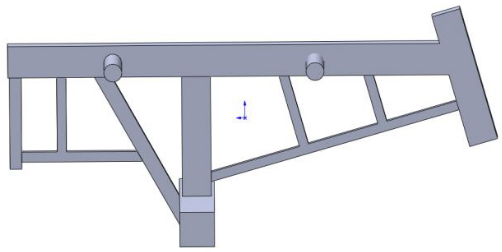


FIG. 2 Three-dimensional model of the rack

4.ANSYS Workbench finite element analysis of 2 frame

(1) Material parameter setting

When analyzing in the ANSYS Workbench, you first need to set the material properties. 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/m³, yield limit 345MPa, strength limit 490~620MPa. After material definition, the original weight of the rack is 5795.5kg.

(2) Grid division

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

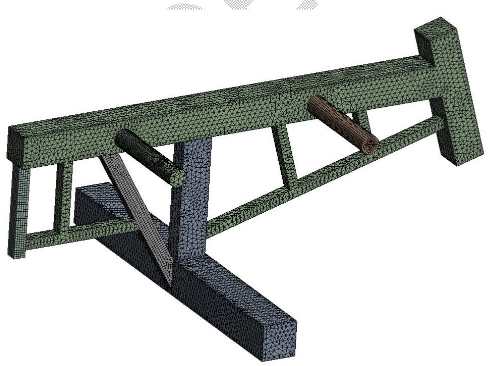


FIG. 3 Rack meshing diagram

(3) Impose boundary conditions

Tension machine force is passive, the tractor pulls the wire rope, the wire rope is connected with the wire through the connector, pull the wire, the wire wrapped around the tension wheel to drive the tension wheel rotation. The rated power of the tensioning machine is 80KN. Therefore, the maximum tension is 80KN. The acceleration of gravity is 9.8m/s squared in the downward direction. Since the frame is analyzed this time, the specific boundary conditions are imposed as shown in Figure 3.

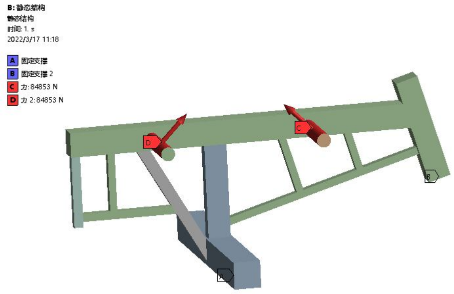
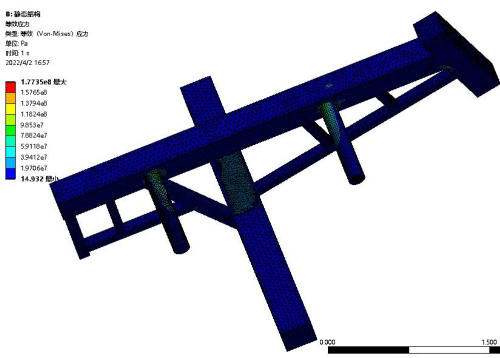


FIG. 4 Boundary conditions applied

(4) Result calculation and analysis

Through the calculation of ANSYS Workbench software, the equivalent stress cloud diagram of the frame is obtained. It can be seen from FIG. 4 and 5 that the minimum equivalent stress is 14.932Pa and the maximum equivalent stress is 177.35MPa, which is mainly concentrated on the circular bracket bearing the tension wheel. The rest of the stress is generally low, so these low stress parts can be optimized.



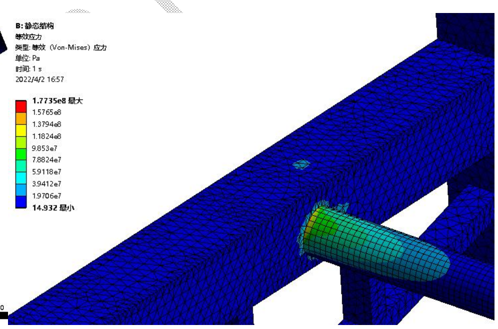


FIG. 5 Overall stress cloud diagram of the frame FIG. 5 Stress cloud diagram of the frame part

5. Optimization scheme and response surface optimization

5.1 Optimization Scheme

In order to improve the statics performance of the frame and reduce the quality of the frame effectively, it is necessary to optimize the topology of the original frame. According to the total deformation and displacement cloud Figure 6 of the frame, the main beam, axle and bracket with less deformation are included. Therefore, the dimensions of these three are selected as the optimization parameters. Main beam D1, axle D2 and support D3 correspond to 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. In order to ensure that the overall structure is not affected by too much, the value of the material is minimized. Here, the setting changes to 20%.

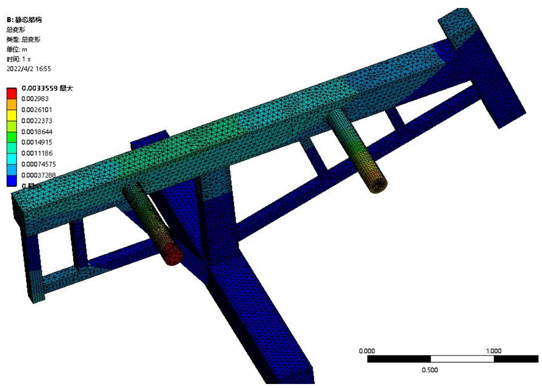


FIG. 6 Total deformation cloud

5.2 Response surface optimization

Response surface method (RSM) is an optimization method that combines experimental design with mathematical statistics to build empirical models. Based on experimental design, empirical formula or numerical analysis, it conducts continuous experimental evaluation of the set of design points in the design space and constructs the global approximation of objectives and constraints [8-10]. The experimental design in this paper adopts Latin square cube sampling design, which is known as a "space-filling design" and requires less sample points when sampling the search space of optimization variables [9-12]. The sample type is customized, and 15 sample points are selected. According to the value range set in advance in this paper, 15 sets of input parameters will be automatically generated, and 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

As can be seen from the design points, the minimum mass m= 5103.3kg, the maximum mass m= 5894.8kg; The minimum maximum equivalent maximum stress eqσ=166.66MPa, the 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 was used for optimization, and the initial number of samples was selected as 3000, the number of samples in each iteration was 600, the maximum allowable Pareto percentage was 70%, and the maximum number of iterations was 20 times to generate 3 candidate points[13-16]. The solution objective is set as minimum geometry mass and minimum total deformation.

Table 2 Candidate point table

	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

After optimization, the frame weight of the tension machine was reduced from 5795.5kg to 5383.4kg, a reduction of 7.11%. 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) and meets the construction requirements.

6. Conclusion

The author established the three-dimensional model of tension machine frame through Solidworks and imported it into ANYSY Workbench to establish the finite element analysis model. Under the extreme working condition, the author carried out the statics analysis and found that the stress on the frame part was small and the structure was small, so there was a large space for optimization.

The frame weight of the tension machine was reduced from 5795.5kg to 5383.4kg, a reduction of 7.11% after optimization,. And the maximum total deformation and equivalent maximum stress are reduced, improve the stability of the tension machine construction, to meet the construction of the required stiffness and strength. 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.