2012 International Conference on Solid State Devices and Materials Science

Study on Design Method of Series Traction Rod of Heavy Machinery

Guozhi Zhang

Abstract

The integrated design method of series traction rod of heavy machinery was studied. Based on topology optimization, parameter optimization design and reliability analysis, its integrated design method was proposed. Its topology optimization design and parameter optimization design were done. Moreover, based on probabilistic finite element method, the influence of the randomness of the design parameters to its maximum deflection and maximum equivalent stress was studied. The method and basis are supported for design and development of series traction rod of heavy machinery.

Keywords: series traction rod; probabilistic finite element method; the integrated design; topology optimization; parameter optimization design; reliability analysis

1. Introduction

As mankind enters the digital era, in recent years, modern techniques, such as optimizing design and reliability analysis, have been involved in a lot of engineering applications. The traditional design methods rely mainly on experience to design with the safety factor approach, using materials are redundancy or lack so as to lead to materials wast and fracture or other faults because of lack of strength[1-3]. However, multiple softwares or modules must be used to form a complete system development process to complete a project[4]. At present, how to effectively apply computer simulation technology for new product development and efficient integration of design engineering applications is a
problem to solve urgently, including topology optimization, parametric optimization and reliability analysis[5,6].

For different industries, methods and techniques are different, the study and application of key structure in the heavy machinery is not systematic and in-depth. Series traction rod of heavy machinery is the key carrying parts of heavy machinery, its reliability directly affects the performance of heavy machinery. Because the structure of series traction rod of heavy machinery is similar and its production quantities are very large, optimization, reliability analysis for it can effectively reduce supplies and ensure the reliability of its work. Thus, in this paper, based on FEM, topology optimization, parameter optimization, reliability analysis and other modern design methods, integrated design approach for it to study in order to find the best optimization and reliable design.

2. The design scheme of series traction rod of heavy machinery based on simulation technology

In this paper, based on virtual prototype, combining with theory(materials mechanics and composite materials mechanics) with FEM, the overall research program for series traction mechanism of heavy machinery was proposed and it is shown in Fig.1.

![Diagram](image)

Fig.1 The overall research program for series traction mechanism of heavy machinery

3. The profile of working condition of series traction rod of heavy machinery

The traction rod of heavy machinery is the traction device of heavy machinery, which plays the role of pull and carry the load. It is as shown in Fig.2. It is generally welded structure, which is widely used in hydraulic support machinery. Its working condition is very poor. The carrying loads are bending loads and axial loads.
4. The optimization design and reliability analysis of series traction rod of heavy machinery

**Topology optimization of the overall structure.** In order to determine the main design of structure of traction rod of heavy machinery, its preliminary analysis of the topology optimization was done. Its finite element model of topology optimization is shown in Fig.3 (a), and the initial topology optimization results is shown in Fig.3 (b). From the calculation results, the load point is as the cutoff point. The stiffness of its left part need to strengthen, while the stiffness of its right part is higher relatively. Therefore, to improve its overall structural stiffness, ribs in its left part should be set more, and the rib is near the load position.

![Initial finite element model of topology optimization](image)

(a) The initial finite element model of topology optimization

![Topology optimization result](image)

(b) Topology optimization result

**Parameter optimization of the overall structure.** Based on its topology optimization results, its optimization based on non-uniform stiffness and parameter optimization were done. Design parameters and corresponding symbols used in parameter optimization is shown in Tab.1. Optimizeation object is its total volume, the selected optimization caculation methods is the sub-problems optimization caculation methods. The maximum number of iterations was set 100. Convergence iteration number is 11 times. The final optimization results is shown in Tab.2.
Tab.1 The optimization parameters

<table>
<thead>
<tr>
<th>Design variables</th>
<th>1st rib Position L1(mm)</th>
<th>2nd rib Position L2(mm)</th>
<th>3rd rib Position L3(mm)</th>
<th>4th rib Position L4(mm)</th>
<th>1st rib thickness T1(mm)</th>
<th>2nd rib thickness T2(mm)</th>
<th>3rd rib thickness T3(mm)</th>
<th>4th rib thickness T4(mm)</th>
<th>constraint condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Values Minimum</td>
<td>500</td>
<td>1300</td>
<td>1600</td>
<td>1900</td>
<td>15</td>
<td>15</td>
<td>15</td>
<td>15</td>
<td>Maximum deflection UUMIN (mm)</td>
</tr>
<tr>
<td>Maximum Values</td>
<td>1300</td>
<td>1500</td>
<td>1700</td>
<td>2200</td>
<td>25</td>
<td>25</td>
<td>25</td>
<td>25</td>
<td>Maximum equivalent stress SEQV (MPa)</td>
</tr>
</tbody>
</table>

Tab.2 The optimization results

<table>
<thead>
<tr>
<th>Design variables</th>
<th>1st rib Position L1(mm)</th>
<th>2nd rib Position L2(mm)</th>
<th>3rd rib Position L3(mm)</th>
<th>4th rib Position L4(mm)</th>
<th>1st rib thickness T1(mm)</th>
<th>2nd rib thickness T2(mm)</th>
<th>3rd rib thickness T3(mm)</th>
<th>4th rib thickness T4(mm)</th>
<th>Subjected condition</th>
<th>Design object</th>
<th>Total volume VV (mm^3)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>904.96</td>
<td>1383.0</td>
<td>1661.5</td>
<td>2060.8</td>
<td>16.717</td>
<td>15.18</td>
<td>15.83</td>
<td>15.074</td>
<td>Maximum deflection UUMIN (mm)</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Maximum equivalent stress SEQV (MPa)</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Total volume VV (mm^3)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Tab.3 The final optimal design results

<table>
<thead>
<tr>
<th>Design variables</th>
<th>1st rib Position L1(mm)</th>
<th>2nd rib Position L2(mm)</th>
<th>3rd rib Position L3(mm)</th>
<th>4th rib Position L4(mm)</th>
<th>1st rib thickness T1(mm)</th>
<th>2nd rib thickness T2(mm)</th>
<th>3rd rib thickness T3(mm)</th>
<th>4th rib thickness T4(mm)</th>
<th>Subjected condition</th>
<th>Design object</th>
<th>Total volume VV (mm^3)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>905</td>
<td>1383</td>
<td>1662</td>
<td>2061</td>
<td>15</td>
<td>15</td>
<td>15</td>
<td>15</td>
<td>Maximum deflection UUMIN (mm)</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Maximum equivalent stress SEQV (MPa)</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Total volume VV (mm^3)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Its internal structure and force diagram was shown in Fig.4. Where, $F_1 = 611764.4$ N, $F_2 = 163921.8$ N. Based on optimization results, the final design parameters used in engineering is shown in Tab.3, and its optimization structure is shown in Fig.5. Its weight reduce 23.4% after optimization, through comparing with the weight of original design.
Reliability analysis of the overall structure. Its parameters and corresponding symbols in reliability analysis are shown in Tab.4. With ANSYS PDS, the effects of random fluctuations of these parameters to its maximum deflection were studied. The numerical methods used is central composite sampling and Monte Carlo extending sampling method, the number of iterations is 81.

<table>
<thead>
<tr>
<th>Mean value and standard deviation</th>
<th>1st rib Position L1(mm)</th>
<th>2nd rib Position L2(mm)</th>
<th>3rd rib Position L3(mm)</th>
<th>4th rib Position L4(mm)</th>
<th>1st rib thickness T1(mm)</th>
<th>2nd rib thickness T2(mm)</th>
<th>3rd rib thickness T3(mm)</th>
<th>4th rib thickness T4(mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean value</td>
<td>905</td>
<td>1383</td>
<td>1662</td>
<td>2061</td>
<td>15</td>
<td>15</td>
<td>15</td>
<td>15</td>
</tr>
<tr>
<td>Standard deviation</td>
<td>10</td>
<td>10</td>
<td>10</td>
<td>10</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

Sensitivity analysis results are shown in Fig.6. From Fig.6 (a), its maximum deflection is more sensitive for its ribs thickness and its maximum deflection is most sensitive for its second rib thickness, while its maximum deflection is not sensitive for its ribs support positions. From Fig.6 (b), its maximum equivalent stress is more sensitive for its third rib thickness and support position, and not sensitive for other parameters. Iterative sampling results are shown in Fig.7. From Fig.7 (a), the changing range of its maximum deflection is from 8.799mm to 8.807mm. From Fig.7 (b), the changing range of its maximum equivalent stress is from 394.8Mpa to 406.47MPa. Based on above analysis results, in order to ensure its working reliability, design parameters and support positions of key ribs should be strictly controlled.
Fig. 6 Sensitivity analysis results

(a) Maximum deflection

(b) Maximum equivalent stress

Fig. 7 Iterative sampling results

(a) Maximum deflection

(b) Maximum equivalent stress
5. Conclusions

The conclusion is followed through the study of the paper:

(1) Based on topology optimization, parameter optimization design and reliability analysis, integrated design method of series traction rod of heavy machinery was proposed.

(2) Its finite element model was established, with this model, topology optimization and parameter optimization design of typical structure traction rod of heavy machinery were done, and its optimization design was obtained.

(3) With central composite sampling and Monte Carlo extending sampling method, the effects of random fluctuations of these parameters to its maximum deflection were studied.

The study in the paper provides a new analysis method and engineering design basis for the study on the design and performance analysis of series traction rod of heavy machinery.

Acknowledgements

This work was financially supported by the Natural Science Foundation of Henan Province (2010C460007) and Innovation Foundation of Xinxiang University.

References