Finite element analysis and optimization of workover derrick based on ANSYS Workbench

Jinlong Wang1, Junliang Zhang2,* and Xiaqi Zhang3

1Weifang University, Weifang, Shandong 261061, China
2Shandong Provincial Engineering Laboratory of Solar Energy Utilization Core Component Performance Testing and Optimization, Dezhou University, Dezhou, Shandong 253023, China
3Shandong University of Science and Technology, Qingdao, Shandong 266590, China
*Corresponding author’s workplace e-mail: zhangdzu@126.com

Abstract. Based on the actual workover operating conditions, this paper designs a workover operating machine for lifting and running tubing at the wellhead, and use SolidWorks to draw a three-dimensional model of the workover rig, through ANSYS Workbench, the main component of the workover rig is analyzed for the force under load. According to the results of static analysis and modal analysis, it is concluded that the main stress parts of the workover derrick are on the two uprights where the robot arm is installed, and the upper part of the workover derrick is most prone to resonance, after multi-objective optimization of the workover derrick, the deformation and the maximum stress of the workover derrick have been reduced by 9.5% and 9.8%, respectively. This research content provides a reference for the design of workover rigs in the future.

1. Introduction
Petroleum is the fastest and most used resource in human resource utilization, it can be used almost as much as industrial construction or as a fuel for ordinary transportation. With the increase in petroleum consumption, petroleum extraction is also facing huge challenges. As a key link in petroleum extraction, workover operations play an important role, among which pipe string operations at the wellhead are common operations. At present, most workover operations are still traditional manual operations, the working environment is relatively harsh, the labor intensity is high, and the risk is high. With the continuous development of society, intelligence and automation have become the main development trend of oilfield workover equipment[1]. Therefore, it is of great significance to develop automated equipment that is in line with workover operations to improve work efficiency, and to carry out research on tubing grabbing and transporting robots during workover operations[2]. According to the actual workover environment, this paper designs an automated workover rig for raising and lowering tubing at the wellhead, so as to improve the existing workover environment and improve the efficiency of workover operations.

2. Spatial structure diagram of workover operation machine
According to actual production needs, this paper designs a workover rig, and uses SolidWorks as a tool to draw a three-dimensional model of the workover rig. The workover rig includes three parts: a workover derrick, a transfer manipulator and a knob manipulator. Figure 1 is the space structure.
diagram of the workover rig. However, as the skeleton of the entire machine, the workover derrick is the basis for the installation of all mechanical components[3]. The performance difference will directly affect the safety of the robotic arm. Therefore, this article will conduct in-depth research and analysis on the workover derrick. The three-dimensional model of the workover derrick is shown in Figure 2.

3. Finite element analysis of the workover derrick

3.1. Statics analysis of workover derrick

Static analysis is the most basic analysis in finite element analysis, it mainly analyzes the deformation and stress distribution of structural parts under static load, so as to judge the parts that are prone to damage and make improvements. According to the theory of classical mechanics, the general equation of the dynamics of an object can be obtained as:

\[
[M][x^s] + [C][x'] + [K][x] = \{F(t)\}
\]

Among them, \([M]\) is the mass matrix; \([C]\) is the damping matrix; \([K]\) is the stiffness matrix; \(\{x\}\) is the displacement vector; \(\{x'\}\) is the velocity vector; \(\{x^s\}\) is the acceleration vector; \(\{F(t)\}\) is the force vector.

Based on the above-mentioned statics theory formula, the statics analysis of the workover derrick is carried out, and the constraints are imposed on the workover derrick according to the working conditions of the workover derrick, and then the grid is divided and the grid independence is verified. The relationship between the grid size and the analysis results is shown in Figure 3. To meet the requirements of calculation accuracy, this paper selects the grid size as 15 mm, and the workover derrick after the grid is divided is shown in Figure 4, a total of 1315179 nodes and 601426 units were obtained.
Figure 3. Grid independent relationship diagram

Figure 4. Meshing diagram

After static analysis, the deformation and stress distribution of the workover derrick are shown in Figure 5 and Figure 6. It can be seen from Figure 5 that the main deformation area of the workover derrick is on the upper part of the workover derrick and the column where the robot arm is installed, and the maximum deformation is 0.00884mm; from the stress distribution diagram in Figure 6, it can be concluded that the stress is mainly concentrated on the column part where the robotic arm is installed, and the maximum stress is 1.4408MPa.

Figure 5. Total deformation cloud diagram before optimization
Figure 6. Equivalent stress cloud diagram before optimization

3.2. Modal analysis of workover derrick

Modal analysis is the analysis to calculate the vibration characteristics of the structure, it can help designers analyze the natural frequency and vibration mode of the structure, so that the structure design can avoid resonance and avoid safety accidents[4]. Modal analysis can be obtained from Newton's theory of mechanics, the general equation expression of dynamics is:

$$[M]\{\ddot{x}\}+[C]\{\dot{x}\}+[K]\{x\} = \{F(t)\}$$

Where: $[M]$ is the mass matrix; $[C]$ is the damping matrix; $[K]$ is the stiffness matrix; $\{x\}$ is the displacement vector; $\{\dot{x}\}$ is the velocity vector; $\{\ddot{x}\}$ is the acceleration vector; $\{F(t)\}$ is the force vector.

This article mainly conducts a modal analysis of the workover derrick in a free state, so the influence of damping on the analysis results can be ignored. At the same time, the natural frequency and mode shape of the workover derrick itself will not change with the change of the external load, so
the external load is also 0. In this way, the equation expression (2) is simplified to:

\[
[M]\{\ddot{\mu}\} + [K]\{\mu\} = \{0\}
\]  

(3)

When harmonic vibration occurs, that is \(\mu = U \sin(\omega t)\), the equation expression (3) becomes:

\[
([K] - \omega^2[M])\{\phi_i\} = 0
\]  

(4)

Where, \(\omega_i\) is the i-th natural frequency of the structure, \(\{\phi_i\}\) is the i-th mode shape vector of the structure.

After modal analysis, the first six natural frequencies and mode shape cloud diagrams of the workover derrick are shown in Figure 7.
According to the modal analysis deformation cloud map of the workover derrick, it can be seen that the deformation area of the workover derrick is mainly concentrated on the upper part. Among them, the maximum deformation occurs at a frequency of 90.084 Hz and the deformation is 5.6672 mm, and the minimum deformation occurs at a frequency of 47.488 Hz and the deformation is 1.4922 mm. Therefore, the upper part of the workover derrick is the place where resonance is most likely to occur, and according to the frequency of deformation, the workover derrick should be avoided to work in this frequency range.

4. Multi-objective optimization of parameters of workover derrick

Multi-objective optimization is a trend in the development of optimization now, it can achieve the optimization of multiple objectives, so as to obtain the optimal solution\[^5\]. Avoid the problem of reduced performance of structural components caused by single-objective optimization\[^6\]. This article uses the ANSYS Workbench multi-objective optimization module to perform multi-objective optimization of the workover derrick. The optimized static analysis deformation and stress cloud diagram is shown in Figure 8 and Figure 9.

From the deformation cloud diagram and the equivalent stress cloud diagram, it can be seen that the optimized workover deformation is 0.008mm, which is 9.5% less than before optimization, the maximum stress after optimization is 1.3119MPa, which is 9.8% less than before optimization.

Figure 7. Deformation cloud diagram of the first six modes of the workover derrick

Figure 8. Total deformation cloud diagram after optimization

Figure 9. Equivalent stress cloud diagram after optimization
5. Conclusion
Through in-depth research and analysis of the workover derrick, this article mainly draws the following conclusions:

(1) The main stressed parts of the workover derrick are on the two uprights where the robotic arm is installed, and the upper part of the workover derrick is most prone to resonance;

(2) After the multi-objective optimization of the workover derrick, the deformation and the maximum stress of the workover derrick have been reduced by 9.5% and 9.8%, respectively.

Acknowledgments
This work was supported by funding from Shandong Provincial Engineering Laboratory of Solar Energy Utilization Core Component Performance Testing and Optimization.

References
[1] Jiao W G, Zhu K Q, Hu Y and Hu S Q 2017 Design of virtual prototype of manipulator for workover rig platform J. Oil Field Equipment. 46(02) 35-38.
[2] Geng Y G, Gu Q F, Wang J W, Huang S, Sun L H and Wang T F 2016 Development of a multifunctional manipulator for starting and running tubing for minor repair operations J. Oil Field Equipment. 45(12) 66-70.
[3] Zhang X Q 2018 Finite element analysis of static strength of stationary operating derricks J. China Petroleum and Chemical Standards and Quality. 38(19) 105-106.
[4] Wu W X, Deng K F, Xu F Y, Hua J and Fu X Q 2015 Modal analysis of 450T offshore derrick and suggestion of drilling technology J. Oil Field Equipment. 43(12) 64-67.
[5] Luo C Y and Dong F 2013 Design optimization of the base structure of ZT160 workover rig based on ANSYS J. Journal of Qiqihar University (Natural Science Edition). 29(05) 76-79.
[6] Guan F, Huang L H, Zheng L W, Yu Q L, Li P and Li J 2009 Finite element analysis and optimization design of the base of offshore workover rig J. China Petroleum Machinery. 37(09) 38-41+184.