World Scientific Research Journal (WSRJ) ISSN: 2472-3703
www.wsr-j.org

Finite Element Analysis of Locked Drill Pipe of Rotary Drilling Rig

Teodor Grigorie and Nitaigour Lucian Mahalik

California State University, Fresno, CA 93740, USA

Abstract: Rotary drilling rig is a kind of mechanical equipment for the construction of earth-boring and perfusion piles, and it is used in conjunction with drill rods and drilling tools. With the advent of more and more high, large and heavy construction projects, traditional construction methods and construction equipment have been unable to meet the requirements of the current construction. It is based on this background that rotary drilling rigs have achieved considerable development. The drill pipe is an essential tool for the construction of rotary drilling rigs. Generally, the drill pipe is subjected to large torque, large drilling pressure, and slender rods during construction. The structure easily causes troubles such as instability of the press rod, which makes the force condition of the drill rod during the drilling operation extremely complex, and greatly increases the risk of the drill rod being damaged. Based on the painful lessons learned during the construction, the weak links (stress concentration areas) of the drill pipe structure were identified, and the failure analysis of the drill pipe was accurately performed, and reasonable interpretation became urgently important.

Keywords: Portable-type earth drill holder, finite element, mode analysis.

1. INTRODUCTION

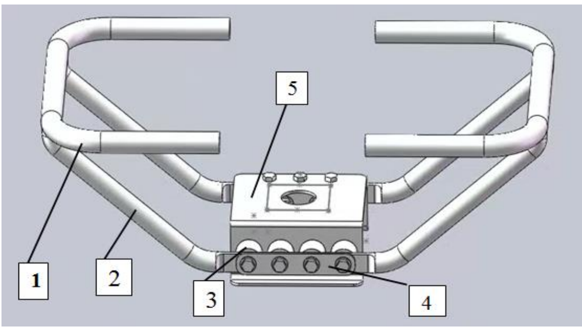
Hand-held digging machine is also called portable digging machine. It is a special machinery for afforestation and fertilization in mountainous, hilly and gully areas. Its powerful and powerful, high efficiency, suitable for a variety of terrain, easy to carry and wild field operations. Since the hand-held digging machine generally employs a two-stroke engine, it inevitably generates a large vibration during use. However, the hand-held digging machine requires the operator to perform handheld operation at the scene, so the reliability of the machine bracket part is directly Affects the use of the entire machine.

Modal analysis is used for vibration measurement and structural dynamics analysis to obtain modal information such as natural frequency and mode shape of the system, which is beneficial for guiding the mechanical design. In recent years, this technology has been widely used in the development and optimization of mechanical equipment. Related researchers have optimized the modal properties of key components to ensure the stable operation and safety of machinery: In 2011, Huang Handong and others used ANSYS/LS-DYNA to cut sugar cane. For finite element simulation [1], in 2008 Zhang Libin et al. used the LMS Test.Lab test system to test and analyze the vibration of small agricultural work machines [2]. In 2003, Zuo Yanyan conducted vibration tests on the mid-horsepower tractor bonnets. Vibration study [3]; In order to further improve the safety and operation performance of digging machines, this paper intends to use a modal analysis method combining theory and experiment to conduct in-depth research on the modal attributes of the 3WT-300 portable digging machine bracket part, Improve and provide optimization solutions.

2. DIGGER BRACKET MODEL ESTABLISHMENT

2.1 Establishment of 3D Model of Digger Support

In this paper, a large-scale three-dimensional software SolidWorks is used to establish a digger bracket model. In order to reduce the workload of modeling and improve the computational efficiency, some necessary simplifications and assumptions are made for the digging machine frame: partial chamfering and rounding are neglected, and the analysis is neglected. The thread features that have a smaller impact but occupy more memory, ignoring the parts with lower mass and having no significant effect on the structural rigidity; assume that the material is linearly elastic and uniform and continuous; the connections between the various components are considered to be rigid connections. The simplified 3D model is shown in Figure 2.1.



1. Handle 2. Support bar 3. Rubber spring 4. Connection board 5. Base

Fig.2.1 Portable-type earth drill holder mode

2.2 Establish a finite element model

Because the vibration and noise analysis does not require high grids, the method of automatic division and manual division is adopted, and tetrahedron units are used to divide the sections. In the selection of element type, the tetrahedron element is not as accurate as the hexahedron element, but the tetrahedron element is easy to generate a mesh automatically (the element has the advantage of adaptability), especially for a structure having a complex geometry such as a digger, Has great modeling advantages. Therefore, tetrahedral elements are used in consideration of both accuracy and modeling efficiency.

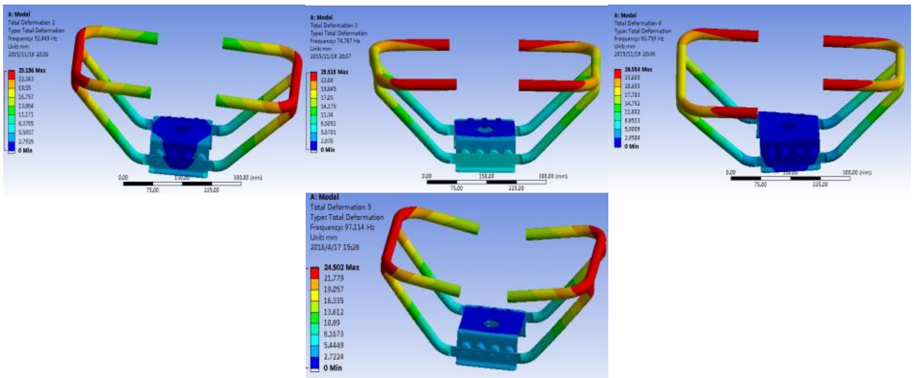
The bracket is an important part of the digging machine body, and it is subjected to various loads. Its vibration characteristics have an important influence on the overall vibration of the digging machine, so it is required to have sufficient rigidity and strength. Finite element meshing was performed on the simplified stents. A total of 8195 cells and 16729 nodes were used. The bracket material is 45 steel. The material parameters were: density 7850 (kg/m^3), Poisson's ratio 0.269, and Young's modulus 210 GPa.

3. FINITE ELEMENT ANALYSIS OF DIGGER SUPPORTS

The modal analysis of this study uses a free-mode analysis method. Since the first 6 modes of the free mode are rigid modes, the first mode in the true sense starts from the calculated seventh stage. In the modal analysis, the vibration mainly occurs in the first few frequencies, so the solver is set to 10th-order modal analysis. The calculated first 10 modes are the first 4 modes in the true sense, as shown in Table 1. The natural frequency and mode shape description of the stent shown. According to the theory of vibration, it is known that the low-order modes play a major role in the process of structural vibration. The higher-order modes have little effect on the response, and the attenuation is faster, so only the lower-order modes need to be considered. Only the first four-order modal shapes of the stent are shown here, as shown in Fig3.1, and the stent natural frequency table 1 is given.

Table 1 Natural frequencies of holder

Order	Natural frequency (Hz)
1	52.949
2	74.787
3	83.759
4	97.114



a.1st order mode b.2st order mode c.3st order mode d.4st order mode

Fig.3.1 1～4st order mode of holder

It can be seen from the deformations of the brackets shown in Fig2.1 under different vibration modes: Under the first vibration mode, a bending vibration of the bottom plate in the XOZ plane generates a large displacement, and there is a large vibration angular displacement at the connection plate. There is a bending vibration displacement in the XOZ plane at the transition to the support rod connection; in the second vibration mode, the base generates a bending vibration in the XOZ plane at the side of the bolt connection, and the base moves axially along the Z axis, resulting in a large handlebar. Vibration angular displacement; under the three-order vibration mode, the entire bracket in the XOZ plane produces bending vibration, the handle twists along the Z axis, and the vibration angle displacement of the base reaches a maximum; in the fourth vibration mode, the angular displacement of the base follows the order As the times increase, the stent will begin to produce torsional oscillations. It is noteworthy that the low-order vibration modes, on the one hand, cause the base to produce a larger angular displacement, and on the other hand, the handlebars and the support rods are connected at a large bending deformation, and these bending deformations make it easy to connect them. Bend cracks appear.

4. COMCLUSION

The parameterized model of the digger frame was established through large-scale three-dimensional software Solidworks, and the modal analysis of the frame was performed based on the ANSYS Workbench to obtain the first four modal frequencies and modes of the frame. Through the comparison of mode shapes, it can be seen that the bracket has a large deformation at the connection between the handlebar and the support rod, which is easy to damage, and the deep processing of the material or the reinforcement of the material at this position should be performed to improve the deformation resistance of the part. Avoid breakage in this part, causing damage to the operator.

REFERENCES

[1] Huang Handong,Wang Yuxing,Tang Yanqin,et al.Finite element simulation of sugarcane cutting process[J].Transactions of the Chinese Society of Agricultural Engineering, 2011, 27(2):161-166

[2] Zhang Libin, Jiang Fan, Wang Yangxi, et al. Vibration testing and analysis of agricultural work machines based on LMS Test.Lab[J].Transactions of the Chinese Society of Agricultural Engineering,2008,24(5):100-104

[3] Zuo Yanyan, Fang Yuying. Vibration Reduction of Tractor Engine Hoods in Middle Horsepower[J]. Journal of Agricultural Engineering, 2003, 19(4): 107-110