Modal analysis on swinging structure of hydraulic press

Wang Xiaoying¹,a*, An Xiaowei²

¹College of Mechanical Engineering Shandong Huayu Institute of Technology Dezhou, Shandong, 253000, China
²College of Mechanical Engineering Shandong Huayu Institute of Technology Dezhou, Shandong, 253000, China
¹a Corresponding author’s e-mail: wxy@huayu.edu.cn

Abstract—In the actual work of hydraulic press, in order to ensure the design of auxiliary punching structure, that is, swing punching structure runs smoothly, the modal analysis of swing punching structure is carried out by using ANSYS software. The first 40 natural frequencies are determined, and the first 10 modes are analyzed. The shape variables and deformation modes are determined. Thus, the actual vibration response of the structure under the action of various external or internal vibration sources in each frequency band can be predicted to avoid resonance phenomenon in the work. At last, the aim of safety production, material saving and cost reduction is achieved.

1. INTRODUCTION
The swing punching structure is added to the forging and forging machine, which ensures that the rough and punching process of the workpiece blank can be completed on the same hydraulic press. This structure is simple in design, small in space and easy to operate. It can improve the work efficiency and ensure that the workpiece has high repeated positioning accuracy when moving in and out. The production cost of ring forging is significantly reduced, but in the actual work process, the pendulum punch structure is installed on the column of hydraulic press in the form of cantilever, so it is easy to appear stress concentration and resonance phenomenon. Swing punch structure is shown in Fig 1.

![Swinging structure diagram](image-url)
2. THEORETICAL BASIS OF MODAL ANALYSIS
Any structure has its own natural frequency. Resonance should be avoided in mechanical structure design and engineering problems\textsuperscript{[1,2]}. The vibration mode is the inherent whole characteristic of elastic structure. Through modal analysis, the main modal characteristics of the structure in a certain frequency range which is easily affected can be obtained. It can predict the actual vibration response of the structure under various internal or external sources\textsuperscript{[3,4]}.

The natural frequency is the inherent property of the system, independent of the external load, and its undamped free vibration equation is as follows:

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

In the formula, $[M]$ is the mass matrix; $[K]$ is the stiffness matrix; $\{x\}$ is the displacement vector; $\{\ddot{x}\}$ is the acceleration vector. The characteristic equation of the above formula is:

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

In the formula, $\omega_n$ is the natural frequency and $\phi_i$ is the corresponding mode shape.

3. MODAL ANALYSIS OF SWINGING STRUCTURE

3.1. Finite element modelling
Before the finite element modeling of swing arm structure, the actual structure is simplified. The purpose is to reduce the cost and scale of finite element analysis, and to ensure the calculation accuracy of finite element analysis is the precondition of simplification\textsuperscript{[5]}.

First, the whole structure is treated as a continuum. The swing arm of hydraulic press uses steel plate welding structure, the welding quality is reliable, and has no effect on the force transmission effect, so the whole structure is treated as a continuum.

Secondly, in order to simplify the model, simplify the calculation and save time, the structure of the model is simplified without affecting the analysis results. The transmission part connected with the column and the screw thread, groove and limit block which have little influence on the analysis results in the swing arm structure are removed.

Third, because the punch is a solid iron block with a diameter of about 300 mm and a height of 560 mm, its rigidity is much greater than the swing arm part. Therefore, the maximum weight of the punch is converted into equivalent surface force applied to the upper surface of the punch fixing plate of the swing arm, and the punch is no longer considered\textsuperscript{[6,7]}.

The simplified model is shown in Fig 2.

![Figure 2. Swinging structure solid model](image-url)
Four node tetrahedron element is selected to simulate the swing arm structure. Because the mass and stiffness of the swing arm structure are uniformly distributed, the uniform mesh is adopted. After discretization, the finite element model has 208278 nodes and 329650 elements, and the finite element model of pendulum structure is shown in Fig 3.

![Finite element model of swing impact structure](image)

Figure 3. Finite element model of swing impact structure

### 3.2. Modal analysis results and modes

The lower mode energy accounts for a large proportion, which has a great influence on the structural response\[^{[8]}\]. Therefore, the first 40 natural frequencies and modes of shimmy structure are solved, and the first 20 natural frequencies are shown in Table 1.

| Frequency order | Natural frequency (Hz) | Frequency order | Natural frequency (Hz) |
|-----------------|------------------------|-----------------|------------------------|
| 1               | 21.050                 | 11              | 1124.6                 |
| 2               | 25.636                 | 12              | 1159.2                 |
| 3               | 80.640                 | 13              | 1278.1                 |
| 4               | 98.933                 | 14              | 1415.2                 |
| 5               | 252.31                 | 15              | 1503.4                 |
| 6               | 431.13                 | 16              | 1517.0                 |
| 7               | 620.43                 | 17              | 1708.5                 |
| 8               | 747.55                 | 18              | 1839.0                 |
| 9               | 868.50                 | 19              | 868.50                 |
| 10              | 1050.5                 | 20              | 2044.2                 |

The change trend of natural frequency of the first 10 modes is shown in Fig 4.

![Histogram of natural frequency of the first 10 modes](image)

Figure 4. Histogram of natural frequency of the first 10 modes
The first 10 modes are shown in Figures 5 to 14. As can be seen from the first mode diagram in Fig. 5 and the second mode diagram in Fig. 6, the end of the swing arm is deflected and bent downward.

The third mode diagram in Fig 7 shows the upward bending of the rear side of the swing arm plate, and the fourth mode diagram in Fig 8 shows the downward bending of the rear side of the swing arm plate.
It can be seen from the fifth mode vibration diagram of Fig. 9 that the end of the swing arm plate is bent upwards and the middle part is bent downward. Fig 10 shows the torsional deformation of the swing arm in the sixth mode diagram.

It can be seen from the 7th mode diagram in Fig 11 that at this time, the rear end of the swing arm plate is bent downward, and the middle part of the rear side is bent upward, accompanied by torsional deformation. Fig 12 the 8th mode diagram shows that the back side of the plate end is bent upward.
Figure 11. The seventh mode diagram of swing arm

Figure 12. The eighth mode diagram of swing arm

From the ninth mode vibration diagram of Fig 13, it can be seen that the rear end of the swing arm plate is bent upwards, and the middle part of the rear part is bent down, accompanied by torsional deformation. The 10th mode diagram in Fig 14 shows that the plate part is bent and twisted, and the vertical plate is also bent and deformed.

Figure 13. The seventh mode diagram of swing arm
Through the above analysis, it can be seen that the vibration deformation of the structure is mainly bending deformation\(^9\). When the frequency is high, the platform and the vertical plate of the pendulum punch structure have great variability and distortion. With the increase of the natural frequency, the deformation increases gradually\(^10\). When the frequency is 1050 hz, the maximum deformation is 12.567 mm.

However, the low modal energy accounts for a large proportion in the actual work of hydraulic press, so these low-order frequencies, especially the previous order values, should be avoided in the work, so as to avoid equipment damage or casualties caused by resonance phenomenon.

4. CONCLUSION
The modal analysis of shimmy structure is carried out by using ANSYS software, the first 40 natural frequencies are determined, and the first 10 modes are analyzed. It effectively avoids the resonance phenomenon of hydraulic press in the work, so as to achieve the purpose of safety production.

ACKNOWLEDGMENT
This achievement is the content of Shandong Huayu Institute of technology science and technology project "structural optimization of large ring forging hydraulic press" (2019kj04), thanks for the financial support from the school.

REFERENCES
[1] Dhirendra Pratap Singh, Sanjay Mishra, Rajesh Kumar Porwal. Modal Analysis of Ultrasonic Horn using Finite Element Method[J]. Materials Today: Proceedings, 2019, 18 (Pt 7).
[2] Du Qingchuan, Cheng Ying, Ren Peirong. Finite element model modification of airframe based on experimental modal analysis [J]. Mechanical design, 2019, 36 (S2): 14-19.
[3] Wu Yuhui. Modal analysis and experimental verification of tank bottom of aerospace structure [C]. Chinese society of vibration engineering. Proceedings of the 13th National Conference on vibration theory and application. Chinese society of Vibration Engineering: Chinese society of vibration engineering, 2019: 228-232.
[4] Wei Hongxin, Wang Zhisen, Hu Yu, Liu bangxiong. Finite element modeling and modal analysis of steam turbine rotor [J]. Nanfang agricultural machinery, 2019, 50 (19): 53-54.
[5] Yu Chong, Zhang Ling. Finite element modal analysis of main frame of crawler moving crusher [J]. Cement engineering, 2019 (05): 54-55.
[6] Du Ren, Li Jiadong, Zhao Zhongxian, Meng Qikai, Zhao Zikai, Liang CE, Zhao Chuan, Liu Chunyu. Modal analysis of the frame of disc mower and conditioner based on ANSYS Workbench [J]. Mechanical design, 2019, 36 (S2): 63-66.
[7] Yang Desheng, Jiang Qingfeng, Liu Qing, Li Xianbin, Rao Zhiming. Structural strength and modal analysis of subframe based on ANSYS [J]. Manufacturing automation, 2019,41 (09): 12-14.

[8] Zhang Kun, Lu Yu, Chen Lidong, Guo Guangliang, Jia Guoqing, Wu Yu. Finite element analysis of track support frame of track type curb slip form machine based on ANSYS Workbench [J]. Journal of Hebei Normal University of science and technology, 2019,33 (03): 57-61 + 80.

[9] Xu Zijing, Wang Juan, Li Minghai. Strength check and modal analysis of connecting rod based on ANSYS Workbench [J]. Railway locomotive and motor car, 2019 (08): 1-5 + 19.

[10] Chen Yingfei, Yin Xiangyun, Wang Chao, Zhang Fan, Xu Huining, Yin Guofu. Modal and harmonic response analysis of dynamic calibration device of high speed rotating axle balance based on ANSYS Workbench [J]. Mech., 2019,46 (07): 11-16 + 35.