

Modal analysis of impact test platform

Yang Lin¹, Mengmeng Song^{2,*}, Xiaojun Zhou¹ and Shungen Xiao^{1,2}

¹ School of Mechatronic Engineering and Automation, Shanghai University, Shanghai, China.

² School of information, Mechanical and Electrical Engineering, Ningde Normal University, Ningde, China.

*Corresponding author e-mail: 544824964@qq.com

Abstract. In this paper, the impact dynamics and nonlinear finite element method are analysed systematically. The static analysis of the test platform was carried out, and the result of the static analysis showed that the stress concentration existed in the connector and it had a large strength margin. Modal analysis is carried out on the test platform. Through modal analysis, the natural frequency of the test platform is obtained. The impact load response is determined to be the category of complex impact by comparing the impact loading time of the hammer system.

1. Introduction

Impact refers to a sudden change in the state of a system, such as acceleration, velocity, displacement and other physical phenomena, when the system is subject to external excitation for a very short time (usually measured in milliseconds). When the system is stimulated by transient aperiodic excitation, impulse excitation or transient step excitation, then the system will have a change of state, which is corresponding to the impact response of the excitation [1]. The causes of impact and vibration are different, and it has the following characteristics. Non-periodicity: since the impact is usually completed in a very short time, its load cycle is not periodic like vibration. Intensity: from the perspective of impact energy, the impact process is abrupt and drastic energy release, energy transfer and energy conversion. Transient: the process of impact formation and transmission is unsteady and discontinuous, and the duration of the impact process is short. Therefore, impact is transient.

The impact also causes the device to cause natural frequency response and forced vibration, which causes damage or even failure to the strength and performance of the structure. If one part of the structure is suddenly subjected to the impact load, the overall structure will show the impact response, and will reach the maximum in a relatively short time. The dynamic response of the structure is generally manifested as the deformation of the structure. This deformation quantity is not fixed and will change continuously with time. Such unpredictable changes may cause the destruction of the whole structure, or local penetration or fracture of the structure.

In practice, the impact wave is not regular, but in the real simulation process, some specific impact can be represented by the ideal regular wave. For the impact load of different waveform, the system will have different dynamic response. The response of the structure under the impact load mainly depends on the material parameters, geometric dimensions, load duration and load peak, so the dynamic response of the structure is very complicated.



Impact is generally divided into simple impact and complex impact. The maximum response of a structure under impact load is related to the natural frequency f_n (or natural period) of the structure and the duration of impact τ . When $T_n < \tau$, the maximum response of the structure is uncertain, which may exceed twice the peak value of the impact wave. When $T_n > \tau$, the impact response is much weaker. Therefore, when $T_n < \tau$, the impact can be called complex impact, while it is called simple impact [2].

Under the impact load of the structure, impact dynamics mainly studies the dynamic response of the structure and the transmission of stress waves. For the dynamic response under impact load, the research content is the deformation of structure and the relationship between fracture and time, ignoring the transfer process of impact response between regions. For the stress wave under the impact load, it mainly studies the transfer problem of the impact response in the region and the local disturbance problem, and studies the dynamic response as a process.

2. Finite element model establishment

In the finite element modeling of the structure, the appropriate element type and model size are selected on the premise of ensuring the reliability of the calculation results. The accuracy of the calculation results and the size of the model are contradictory quantities. In order to complete the finite element analysis with the least resources, consideration must be taken [3].

2.1. Mesh

Because the impact time of the hammer system is 0.2s. It is appropriate to make the impact load occur within the span of 120 units. The height of the test platform is 1197mm, and the minimum grid size is estimated to be 9.975mm by calculation. Therefore, the minimum grid size is set to be 10mm to divide the test platform [4].

2.1.1. Unit grid. The test platform is divided into grids by the combination of manual section method and automatic subdivision method, as show in Fig.1.

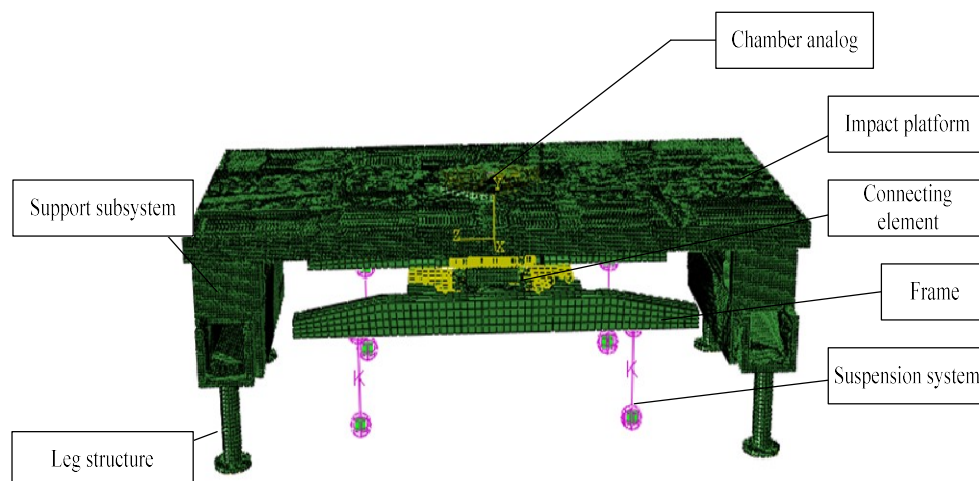


Figure 1. The finite element model of the test platform.

2.2. Material definition

Material definition can simulate most linear and non-linear engineering materials, such as rubber, metal, soil etc. This model mainly adopts conventional metal engineering materials [5]. Detailed engineering materials of the test platform are shown in table 1.

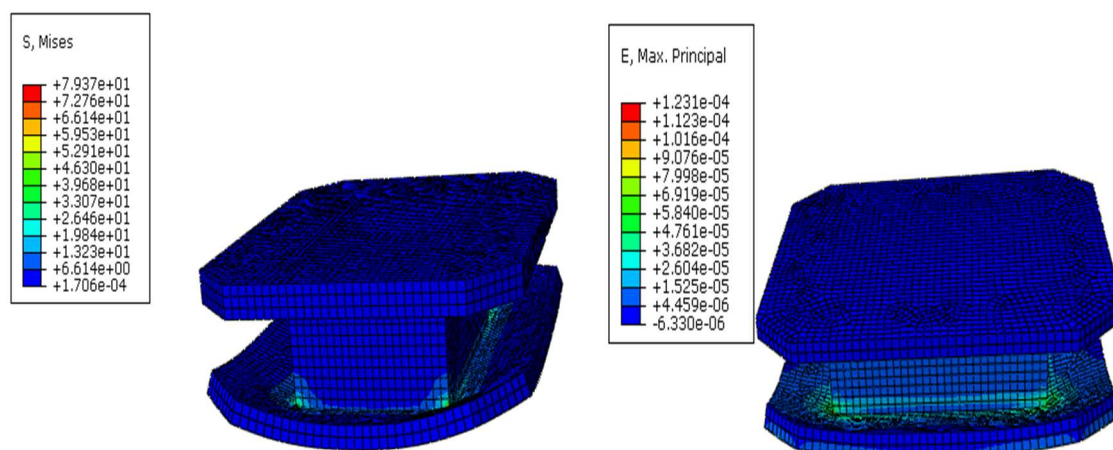
Table 1. Detailed performance parameters of all parts and components.

Name	Material	Modulus of elasticity (MPa)	Poisson's ratio	Density (t/mm ³)	Ultimate strength(MPa)	Allowable stress(MPa)
Chamber analog	Alloy steel	2.07E+05	0.3	7.8e-9	980	557
Impact platform	Alloy steel	2.07E+05	0.3	7.8e-9	980	557
Connecting element	Carbon structural steel	2.07E+05	0.27	7.8e-9	390	160
Frame	Joist steel	2.07E+05	0.29	7.8e-9	885	490
Support subsystem	Carbon structural steel	2.07E+05	0.27	7.8e-9	390	160
Leg structure	Carbon structural steel	2.07E+05	0.27	7.8e-9	390	160

3. The results of the analysis

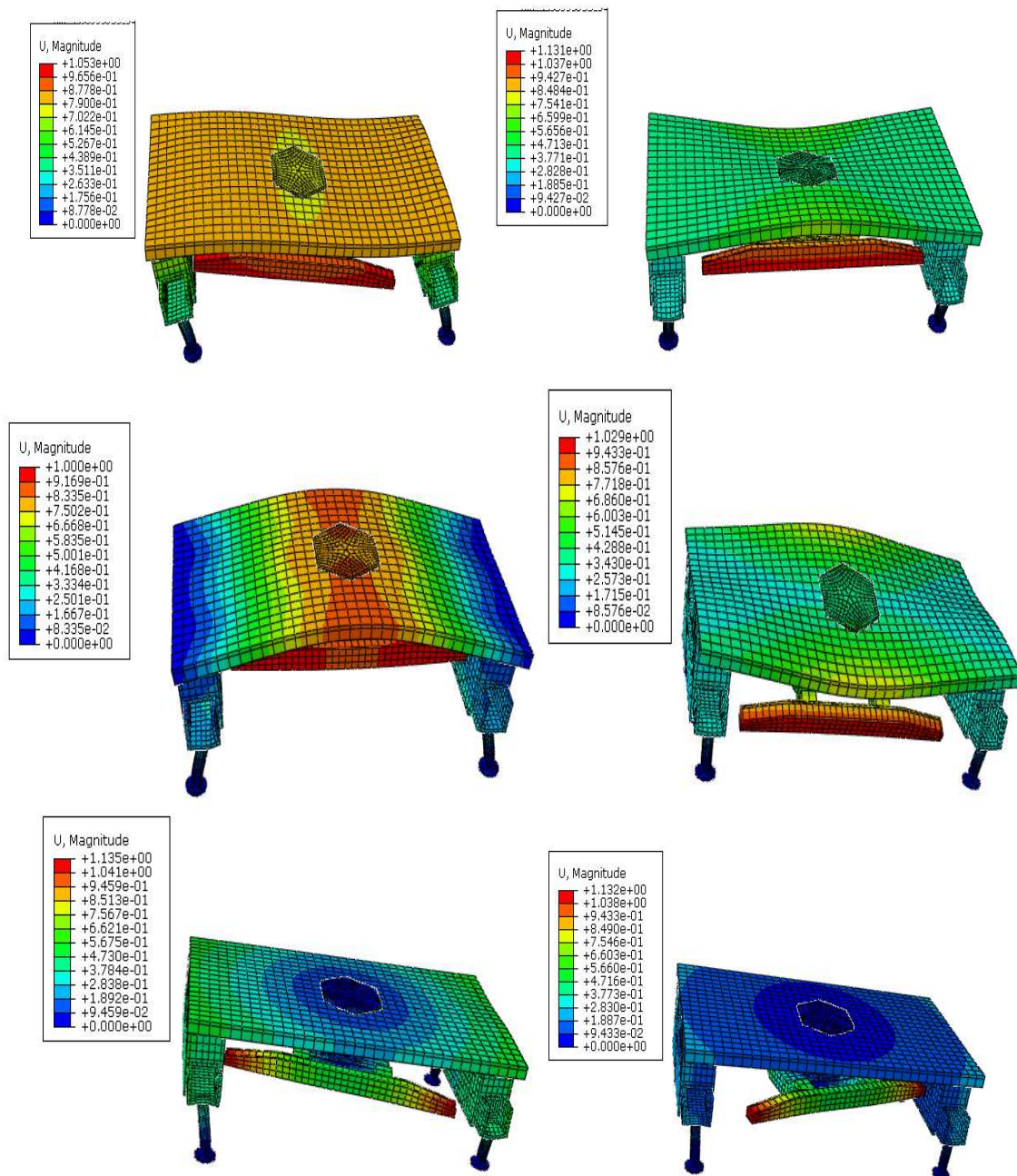
In the non-test state, the entire test bench is supported by the connecting element and the frame, so it is necessary to carry out finite element analysis of the connecting element. When the connecting element bears the weight of the impact platform and the supporting subsystem, the distribution of stress and strain and performs the strength check calculation.

The stress-strain cloud diagram of the connecting parts is shown in Fig.2. By analyzing the stress-strain cloud diagram of the connecting parts, it can be known that the stress concentration phenomenon exists in the connecting parts, and it is mainly concentrated between the lower end face and the middle support plate of the connecting parts. The maximum stress value is 79.37MPa and the maximum strain is 0.000123. The strain distribution position and stress distribution are basically the same [6].

**Figure 2.** Stress-strain diagram of connecting element.

For a model, as long as its structure and constraint conditions are determined, its mode is also determined accordingly, and the model's mode is calculated without considering the force of the model. For modal analysis, only the four supporting structures of the test platform need to be restrained.

There are many different methods for solving eigenvalues in modal analysis, among which Transformation method, Tracking method and Lanczons method are the most widely used. Lanczons method is very efficient to solve the eigenvalue of large matrix, which simplifies the solving process. In this paper, the modal analysis of the impact test platform is carried out by the Lanczons method. The first 8 natural frequencies and the corresponding mode of vibration of the test platform are extracted because the low order natural frequency and the corresponding mode have great influence on the dynamic characteristics of the structure, as show in Fig.3.



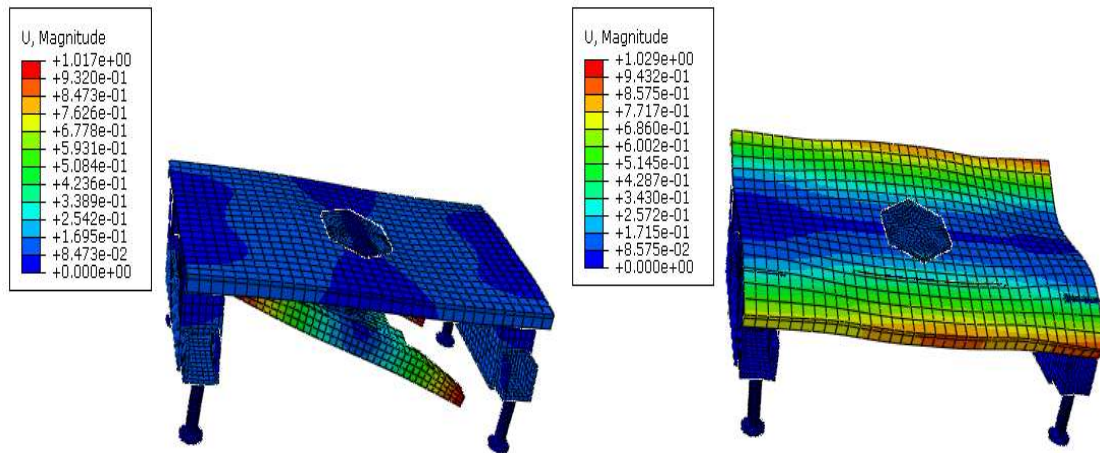


Figure 3. The first 8 natural frequencies of the test platform.

The frequency of the test platform, as show in table 2.

Table 2. The frequency table of the test platform.

Order number	Frequency(HZ)	Modal characteristics
1	27.837	Lateral bending
2	34.705	Reverse bending
3	45.919	Plane bending
4	47.078	Second order lateral bending
5	52.033	Frame bending
6	69.327	Frame torsion
7	76.220	The frame bends sideways
8	87.463	torsional bending

According to the first frequency (27.837HZ), the inherent period of the test platform is 0.036s, and the impact load duration is 0.2s. By comparison, it can be found that the inherent period of the test platform is less than the duration of the impact load. Therefore, the impact studied in this paper is a complex impact category.

4. Conclusion

Modal analysis is carried out on the test platform. Through modal analysis, the natural frequency of the test platform is obtained. The impact load response is determined to be the category of complex impact by comparing the impact loading time of the hammer system.

Acknowledgments

This work was financially supported by Education and Research Project for Young and Middle-aged Teachers in Fujian Province (Grant No. JT180597), Young teacher special project of Ningde Normal University (Grant Nos. 2017Q104 and 2018Q102), Major training project of Ningde Normal University (Grant No. 2017ZDK18).

References

- [1] Qiang Yu, K. Watanabe, T. Tsurusawa, M. Shiratori, M. Kakino and N. Fujiwara, "The examination of the drop impact test method," The Ninth Intersociety Conference on Thermal and Thermomechanical Phenomena In Electronic Systems (IEEE Cat. No.04CH37543), Las Vegas, NV, USA, 2004, pp. 336-342.

- [2] E. H. Wong, K. M. Lim, N. Lee, S. Seah, C. Hoe and J. Wang, "Drop impact test - mechanics & physics of failure," 4th Electronics Packaging Technology Conference, 2002., Singapore, 2002, pp. 327-333.
- [3] Jianxu Shi, David Chopp, Jim Lua, N. Sukumar, Ted Belytschko, Abaqus implementation of extended finite element method using a level set representation for three-dimensional fatigue crack growth and life predictions, *Engineering Fracture Mechanics*. (2010) 2840-2863.
- [4] Logan D L. A first course in the finite element method, forth ed. Cengage Learning, 2007.
- [5] Djerassi. S, Three dimensional, one point collision with friction , *Multi-body System Dynamics*, 2012, 27(2), pp. 173-195.
- [6] T.E. Bruns, Zero density lower bounds in topology optimization, *Computer Methods in Applied Mechanics and Engineering*. (2006) 566-578.