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

Modal Analysis of Refrigerator Compressor Shell Based on ANSYS Workbench

Lei Song [a], Lanqing Zhang, Shen Zhang, Tongyu Lan

College of Mechanical and Electronic Engineering, Shandong University of Science and Technology, Qingdao, 266590, China.

[a]songley2017@163.com

Abstract: The 3D model of compressor shell is established by using Pro/E, then importing into finite element analysis software ANSYS Workbench, the modal analysis is carried out to obtain the natural frequencies and the vibration-made vectors of the first 6 orders. The modal analysis results show that the working frequency of the compressor is much less than the first order natural frequency, which will not cause the shell resonance. The maximum deformation of the shell is at the support of the machine foot, which provides a theoretical reference for the optimal design of the compressor.
Keywords: compressor shell; modal analysis; ANSYS Workbench

---

1. INTRODUCTION

Taking a certain type of refrigerator compressor as the research object, this is a single rotor rolling piston compressor, also known as a fully enclosed reciprocating compressor [1]. Compressor is the core of refrigerator, and also the source of vibration and noise. The noise and vibration produced by the compressor will radiate through the shell, so the shell design plays an important role in reducing the vibration and noise of the compressor [2]. In this paper, the modal parameters of the compressor shell are obtained by using the finite element software for modal simulation analysis of the shell, which provides a theoretical basis for improving the noise radiation of the shell.

2. 3D SOLID MODEL ESTABLISHMENT

Before the modal analysis of ANSYS Workbench, the three-dimensional solid model of compressor shell is built by Pro/E. In order to improve the accuracy of simulation, the model is built according to the prototype of compressor shell, without too much simplification [3]. The model is shown in figure 1.

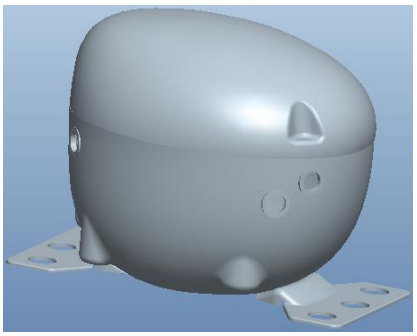


Fig. 1 3D solid model of compressor shell

3. FINITE ELEMENT MODEL ESTABLISHMENT

The 3D model established by Pro/E software can be seamlessly connected with ANSYS Workbench. The shell 3D model can be directly saved as .stp format file and imported into the finite element simulation software Geometry module for modal analysis [4]. The material of compressor shell was set up as cast iron, density 7850kg/m^3, Poisson's ratio 0.3, and elastic modulus 200GPa. Enter the Modal module to mesh it, choose the tetrahedron mesh division mode [5], the cell size is set to 10 mm, The compressor shell consists of 12637 units, 25588 nodes, and the finite element model as shown in Figure 2.

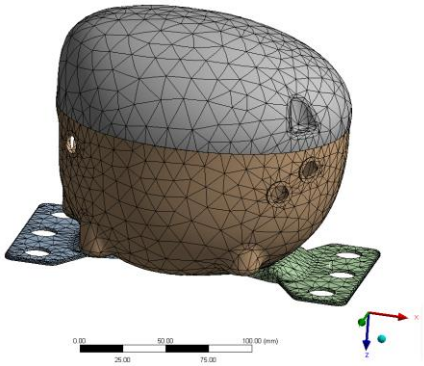


Fig. 2 Finite element model of compressor shell

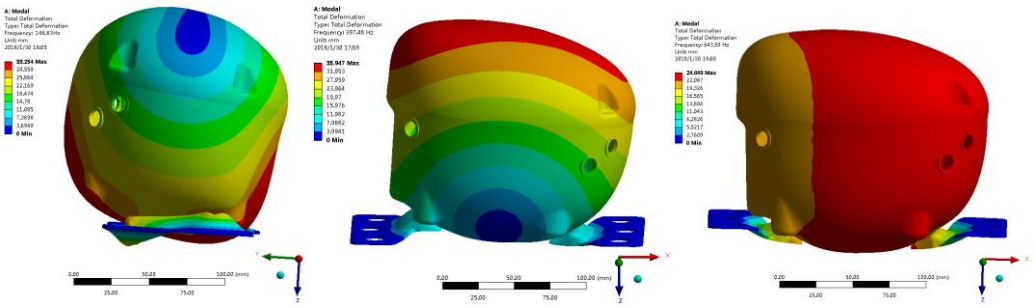
4. BOUNDARY CONDITION AND SOLUTION

The compressor is fastened to the supporting plate of the refrigerator by bolts at the foot of the machine, so in the finite element simulation of the compressor, the displacement of the compressor foot is completely restrained, so that the boundary conditions of load and constraint are consistent with the actual situation.

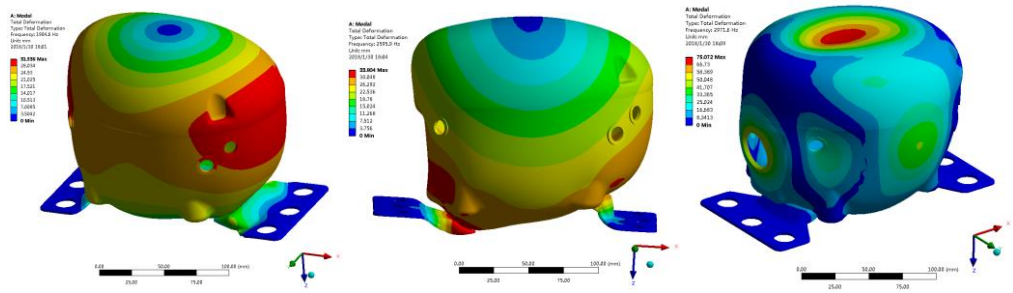
In the practical analysis, the low order natural frequency and its modal mode have great influence on the dynamic characteristics of the structure, so this paper only analyzes the first 6 order modal characteristics of the shell. The first 6 natural frequencies of the compressor shell are shown in Table 1, and the first 6 modal modes are shown in figure 3.

Table 1 Natural frequency of compressor shell

Modal order	Natural frequency/Hz
1	146.83
2	397.48
3	643.59
4	1904.6
5	2595.9
6	2971.6



(a)The first modal mode (b)The second modal mode (c)The third modal mode



(d)The fourth modal mode (e)The fifth modal mode (f)The sixth modal mode

Fig. 3 Modal mode of compressor shell

5. MODAL ANALYSIS RESULTS

The rated speed of compressor motor is 3000r/min, and the rated working frequency is 50Hz. From table 1, we can see that the first six natural frequencies of the shell are all large, the first natural frequency is 146.83Hz, far greater than the working frequency 50Hz, so the shell will not resonate. The first mode is the shell rotating around the X axis; the second mode is the shell rotating around the Y axis; the third mode is the shell displacement along the Z axis; the fourth mode is rotating around the Z axis; the fifth mode is the bottom deformation of the shell, especially the deformation of the machine foot; and the sixth mode is the concave and convex deformation around and the top of the shell. The dangerous position of the compressor shell is the support of the machine foot. It should be paid more attention to when the structure is designed.

6. CONCLUSION

In this paper, the 3D solid model and finite element modal analysis of compressor shell are carried out by using Pro/E and ANSYS Workbench software. The first six natural frequencies and the corresponding modal modes of the compressor shell are obtained by modal analysis. The results show that the natural frequency is generally larger than the working frequency 50Hz of the compressor, the compressor does not produce resonance, and the vibration modes are mainly the rotation and displacement deformation of the shell and the concave and convex deformation indicated by the shell. The maximum deformation appears at the support of the machine foot, which provides reference for the analysis of the compressor's dynamic characteristics.

REFERENCES

[1] Wang Fumao, Song Lei, “Dynamic Analysis and Test of 2-DOF Vibration Isolation System of a Refrigerator Compressor”, Noise and Vibration Control, 2017, Vol. 37 (3), p93-95

[2] Wang Xiaolong, “Vibration characteristics Research of Refrigerator Compressor”, Hefei University of Technology, 2014

[3] T. Wollmann, N. Modler, M. Dannemann, A. Langkamp, “Design and testing of composite compressor blades with focus on the vibration behaviour”, Composites: Part A. 2016, p1-7

[4] Liu Changling, Chen Jianyi, Li Qingping, “Modal and Harmonic Response Analysis of Crankshaft of a six Cylinder Compressor by ANSYS”, Fluid Machinery, 2012, Vol. 40 (8), p17-21, 26

[5] Lv Duan, Zeng Dongjian, “Finite Element Modal Analysis of V8 Engine Crankshaft Based on ANSYS Workbench”, Machinery Design & Manufacture, 2012, Vol. (8), p11-13