Modal Analysis of Thrust Coupling Based on ANSYS Workbench

Changkun Han*  
School of Beijing University of Chemical Technology, Beijing, China  
*Corresponding author: hanchangkun@yeah.net

Abstract. Through nx12.0, a solid model of a thrust coupling is established, which is transformed into a finite element model by using workbench 19.0, and modal analysis is carried out. The sixth order natural frequency and critical speed of the thrust coupling are obtained, and the first six order vibration figures are obtained, which clearly shows the natural vibration characteristics of the structure, and at the same time, it is the vibration analysis and structural optimization of the thrust coupling. The design provides a theoretical basis.

1. Introduction  
In recent years, the number of accidents caused by mechanical vibration is common. Couplings are the main connecting parts of the shafting system. Abnormal vibrations of the coupling will seriously affect the safe operation and production efficiency of the equipment. When the coupling fails, it will cause the production line to stop production, even huge economic losses. Modal analysis is one of the scientific methods to study the dynamics of structures. The inherent vibration characteristics of vibrating members can be obtained. Each mode has its fixed frequency, damping ratio and mode shape.[1]  

Based on the above reasons, this paper takes the thrust coupling installed on a compressor as the research object, and simplifies the model to establish a solid model. The ANSYS Workbench 19.0 platform is used to perform finite element analysis on the model. Analysis, the natural frequency and mode shape were obtained, the weak parts in the operation of the coupling were determined, and credible suggestions were put forward.

2. Thrust coupling  
Couplings are important connecting parts of the transmission system of the mechanical system. They play a role in transmitting force and torque, compensating shaft displacement, damping and absorbing vibration, safety protection, improving transmission efficiency, etc. There are many types, which are widely used in aerospace, ship, locomotive, automobile, petrochemical and other rotating machinery and equipment.[2]  

The analysis object of this article is the thrust coupling adopted by a company. As a type of flexible coupling, it is similar to the diaphragm coupling, taking into account the excellent performance of the diaphragm coupling: large transmission power, high speed, It has the advantages of vibration reduction, high safety protection efficiency, etc. At the same time, it can withstand large axial thrust and resist axial vibration.
3. Solid model of thrust coupling
The establishment of this model relies on the powerful and convenient modeling functions of NX12.0, especially for complex component models, which can effectively shorten the modeling cycle, improve the modeling quality, and assemble according to the component drawings to obtain a complete thrust coupling model. Relying on the convenient ANSYS-NX12.0 conversion interface and using Parasolid format files to achieve accurate data conversion, NX12.0 and ANSYS software have complementary advantages, while fully displaying the modeling advantages of NX12.0 software, and can use ANSYS Workbench 19.0 software Analysis. [3]

According to the drawings of the thrust coupling, each part model is established in NX12.0, and the assembly is made according to the contact relationship between each part to obtain a complete model of the thrust coupling.

![Figure 1. Solid model of thrust coupling](image)

4. Finite element model of thrust coupling
This paper uses ANSYS Workbench19.0 platform to perform modal analysis of thrust coupling. ANSYS Workbench 19.0, as the most advanced CAE software, provides users with simulation modules including: structure, fluid, electromagnetic, heat transfer, and other fields. It is the industry's most advanced engineering simulation technology integration platform, with intuitive and friendly interface, convenient pre-processing and post-processing functions, and its extensive solution functions.

4.1. Defining component and material properties
After importing the model into Workbench 19.0, it is necessary to use the layer unit solid45 for the diaphragm assembly and the solid186 unit for the remaining parts according to the existing model, and adjust the contact relationship between the parts according to the corresponding contact method.

Define the material properties. For this model, the diaphragm material is 65Mn, the elastic modulus is 207GPa, the Poisson's ratio is 0.35, and the density is 7850 kg/m$^3$. The other components are system default model structural steel, the elastic modulus is 200GPa, the loose ratio is 0.30 and the density is 7850 kg/m$^3$.

4.2. Meshing
As the main link of finite element analysis, grid division can best reflect the idea of finite element. The quality of the web site not only affects the efficiency of model analysis, but also directly affects the accuracy of analysis results. Therefore, according to the existing hardware, without affecting the accuracy of the calculation results, the method of dividing the mesh can be appropriately selected to save calculation time.

The multi-region method is adopted for this model. This method controls the surface of parts to generate tetrahedral and hexahedral meshes. The model after the mesh division is shown in Figure 2, and the mesh model of the diaphragm is shown in Figure 3.
4.3. Impose constraints
In the modal analysis, the displacement constraints in the X, Y, and Z directions and the rotation constraints in the X and Y directions of the left and right end faces of the two half-couplings need to be restricted based on the movement characteristics of the components so that they can only rotate around the Z axis. That is, it can only rotate around the axis.

4.4. Modal parameter setting
Workbench 19.0 provides five modal solution methods: Direct, Iterative, Unsymmetric, Supernode, and Subspace. [4] By comparing the comparison of each method and analysis of its advantages. This paper uses the subspace method (Subspace), which is suitable for solid element models, requires less memory, and is suitable for extracting fewer modal orders.

When performing a modal analysis of a thrust coupling, it is not necessary to obtain a complete natural frequency and mode shape. It is known that the maximum speed of the thrust coupling under operating conditions is 946r/min, which is in the low speed range. The effect of low-order frequency on vibration is more significant, so the sixth-order modal frequency and mode are set to be solved.
4.5. Modal solution analysis
The 6th order mode vibration frequency and vibration model are obtained by solving the solver. The following is the mode shape of the 6th order mode. The Campbell diagram of the thrust coupling is obtained under the same conditions.

![First mode shape](image1.png)

**Figure 4.** First mode shape

![Second mode shape](image2.png)

**Figure 5.** Second mode shape.

![Third mode shape](image3.png)

**Figure 6.** Third mode shape
Figure 7. Fourth mode shape

Figure 8. Fifth mode shape.

Figure 9. Sixth mode shape
Through the analysis of the above mode shapes and campbell, the natural frequency of each order, the maximum displacement, and the calculated critical speed can be obtained.

| Order | Natural frequency (Hz) | Maximum displacement (mm) | Critical speed (rad/s) |
|-------|------------------------|---------------------------|------------------------|
| 1     | 32.54                  | 0.83                      | 204.43                 |
| 2     | 617.53                 | 0.72                      | ——                     |
| 3     | 792.44                 | 1.52                      | ——                     |
| 4     | 792.62                 | 1.50                      | ——                     |
| 5     | 849.60                 | 1.13                      | ——                     |
| 6     | 849.99                 | 1.12                      | ——                     |

Through the analysis of the first 6 order models, the 1st order mode is a rigid body mode, and the remaining 5th order modes are torsional modes. Among them, the 3rd and 4th order vibrations have the largest deformation, and 1st, 4th and 6th are forward. The second, third and fifth are counter-precessions. When the angular velocity is 204.43 rad/s, the modal frequency is the same as the operating frequency, and resonance occurs.

5. Conclusion
Modal analysis is the most basic linear dynamics analysis. Through the analysis of the natural frequency and shape of the thrust coupling, it clearly reflects that the deformation of the coupling flange is greatly deformed, and the structure should be carried out. Further strength check and optimization design to avoid the large impact of resonance. It also provides a reliable basis for other parts in other dynamic analysis and evaluation.

Figure 10. Two or more references.
References
[1] Liang Jun, Zhaodeng Feng. Summary modal analysis method [J]. Modern Manufacturing Engineering, 2006 (08): 139-141
[2] Zhou Mingheng. Coupling Selection Manual [M]. 2001.
[3] Yi Dengli, Zeng Hong, Wang Xinfeng, et al. Model import problem in ANSYS analysis of UG modeling of aluminum alloy wheels [J]. Journal of Liaoning University of Technology (Natural Science Edition), 2008, 28 (1): 36-38.
[4] G.R. Mettam, Chen Yanxia, Chen Lei. ANSYS Workbench engineering applications and selected cases [M]. 2012.