The Application of Mechanical Structure Dynamic Design in Mechanical Product based on ANSYS

Bohai He¹*, Zhengkun Zhao²
¹Zhejiang Industry Polytechnic College, Shaoxing, 312000, China
²Beijing University of Chemical Technology, Beijing, 100029, China
*Corresponding author (E-mail: hebohai@163.com)

Abstract
With the development of science and technology, the advanced analytical methods in mathematics and computer have already been used in all aspects of industrial design, which have raised the productivity and reduced the production cost. The traditional mechanical product design process needs experience judgment as well as physical verification, and it extends the time limit for a project, reducing the competitiveness of the product, therefore the mechanical structure dynamic design software has become the research hotspot, how to simulate the physical condition and dynamic physical situation in the design has become the main direction. Now there is a lot of mechanical simulation software, of which the mechanical structure simulation software based on the finite element analysis has occupied an important position in the design and development of mechanical product. Finite element analysis software headed by ANSYS software can significantly improve the progress of the mechanical dynamic structure design and improve the yield of the product, and it has played an important role in the field of mechanical design. Beginning with a theoretical basis of ANSYS simulation software, this paper expounds the theory of finite element analysis and introduces the mechanical structure dynamic design and optimization process based on ANSYS, finally the example of mechanical structure dynamic design based on ANSYS is given.

Key words: ANSYS, Mechanical structure, Dynamic design, Finite element analysis, Mechanical product

1. INTRODUCTION

The traditional mechanical design method is to combine with the experience and mathematical physics knowledge, and produce real object to verify and improve. This method is old and low productivity, and it also cannot verify the dynamic characteristics and stability of mechanical structure. With the development of mathematical theory and computer software, the dynamic design of mechanical structure has become the industrial base fusing various kinds of high-level knowledge. In the design of the mechanical structure, a lot of structure is changing, conducting mechanical motion under large environment, therefore the mechanical stress, temperature, friction and impact pressure in their place will be changed accordingly, these physical factors is impossible task for ordinary designers, so the software owning the ability of dynamic design needed to be adopted to conduct aided design analysis and study the dynamic characteristics of the mechanical structure and carry out the subsequent design of optimizing institution. Of which, finite element analysis software headed by ANSYS has played an enormous role in mechanical structure, which has simplified the design, making it more reasonable and efficient, and also saved the cost. Some researchers have studied slot comparison analysis of mechanical seal ring based on ANSYS, and the optimized design is given (Liu, 2016). ANSYS also can conduct mode analysis in the study of mechanical shaft parts, evaluating performance (Lee, 2015). ANSYS also can analyze electrodynamics research and mechanical stress analysis under the circuit situation of high pressure and large cross-section cable, improving the stability of the high pressure and large cross-section cable (He, 2016). The purpose of this research is to analyze the use of ANSYS in dynamic design of mechanical structure and study the application of ANSYS in mechanical product design, finally giving the application example of ANSYS.

2. THE FINITE ELEMENT ANALYSIS THEORY

The finite element method has combined the latest research results of the mathematics and mechanical engineering subjects (Krynkin, 2015). In the 1860s, Tuna and Clausius have put forward limited triangulation method of solving plane stress, and this is the birth of the finite element analysis. As a kind of numerical method, the finite element analysis theory combined with the physical model and parameters can solve large-scale problem of physical numerical solution, so the theory of finite element analysis has been widely used in various fields of mechanical design (Vasilev, 2015).

The theoretical thought of finite element analysis is to make concrete structure into a small unit, and there has same node between units, the shape and size of the unit relate to the accuracy required by calculation (Zhao,
2016). After dividing the unit, then the unit is conducted physical analysis by the given initial conditions, the obtained result can be taken as the initial condition of other small unit to conduct physical operation. After the traversal of all units, physical properties analysis of the whole structure can be produced. So the finite element analysis method is a method of numerical approximation, the structure calculation is closely related to unit division, if the division is proper, the calculation results of finite element is the same with actual physical result.

The process of finite element analysis can be divided into the following five steps:

Step 1, to divide the unit and conduct interpolation. In this step, conducting division needs to combine the physical characteristics of the structure and geometry, when dividing sometimes dense and sometimes sparse.

\[ q = Ca \]  \hspace{1cm} (1)

Step 2, Analyze the unit. In this step, combining physical field model, physical properties calculation of unit can be operated.

\[ K = \int B^T DBdV \]  \hspace{1cm} (2)

Step 3, The equations after unit analysis. In this step, sort out the analyzing result of last step, list the solving equations.

\[ f = Kq \]  \hspace{1cm} (3)

Step 4, Solving the equation. The main problem in this step is to choose which algorithm for solving equation set, obtaining transfer parameters

Step 5, To calculate physical parameters. In this step, according to transfer equation of previous step, combining with the final physical parameters, the required physical properties is calculated out. Of course, in the process of finite element analysis, there are also some matters needed to be paid attention to, for example, the number of unit should not too much, if necessary, increase the number of units and optimize the calculation cost; To select appropriate unit division method and the solving method; the physical limits features of structure should be also considered, for instance the stress has a limit, it will deformation after exceeding; In order to avoid difficulty analysis of equation set, the periphery conditional value which is easy to find out should be given.

The unit division in the theory of finite element analysis is very important, good unit classification can simplify the calculation and save the computation time. When dividing the unit, the unit should not be too much or too little, the appropriate number should be chosen, with rare in regular place and more in un-regular place; In general, triangle unit division method is used to divide, and if the result is bad, polygon should be selected to divide; The dividing way of unit is also useful in optimization calculation, such as mixed units division, as for complex structure constituted by multiple independent structures, aiming at different structure, carrying out division which conforms to the its physical characteristics can accelerate the operation.

3. THE MECHANICAL STRUCTURE DYNAMIC DESIGN BASED ON ANSYS

In this chapter, firstly we carry out modal analysis of mechanical structure, providing a solid theoretical foundation for dynamic design of the mechanical structure; Adopting ANSYS for structural dynamic design of mechanical structure, there are three advantages, firstly, it can effectively eliminate adverse vibration phenomena which damaging mechanical structure appeared in the process of mechanical movement, such as resonance; secondly, it provides reliable dynamic kinetic parameters for dynamic mechanical institutions optimization; finally, this analysis method make the design more accord with the need of engineering (Zeng, 2015).

3.1 The modal analysis of mechanical structure

Firstly, the concept of the modal analysis is introduced, namely, within a certain amount of time course, the parameters of the vibration model which is used to extract and analyze dynamic structure system. According to the different methods of implementation, the modal analysis can be divided into two kinds including experimental modal analysis which using the experimental way to implementation and computational modal analysis which using calculation to implementation. The two methods both have their own advantages and disadvantages, and the results obtained by experimental method are precise and small error, in agreement with final design, but this method requires to do a large number of experiments, with large investment and long cycle. The computing implementation scheme has the characteristics of fast speed and small investment, while in general low precision, it is often used before designing product. The general structure design can adopt these two kinds of design schemes respectively or at the same time according to the specific situation.

3.1.1 The establishment of mechanical structure modal analysis model

The most important thing of establishing vibration model is how to extract and calculate vibration model parameters with smaller error. According to the finite element method, the whole machine structure can be
divided into several unrelated small modules, according to the interrelation between modules, and the structure is conducted harmonious constraint to construct scattered system intensity equation (Xiong, 2015), namely:

\[ \{R(t)\} = \{F(t)\} - [M]\{X(t)\} \]  (4)

Of which, [M] —— the largest accepted intensity sequences;
\{F(t)\} —— the damping strength sequence;
\{R(t)\} —— the sequence whose stress point changing with time;

The mechanical structure will appear vibration phenomenon which adverse to the structure in the process of dynamic use, according to the dynamic characteristics of structure, resonance frequency of structure is analyzed, and it is the basis of conducting mechanical structure dynamic analysis, the constructed structural vibration mode equation is shown as below:

\[ [M]\{X(t)\} + \sin \omega \{X_o\} = 0 \]  (5)

Of which \( \omega \) - the vibration angle in unit time
\{X_o\} - the node amplitude of initial vibration

As for the extraction of parameters in the dynamic mechanical structure model, the following plans can be chosen as follows:

Power Dynamics. Power Dynamics is suitable for larger model, and space decomposition method is adopted to extract and calculate out the characteristic value of each subspace. According to the characteristic value, analyze the changing situation of structure, the method has the highest efficiency, and it can be finished with the aid of sparse matrix solver, with EQSLV order appointing.

Reduced/Householder. The reduced householder method adopts reduced householder to solve problem, compared with the method of power dynamics, the speed is faster, but as a result of the limitation of reduced householder coefficient matrix, the precise value of this method is very low, not suitable for actual manufacturing.

QR Damped. QR Damped mainly carries out analysis on damping of dynamic mechanical structure, the larger the damping, the lower vibration intensity, and the vibration attenuation will more intensive, according to the relationship between the damping and vibration, the vibration parameters is extracted, this method can solve the problem soon, with high computing efficiency (Liu, 2016).

3.1.2 The modal analysis results of dynamic mechanical vibration model

Through the modal analysis, we obtain the results of the mechanical structure dynamic vibration model. In this section, the parameter extraction methods including Power Dynamics and QR Damped are used to obtain approximate solution of modal analysis. In the process of the whole analysis, there are a lot of the resonant frequencies in the mechanical structure, but the most serious damage to structures is the previous order, therefore, we analyzes the first 8 orders of modal analysis in this paper, See it in Table 1.

<table>
<thead>
<tr>
<th>order</th>
<th>1 order</th>
<th>2 order</th>
<th>3 order</th>
<th>4 order</th>
<th>5 order</th>
<th>6 order</th>
<th>7 order</th>
<th>8 order</th>
</tr>
</thead>
<tbody>
<tr>
<td>frequency</td>
<td>0.96</td>
<td>0.98</td>
<td>1.23</td>
<td>1.92</td>
<td>2.10</td>
<td>2.42</td>
<td>2.20</td>
<td>2.52</td>
</tr>
</tbody>
</table>

3.1.3 The modal analysis result conclusions of dynamic mechanical vibration model

It can be seen from table 2.1 that when the order of the vibration equation is the corresponding order number, the mechanical structure can produce resonance under this vibration frequency, the resonance can make the whole structure intensify, the mechanical structure will produce irreparable damage, causing great harm and economic loss. Therefore, when conducting mechanical structure design, the resonant frequency of structure should be considered in order to improve the strength of the structure.

3.2 The basis of transient mechanical structural mechanics analysis

The machinery operating in full load will cause great impact on several key structures, and therefore this section will conduct instantaneous force analysis of machinery in the process of full load. By using the module in ANSYS system, the instantaneous force situation is analyzed, obtaining mechanical parameter of mechanic under the condition of full load (Greifzu, 2015).

3.2.1 The theory of transient mechanical structural mechanics analysis

This theory analyzes the force condition of structure over a period of time in order to determine the stress impact of key parts of machinery in the full load cases, and also it can analyze the response of machinery’s each movement module under the condition of stress. The basic equations of mechanics key stress:

\[ MY(t) + CY(t) + KY(t) = G(t) \]  (6)

Of which, M - the largest quality
C - damping sequence  
K - the largest bearing sequence  
Y(t) - the accelerated sequence of key structure  
G(t) - the largest amount of shock  

As for the transient structure stress, there are two main analyzing methods: Senior definite integral method and fuzzy matrix addition method.  

Senior definite integral method is a kind of relatively simple algorithm, and the principle is to conduct integral on the value of G(t) in the right of formula 2.3, before conducting integral, the mean value of functional value should be found out. In order to reduce error, the mean value is conducted Gauss sum and then the sum adopts integral to operate.  

Fuzzy matrix addition method is a more complicated but more accurate algorithm, which obscures the value in the results of modal analysis, forming the fuzzy matrix, and then each value in fuzzy matrix multiplies the damping coefficient respectively and adds together, obtaining a single matrix, each value in the matrix is the maximum stress value of the key structure during the course of time.  

3.2.2 The instantaneous dynamic analysis method of mechanical structure based on ANSYS  
In ANAYS system, there are many methods of conducting transient analysis of mechanical structure, generally the senior definite integral method is used, and the following two are concrete realizing method:  

1. Sparse superposition method  
The sparse superposition method uses sparse matrix to calculate the instantaneous stress situation including the linear and nonlinear scheme, it is the most perfect algorithm of system.  
The advantages of sparse superposition method:  
1) The function is perfect, not only calculate the simple mechanical structure but also calculate the complicated mechanical structure  
2) There are two implementing methods including linear and nonlinear, and the two schemes are supplementary to each other, verifying the result of calculation and increasing the redundancy of computing.  
3) Using the sparse matrix, an analyzing calculation can get the corresponding force situation.  
4) The transverse, longitudinal and 45 degrees are allowed to appear in mechanical structure, meeting various effective loads.  

2. Non-complete reduced householder method  
Non-complete reduced householder method is a new implementing scheme, which makes the complex structures reduced order, after it falling into typical structure, each stress points are carried out analysis. The structure chart is complex, some stress points cannot be conducted reduced order, or after the order reduction will lead to great errors. Therefore when conducting reduced order, each stress point should be evaluated to determine the robustness of the stress point, so this is called non-complete method.  
The advantages of the non-complete reduced order method:  
1) the linkage analysis between acceleration and stress can be conducted  
2) The constant of time duration in the course of time can be ensured, easy to calculate the displacement of the mechanical stress points  
3) Small occupancy resources, damping type can be determined.  

When using senior definite integral method, the following aspects should be paid attention to: to determine the linear and nonlinear situation in the structure, the two cases are separated to calculate respectively; to determine appropriate time step, too long time step will cause too great calculation error and too small time step will make the calculation time too long, influencing the design efficiency; to determine the damping type, the damping influences the vibration of mechanical structures, the damping type which conforming to reality highly can reduce the error of the transient response of higher order resonance, making the convergence of vibration equation more rapidly.  

4. ANSYS DESIGN EXAMPLE  
ANSYS can conduct physical simulation, a 1000 N vertical load will be applied to catapult, as shown in Figure 1. The catapult is made from steel pipe, with 40mm external diameter, 10 wall thicknesses and 200GPa elastic modulus. The spring has 5N/mm stiffness.
Figure 1 The node of catapult and the application of spring simulation

The Figure 2 shows the stress analysis of beverage can in the study of nonlinear subsidence, and it indicates that the stress on inside wall of beverage can is greater than that of at the bottom in nonlinear subsidence. This gives the design of beverage can product guidance, that is, enhance the strength of the side wall. The Figure 3 shows that the shell unit is used to analyze the stamping part of the rebound effect.

Figure 2 The stress analysis of beverage can in the study of nonlinear subsidence

Figure 3 The shell unit analyzing the stamping part of rebound effect

The Figure 4 shows manifold pre-tight components (bolt fastening), in this application, there may be many challenges such as gasket and model size, the existence of the bolt, the connection between the parts and complex history of stress. The above analysis can guide the design of manifold pre-tight component and balance
the mechanical stress, as for place where has great mechanical stress, the method of strengthening design is adopted.

Figure 4 The stress analysis of manifold preloaded components

The Figure 5 shows the stress outline of circular prism. The expected result is the state of constant stress around circumference. Too little number of nodes will lead to the shortage of the simulation accuracy (pay attention to display the 8-node element results of stray concentration spots). Therefore in the design many nodes should be adopted to verify the design.

Figure 5 The stress outline of circular prism

The contacted definition allows defining the shell to shell or shell to the entity components (as shown in Figure 6). ANSYS mechanical simulation can adopt standard multi-point constraints (MPCs) to realize compatibility.

Figure 6 Shell to shell or shell to the entity components
5. CONCLUSIONS
In this paper, the theoretical basis for the finite element analysis of ANSYS software is analyzed in detail, and the mechanical structure dynamic design process based on ANSYS software is also introduced, and the factors influencing the design and simulation accuracy are analyzed, providing guidance advice for mechanical structure dynamic simulation. Finally some examples of the mechanical structure dynamic design based on ANSYS are provided in this paper, introducing the advantages of ANSYS aided design in mechanical products. According to the simulation results, the design can be optimized, avoiding the stress problem in application, etc. It can be concluded that in the design of industrial mechanical structure, the engineer should combine with the ANSYS software to optimize the dynamic design, and the mechanical products should be analyzed in detail in order to design mechanical products conforming to the application.

REFERENCES