Finite Element Analysis of Marine Weapon Lift Based on ANSYS

Wanshan Su

1 The 713th Research Institute of China Shipbuilding Industry Corporation, Zhengzhou, Henan, 450015, China
*Corresponding author’s e-mail: sws1986@126.com

Abstract. In order to improve the dynamic characteristics of a shipborne weapon elevator and reduce the adverse effects of harmonic vibration on the elevator under sea conditions, the finite element model of weapon elevator is established by ANSYS/Workbench, and the modal analysis, harmonic response analysis and static strength analysis of the elevator under dynamic load are carried out. The analysis results are obtained: The harmonic frequency is far less than the first natural frequency of the elevator under the external sea condition, so the system will not suffer from resonance damage and meet the strength requirements; the external excitation frequency of the elevator should be avoided from 20 to 40 Hz, and the deformation of the side plate in motion is the largest, but within the control range, it does not affect the docking of adjacent components. Improvement measures are put forward to increase safety factor and anti-vibration ability.

1. Introduction

Presently, with ammunition as the basic unit, the domestic ammunition storage and transportation system carries on its transportation through the bomb chain or magazine form. With the development of guidance technology, the task of long-range fire strike with large caliber naval gun is becoming more and more urgent. The ammunition storage and transportation system, whose basic storage unit is ammunition box, has been applied [1]. The closed missile box can effectively protect the electronic devices of guided ammunition and ensure the accurate strike effect of naval gun. During the battle, each layer of ammunition box needs transporting quickly to meet the continuous shooting, which puts forward higher requirements for the mission reliability of the weapon lift.

Due to the unfavorable marine environment, the instantaneous impact load is applied to the weapon elevator at all times. These loads act on the elevator in the form of the centrifugal force or the heave force [2], whose influence on the lift is much greater than that on the land condition. Therefore, it is of great significance to carry out finite element analysis of weapon lift to ensure the continuous strike ability of naval gun far from the sea.

2. Design of marine weapon lift

At present, most of the marine lifts are traction type, so the towing power chamber should be reserved on the upper deck. The vertical lifting of car is realized by the traction of wire rope. This lifting method not only has been widely used in the field of military and civil affairs, but also meets the requirements of use. However, the vertical well of the transmission scheme occupies a large space, and the deck also needs to reserve a power chamber, which has a great influence on the stealth of the whole ship weapon [3]. Therefore, the weapon lift needs to adopt more flexible transmission form.
The arrayed guide rail helical gear rack weapon lift has been put forward. The weapon lift is driven by the power form of "power-driven machine + planetary reducer + helical gear". The vertical well track adopts splicing rack and array guideway, which ensures the stability of the vertical motion of the lift through the constraint of the compound roller. The structure of the weapon lift is shown in Figure 1.

![Figure 1. Weapon lift structure](image)

The lifting object of the weapon lift is the cartridge box and the inside of the lifting car is the storage position of the cartridge box. The cartridge box is supported by roller bearing, and a set of locking devices are set up inside the car. After the cartridge box is in place, the cartridge box is fixed by the action of electromagnet. The integrated car adopts box frame, and the opening area on both sides can realize the movement of the bullet box between the adjacent docking parts. The integrated transmission system at the bottom of the car makes the lift structure more compact. When the lift is lifted, the main force component is helical gear, and the guide roller bears radial force and axial force. The farther the distance between the upper and lower guide rollers, the smaller the bearing force [4]. Therefore, the vertical guide roller of the car adopts the unilateral cantilever arrangement scheme, which can reduce the shaft size of the car to the greatest extent and provide more effective space for the ammunition cabin.

3. Theoretical basis of finite element analysis

The basic idea of finite element method is to discretize the structure, use finite elements to represent complex objects, connect elements through finite nodes, and then solve them by deformation coordination condition [5]. According to the principle of virtual work, the total potential energy $W$ of the elastic body includes two parts: work $T$ and $U$ done by external force.

Virtual work of external Force on Elastomer: $T = \{\delta\}' [F]'$

Virtual work of stress on Elastomer: $\int_{\Omega} \{\varepsilon\}' \{\sigma\} d\Omega$

According to the principle of Virtual work, $\{\delta\}' [F]' = \int_{\Omega} \{\varepsilon\}' \{\sigma\} d\Omega$, the stiffness matrix of element can be obtained after writing the expression of stress and strain into virtual displacement expression: $[K]' = \int_{\Omega} [B]' [D][B] d\Omega$.

If the stiffness of all elements is superimposed by the stiffness integration method, the stiffness matrix of the structure $[K]$ can be obtained.

After solving the stiffness matrix $[K]$, node force of structure $\{F\}$ can be expressed as follows:
\{ F \} = [K]\{ \delta \}

According to the classical mechanics theory, the dynamic equation of the object is expressed as follows:

\[ [M]\ddot{x} + [C]\dot{x} + [K]\{x\} = \{F(t)\} \]

According to this equation, the displacement of each node of the system and the stress and strain of the corresponding node can be obtained. Through the differential equation of vibration, the eigenvalues of the system and the natural frequencies of each order can be obtained. This paper uses the finite element software ANSYS/Workbench to analyze the modal and transient dynamics of the lift and carries out the statics analysis of the lift under dynamic load.

4. Elevator model processing
The structure of lift car is in the form of integrated frame. Before discretization of the model, the model which has little influence on the whole structure, such as standard parts, can be removed. At the same time, the chamfer, through hole, thread hole and other features existing in the model are ignored, which may lead to the difficulty of grid division and the large number of grid and affect the final calculation accuracy. The square support is used to simulate the roller support of the bullet box in the lift, which has no effect on the whole structure and is convenient to apply the load. The lifting powertrain (including helical gear) is simulated by cylinder. Because the shaft track is installed on the hull, this part of the model may not be imported for analysis. The finite element analysis of lift car is mainly carried out.

There is a welded structure between the parts of the car frame. The joint surface of the two parts is simulated by binding contact. The finite element model of the lifting car after discretization is shown in Figure 2.

![Lifting finite element model](image)

The lift adopts Automatic grid division method and second-order tetrahedral element. The finite element model has 134256 nodes and 41947 elements.

5. Modal analysis of lift
To facilitate the application of constraints and boundary conditions, the vertical direction of the lift car is set to z direction, and the direction of opening door on both sides is y direction. When the lift cantilever rises, four composite rollers constrain their freedom degrees of x and y. In the process of lifting and falling of helical gear, the helical gear restricts the freedom degree of z [6]. The first 6-order natural frequency of the resulting elevator is calculated as shown in Table 1 below, and the corresponding node mode is shown in Figure 3.
Table 1. First 6 natural frequencies of elevator.

| Order 1 | Order 2 | Order 3 | Order 4 | Order 5 | Order 6 |
|---------|---------|---------|---------|---------|---------|
| Natural frequency | 25.6 | 29.3 | 49.5 | 61.4 | 63.5 | 66.3 |

In modal calculation, the eigenvectors corresponding to each eigenvalue are all relative values, which are inconsistent with the actual calculated displacement. Therefore, the results of the modal analysis do not need to compare the magnitude of the deformation amount, and only the vibration relation of each component of the structure needs to be compared [7]. In the first mode, the side plate of lift deforms along a and x. In the second mode, the side plate of X direction deforms. In the third mode, the mounting plate of the roller seat has a large x-direction deformation. In the fourth mode, the lift is twisted with clockwise in the z direction. In the fifth and sixth modes, the lift is twisted with counterclockwise in z direction. By comparing the vibration shapes of each order, it can be seen that there is little difference in the vibration modes of each order of the lift, and the vibration of the mounting plate of the reducer and the compound roller seat is very small. Gear rack meshing and guide roller motion are on this mounting plate, so there is little vibration in the vertical motion of the lift. The stiffness of this part does not need to be increased again, and the main vibration of the structure is on the side plate. This part of vibration can be further evaluated in the subsequent harmonic response analysis.

6. Analysis of harmonic response of lift
According to the condition of Class 5 sea condition of the target ship, when the ship’s fin stabilizer is in operation, pitch period is 5 - 8 seconds, the rolling period is 8 - 13 seconds, and the hanging period is 5 - 8 seconds. The maximum frequency of vibration is 0.2 and the minimum is 0.125; the maximum frequency of rolling is 0.125 and the minimum is 0.077; the maximum frequency of swinging is 0.2
and the minimum is 0.125. Under the excitation of stage 5 sea condition, the minimum vibration frequency of the system is much larger than the excitation frequency, then the system failure under resonance will not occur. The modal analysis only obtains the relative motion of the system and it cannot calculate the exact displacement under external excitation. Therefore, it is necessary to further analyze the harmonic response of lifts. Through the analysis, the displacement response and anti-vibration characteristics of the lift under external excitation conditions can be obtained more intuitively [9].

The harmonic response analysis is a linear analysis, which is used to determine the steady state response of the structure under the load changing with the harmonic law of time. Its motion equation is expressed as follows:

\[-w^2[M] + iw[K](\{\varphi_i\} + i\{\phi_i\}) = (\{F_1\} + i\{F_2\})\]

The harmonic load of analysis is set as the load of shell box and lifting weight under the condition of sea condition and the harmonic response frequency is set at 0 - 50 Hz. From the modal analysis results of the lift, it can be seen that the vibration response of the side plate is large under the condition of motion. Therefore, it is necessary to focus on the analysis of the displacement response characteristics of this part. The x and z direction vibration displacement of the resonant lower side plate is shown in the following figure:

![Figure 4 X-direction vibration displacement of side plate.](image1)

![Figure 5 Z-direction vibration displacement of side plate.](image2)

From Figure. 4 and Figure 5, it can be seen that when the external excitation frequency is below 5Hz, the deformation of the side plate of the lift is very small. The x direction deformation is 0.42 mm and the maximum z direction deformation is 0.097mm. The deformation of both directions is in the elastic control zone. When external excitation is 20-30 Hz, the side plate is deformed step by step. The deformation is large, reaching 10 mm, and the chance of rise and fall is destroyed. When the excitation frequency increases above 30Hz, the deformation of the side plate decreases sharply, and the harmonic response law of the side plate of the lift is consistent with the modal frequency value. The ship operates at sea as low frequency vibration. For the fifth stage sea condition, the response frequency of the external dynamic load is 0.077~0.2, which is far from the frequency of 5Hz, which has a large deformation from the lift. The dynamic vibration resistance of the lift is good, which can meet the needs of the system.

7. **Static analysis under lifting maneuvering load**

During the movement of the lift, in addition to the load of itself and the load of the bullet box, there are centrifugal and axial forces derived from the rolling center of the car along the hull under the action of sea conditions. The force is the harmonic load. In the static analysis of the lift, the amplitude of the lift is selected to superimpose [10]. The equivalent stress and total deformation in the motion of the lift car are shown in Figures 6 and Figure 7.
As can be seen from Figure 6, the maximum equivalent stress of lift car is 104.5MPa under static load and external excitation load. The stress occurs at the oblique reinforcement of the box supported in the car, which is the stress concentration area. The frame material of the lift is Q345A, whose stress value does not reach the yield limit of the material itself. Safety factor is 3.3. However, to improve the safety factor of the elevator, it is necessary to uniformly thicken and round the supporting oblique ribs. The local large stress caused by stress concentration is eliminated. It can be seen from figure 7 that the maximum deformation of the side plate is 0.77mm during the movement of the lift. The deformation is the superposition of its longitudinal and transverse deformation. The longitudinal deformation is the longitudinal deformation, which is elastic deformation and has no effect on the movement of the lift and its docking parts within the control range of the docking of the components. Therefore, the static strength of the lift meets the requirements and there is no hidden danger of safety.

8. Conclusions
The results of modal analysis and harmonious response analysis of lift show that the minimum natural frequency of lift car is 25.6Hz. The maximum frequency of external excitation under the condition of Class 5 sea condition is 0.2Hz. Then the lift can work normally under Class 5 sea condition and there will be no resonance. The external excitation frequency of the lift should be avoided by 20 ~ 40 Hz.

The static strength analysis of the lift shows that the maximum deformation of the side plate of the lift is 0.7mm during the movement of the side plate of the lift. The maximum deformation is within the control range, so that it will not affect the docking of adjacent components.

When the lift moves under Class 5 sea conditions, the maximum stress appears on the inclined tendons supported by the cartridge box. At this time, the stress value is 104.5MPa and the safety factor is 3.3. The stress value is within the range of material strength and it will not cause damage to the structure. To increase the safety factor, the reinforcement plate needs to be thickened and fillet treated later. At the same time, the welding quality needs guaranteeing to ensure the overall strength.

References
[1] CHEN T.F. Research on development of naval gun[J]. Ship Science And Technology,2007,29(S1):3-10.
[2] H.A.J.H. Finite Element for Piezoelectric Vibration[J].International Journal Numerical Methods of Engineering,1970,2:151 ~ 157.
[3] WANG X.J. Design Principle of Naval Gun[M]. Wuhan: Naval Engineering University,2003.
[4] CHENG D.X. Mechanical design manual [M]. Beijing: Chemical Industry Press,2002.
[5] FU Y.H. Foundation of Finite Element Analysis[M]. Wuhan: Wuhan university press,2006.
[6] LIU Z.G, GONG S.G. Modal analysis of naval gun bracket based on ANSYS[J]. Ship Science And Technology, 2010,32(9):52-54.
[7] CAO S.Q. Modal analysis of vibration structure[M]. Tianjin: Tianjin University Press.
[8] GJB1060.1-1991, Requirements for Environmental Conditions of Ships---Mechanical environment[S]. Beijing: National Defense Science, Technology and Industry Commission, 1991.

[9] JIANG S.Y, ZHU S.L. Dynamic Characteristic Parameters of Linear Guideway[J]. Journal of Mechanical Engineering, 2010, 46(1): 92-99.

[10] MOU X.L, LI L.B. Structural Design and Finite Element Analysis of Long Span Box Girder[J]. Mechanical Research and Application, 2009, 22(2): 40-42.