Analysis the Strength and Vibration Characteristics of the Frame of the Anode Trailer by the Finite Element

Hong Yu JIA1, Ai Wei LI1, Ying WANG1, Hai Long WANG2

1 School of Civil Engineering, Inner Mongolia University of Science and Technology, Baotou, Inner Mongolia 014010, China
2 Chongqing Iron and Steel (Group) Mining Co. Ltd, Chongqing, 401325, China
Email:763966616@qq.com

Abstract. In order to locate the specific positions of frame destruction and find reasons of them, it is a feasible to analyze static strength, vibration mode and transient state by ANSYS solid modeling with the basis of design drawing for the single type of anode trailer. The analysis result shows the position of the maximum stress of the frame is at the front end of the lifting frame in the transient analysis, the tearing damage of the front end of the anode trailer frame is caused by the U-shaped frame with carbon block hit the lifting frame when the anode trailer is in reverse loading. In practice, we should avoid excessive impact force when reversing.

1. Introduction
Anode handling vehicle is a necessary assistant tools in the production of aluminum electrolyze. Anode handling vehicle is composed of tractor (main engine) and anode trailer. According to features of anode transportation during the production process in electrolytic aluminum enterprise, environmental circumstance of vehicle application and operational condition of vehicles, closed transportation can be thought as an effective way which can control the emission of toxic gas in the process of residual anode transportation and scattering of dust. Besides, environmental pollution can also be released. Take the front vehicle of the single type of anode transportation vehicle as power plant, mainly composed of front frame, cab, engine device, double stage air filter, hydraulic oil pump, hydraulic motor, gearbox, transmission shaft, driving axle, wheel and hinge device and so on. The hinge device is composed of a double hinged arm and a steering oil cylinder, besides, the front vehicle and the rear vehicle are connected into a whole vehicle by hinge device. Rear vehicle for carrying device, mainly composed of U-shaped frame type car frame, double four-bar fork lift lifting device, lifting hydraulic cylinder, limit structure, driven shaft and wheel and so on.

Frame is an important component of anode Trailer, it often deformed and damaged. In order to analyze ways of frame damaged location preliminarily, analysis of vibration characteristics will be present in this paper.

2. Frame Finite Element Model
The finite element model is established by taking the design drawings of a certain type anode trailer frame as an example. Main dimensions of frame: Length × width × height = 4270 mm × 3000 mm × 1770 mm. The car originally used Q235 steel for assembly welding, and now the use of Q345 steel welding and assembly, the results calculated in this paper will be analyzed and concluded with the two methods. The modeling process of anode trailer frame and lifting frame is modeled by 1:1, the specific
modeling process is as follows [1-3]:

2.1. Unit Selection
The simulated object of this paper is the anode trailer frame, in order to simulate the vibration characteristics of the frame, the model unit of Solid-20 node 186 will be adopted [4-6].

2.2. Material Parameters
Performance parameters of the material are shown in table 1

| Material Parameters     | Numerical Value |
|-------------------------|-----------------|
| Elastic Modulus $E$     | 206 GPa         |
| Poisson's Ratio         | 0.3             |
| Density                 | 7850 kg / m$^3$ |
| Gravity Acceleration    | 9.8 m / s$^2$   |

2.3. Establishment of Model
The reference of solid modeling is frame parameters, millimeter will be adopted as the unit of finite element model during the process of modeling. Model diagram is shown in Figure 1. During the modeling process, the hydraulic cylinder system of the trailer frame and the lifting frame is simplified to be connected with the lifting frame and the trailer frame by the solid block. The process of carbon handling on the trailer frame will not be considered. The trailer frame is in static state, and its net static load is 8 tons. In the calculation of the modal process the trailer frame being analyzed only in the case of its own gravity. When modeling the transient analysis, it is only analysis the different impact force exerted on the model.

2.4. Mesh Generation for Finite Element Model
Free partitioning is used in ANSYS, the partition length is 0.03 m. The model diagram after dividing the mesh is shown in Figure 1.

2.5. Add Constraint and Load
In reality anode trailer frame has the tire, and by the front drive the frame work. In order to simulate the work of anode trailer, a vertical upward constraint is applied to the contact upper surface of the vehicle frame and the tire shaft respectively, the contact point of the front end of the frame and the front of the car, to establish a face and impose full constraint. Surface force is applied to the lifting frame.
3. Finite Element Static Strength Analysis of Frame

Finite element model of frame is calculated by the basis of given parameters [7-9]. When solving the strength of the frame, lifting frame to put the weight of 8 tons anode carbon block. When the frame finite element model is applied load, the 8 tons anode carbon block is to be transformed into a surface force applied on the lifting frame. As shown in Figure 2. With impact of the 8 tons load, the degree of deformation of frame roof is the most obvious in the vertical view, the maximum displacement of tip of the rearmost is 1.154 mm.

4. Finite Element Modal Analysis of Frame

After the finite element model of the frame is imposed constraints, its solution mode, solving the vibration frequency of the frame which plus transverse reinforcement model and the model without transverse reinforcement [10-12]. By Comparison the modal solution between plus transverse bar at the front end of the vehicle frame and the model without transverse reinforcement. the first-ten-step frequencies are listed, as shown in table two. The two frequencies are very close, which shows that the modal analysis of the frame is not affected by the plus transverse bar or without transverse bar. The vibration mode of the frame can be seen from the results of the modal analysis of the anode Trailer, through the multi order modal shapes can be found where the vibration is frequent and the deformation is large place in rear part of frame and the rear upper part of the shed. In the process of the design improvement of the frame, we should more to change the strength and stiffness of these place, in order to avoid damage when the frame vibration.

| Frequency Type | Frequency Number |
|----------------|------------------|
| Frequency of Plus Transverse Reinforcement (Hz) | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
| 16.89 | 32.62 | 54.10 | 59.57 | 65.22 | 16.87 | 32.58 | 54.05 | 59.63 | 65.19 |
| Frequency of no Transverse Reinforcement (Hz) | 17.67 | 33.42 | 55.35 | 59.81 | 65.34 | 17.62 | 33.35 | 55.34 | 59.83 | 65.20 |
5. Finite Element Transient Analysis of Frame

When the model is in the transient calculation, using four-nodes to show the change of impact force with time, as shown in Figure 3: Impcat point.

\[ F(t) = \frac{m \cdot (V - V_t)}{t} \]

$m$ for 8 tons of carbon block, carbon block is stationary, the lift frame of the anode trailer hitting the carbon blocks when is driving in, the impact time is 0.1 seconds. Different speeds of the car hit the carbon block will produce different impact force, different transient analysis results can be obtained by loading different impact forces. Simulation of different speed were hit the rear side of the car frame respectively. Reverse speed in the 2.5 m/s, 3.75 m/s, 5 m/s respectively hit the car frame, the impact force generated respectively are 2000 N, 3000 N, 4000 N. The corresponding results can be obtained by the impact of a certain point on the rear of the trailer lifting frame.

In the selection of impact point, according to the actual situation, impact point should be in the frame and lifting frame by the back position, and the U-shaped frame and the lifting frame impact at a point, the transient analysis of the frame under the impact force is analyzed by the selection of node 2098. Under the single direction impact force of 2000 N, the displacement of the left and right direction of node 2098 reaches to 1.58 cm. The deformation of the front and rear direction of the frame is small, and the deformation is 0.648 cm. When the car frame under the single direction impact force of 2000 N, the maximum resultant displacement occurs at the tail of the frame, and the deformation is the left and right direction of the outward displacement, the size of 0.01098 m. Accordingly, tail outward deformation is caused by the tearing of the front end of the frame.

Under the action of the 3000 N impact force, the displacement of the left and right directions in the horizontal direction of the node 2098 is reached 1.64 cm. The deformation of front and rear direction of the frame is very small, and the deformation is 0.647 cm. When the car frame under the single direction impact force of 3000 N, the maximum resultant displacement occurs at the end of the shed frame, and the deformation is horizontal displacement, the size of 0.016304 m.

Compare the deformation of the frame under the action of 2000 N and 3000 N impact force, the deformation of the frame under the action of the 3000 N impact force is greatly increased, and the deformation of the rear end has reached 0.022 m. Due to the large deformation...
of the rear end of the vehicle frame, has caused damage to the front end of the frame, so this will be the main reason for the tearing damage of the front end of the frame.

Table.3 Combined Displacement and Equivalent Stress under Different Concussion Forces

| Impact force | The maximum resultant displacement value (m) | The maximum equivalent stress value (MPa) |
|--------------|--------------------------------------------|------------------------------------------|
| 200000N      | 0.010908                                   | 211.916                                  |
| 300000N      | 0.016304                                   | 322.981                                  |
| 400000N      | 0.022057                                   | 436.858                                  |

The transient stress analysis under different impact forces is carried out, and the dynamic stress of the frame under different impact forces is compared. Under different conditions, the maximum stress and maximum displacement values are shown in Table 3. The maximum equivalent stress value and the maximum resultant displacement value are increased with the increase of the impact force. The maximum equivalent stress has reached 211.916 MPa under the impact force of 20000 N, the maximum stress point is close to Q235 steel, While the 300000 N impact force when the maximum equivalent stress is close to the maximum stress point of Q345 steel, which will appear damage point on the car body lifting frame. Q345 steel frame has been damaged under the impact force of 400000 N, reversing the car impact frame lead to car body damage. where the maximum equivalent stress is at the front end of the lifting frame (shown in fig), the impact force caused by the collision between the frame and the U-shaped frame with anode carbon block is the main reason for the tearing damage of the front end of the frame.

6. Conclusions

In this paper, through the ANSYS entity modeling, first, the vibration mode of the frame is analyzed, looking for easily damaged parts of the frame, then the frame of transient analysis, simulate the frame in the actual work of different reverse speed, loading of anode carbon block produced different impact force conditions, to find out the place where impact damage is generated and the size of the equivalent stress, so as to analyze whether the frame is damaged, and then better to solve the problem of the destruction of anode frame. The vibration mode of the frame can be seen from the results of the modal analysis of the anode trailer, through the multi order modal shapes can be found where the vibration is frequent and the deformation is large place in rear part of frame and the rear upper part of the shed. In the process of the design improvement of the frame, we should more to change the strength and stiffness of these place, in order to avoid damage when the frame vibration. The finite element model of anode trailer frame is analyzed, and the position of the maximum stress of the frame is found in the front end of the lifting frame. The tearing damage of the front end of the anode trailer frame is caused by the U-shaped frame with carbon block hit the lifting frame when the anode trailer is in reverse loading. In practice, we should avoid excessive impact force when reversing.

References

[1] Zhu H P, Yu J, Zhang J B. A summary review and advances of vibration-based damage identification methods in structural health monitoring[J]. Engineering Mechanics, 2008, 2828(2): 1-17.
[2] Chen T, Han Z X, Lei J J, Static analysis and modal analysis of the frame of type 6470 SUV[J]. Science Technology ang Engineering, 2010, 10(2): 565-568.
[3] Huang C C, Wei Z L, Shen G L, Bin L, Optimization design and experimental verification of LX131 type frame[J]. Coal Mine Machinery, 2009, 30(12): 150-151.
[4] Guo Y L, Zhao Y P, Wang Y Z, Modal frequency response analysis of light vehicle powertrain[J]. Journal of Vibration and Shock, 1998, 17(4): 75-78.
[5] Kim, Vehicle Structural collapse analysis using a finite element limit method[J]. International Journal of Vehicle Design, 2000, 10:70-74.

[6] Miao Z H, Ding J P, Wang X G, Finite element modal analysis of the engine structure of motorcycle frame[J]. Computer Aided Engineering, 2006, 15 (s1): 382-384.

[7] Wang T, Li H L, Research on bus vibration test based on finite element analysis[J]. Bus & Coach Technology and Research, 2013, 5: 46-50.

[8] Hao W H, Xiao X B, Shen H M, Ye Y M, Ansys7.0 case analysis and Application[M]. Tsinghua University Press, Beijing, 2004.

[9] Ling G L, Li Z F, Ansys 14.0 from entry to master[M]. Tsinghua University Press, Beijing, 2013.

[10] Xue D, Zhao Y Y, Finite element static analysis of passenger car frame[J]. Journal of Harbin Institute of Technology, 2006, 38(7): 1075-1078.

[11] Wang Y, Ma H J, Modal test and finite element analysis of car sub frame[J]. China Science and Technology Information, 2007, 3: 87-89.

[12] Guo Y L, Zhao Y P, Wang Y Z, Chen Y, Modal frequency response analysis of light vehicle powertrain[J]. Journal of Vibration and Shock, vol. 17, 1998, 4:75-78.