Research on reliability of cabin structure of a launch vehicle under road impact

Teng Wang, Rui Han, Kexin Qi, Yuxin Zhang and Qi Gong
Shanghai Institute of Electrical and Mechanical Engineering, Shanghai, China

E-mail: tenge1314@126.com

Abstract: A certain type of launch vehicle cabin not only bears the weight of the launching device, but also withstands the transient impact transmitted by the chassis, which is caused by the rough road. As an intermediate link, its structure should have sufficient strength and reliability. The structural strength of this type of cabin frame is simulated and calculated by using the finite element analysis method. The impact conditions are identified to ensure that the simulation input shock conditions can cover the actual impact situation through driving experiment. The simulation results show that the structure is reliable and the structural performance meets the requirements.

1. Introduction
As a highly integrated self-wheeled combat unit, the launch vehicle is usually required to have strong field mobility, obstacle-crossing capabilities and to be able to withstand the impact of uneven roads [1]. The launch vehicle cabin is a large load-bearing structure, which is installed above the chassis frame, and bears the weight of the launch device. As an intermediate link, it not only bears the weight of the launch device, but also withstands the impact transmitted by the chassis. The design of its structure is directly related to the reliability and adaptability of the launch vehicle, so it should have sufficient strength and rigidity.

Zhang [2] et al. established a finite element model of the whole vehicle for a certain type of vehicle. Then they simulated and analyzed the displacement and acceleration of the whole vehicle and passengers in Z direction under the impact load, but they did not pay attention to the structural strength of the vehicle which bears not only the load, but also the impact. Zhang [3] et al. conducted a finite element simulation of impact on a combined airborne launcher, but did not verify the rationality of the input shock conditions for the simulation through experiments.
Aiming at the strength and stiffness requirements of the cabin structure in the development of the launch vehicle, firstly, this paper takes the cabin of a certain type of launch vehicle as the research object and analyzes the form of the impact load on the cabin. And then, this paper adopts the classical pulse shock waveform -- the final peak saw-tooth wave. Finally, the finite element analysis method was used to simulate the structure of the cabin frame, and the experiment was carried out to verify the rationality of the input shock conditions [4-6].

2. Frame structure of cabin
The frame structure of the cabin is shown in Figure 1, it is welded by 5A06 rectangular aluminum tube and aluminum plates. The launcher is installed on the upper surface of the race, and the lower surface of the race is welded to the cabin frame. This part is the main bearing structure of the cabin. The cabin is set on the two girders of the chassis through the six contact surfaces at the bottom of the side beam. The side beam is fixed on the girder of the chassis by bolts, and withstands the impact of the road transmitted by the chassis during driving.

![Figure 1. Frame structure of the cabin.](image)

3. Impact conditions
3.1. Analysis of impact conditions
The time-domain waveform characteristics of the excitation parameters of the impact of road are as follows [7]: The instantaneous value rises from the equilibrium position to the maximum value in a very short time, and then drops to the equilibrium position, which belongs to the type of saw-tooth waveform. And the excitation parameters can be ideally described by mathematical formulas. There are two ways to describe the shock wave. One is to describe the inherent characteristics of shock waveform itself, which can be described by shape, peak value and pulse width (i.e. duration) of the waveform in the time domain, and the main frequency component and frequency range of the shock can also be found by using Fourier spectrum in the frequency domain. The second is to describe the effect of shock on the system. Shock response spectrum can be used, which is a function between
the response peak value of the impact on the undamped or damped single-degree-of-freedom system base and the natural frequency of the system.

The general dynamic equation of impact load in finite element form [8] is as follows:

\[ M\dddot{u} + C\dot{u} + Ku = F \]  

Among them: \( M \) is the mass matrix; \( C \) is the damping matrix; \( K \) is the stiffness matrix; \( u \) is the displacement; \( \dot{u} \) is the velocity; \( \dddot{u} \) is the acceleration; \( F \) is the load. Because the process of impact dynamics is very short, it is necessary to use the minimum time step to calculate the integration.

In the design stage, there is no actual measurement data, the shock waveform of classic pulse is usually used to reproduce the shock waveform [9].

1) Pulse waveform: final peak saw-tooth wave;
2) Peak acceleration: +20g;
3) Pulse width: 11ms;
4) Direction of application: three mutually perpendicular axes.

![Figure 2. Final peak saw-tooth wave](image)

3.2. Verification of impact condition

Combined with the driving experiment on the D-class road [10], the impact test was carried out. The rationality of the input shock condition of the traditional simulation calculations—final peak saw-tooth wave (peak acceleration: +20g, pulse width: 11ms), was verified.

The direct impact load on the cabin was applied by the chassis, so the measuring points were selected on the chassis, and the three-axis sensor was arranged for the impact test. The test points of the chassis were selected as shown in Figure 3, and the specific positions of the three test points (number: 4, 5, 6) are shown in Table 1.
Figure 3. The test points of the chassis.

Table 1. The specific positions of the three test points.

| Number | The location of the measuring point                                      | Picture of test point       |
|--------|--------------------------------------------------------------------------|-----------------------------|
| 4      | The front connecting plate between the left side of the cabin and the    | ![Picture of test point 4]  |
|        | chassis frame                                                           |                             |
| 5      | Middle of left side of the chassis frame                                 | ![Picture of test point 5]  |
| 6      | The back connecting plate between the right side of the cabin and the    | ![Picture of test point 6]  |
|        | chassis frame                                                           |                             |

The impact response spectrum of the test points was analyzed and compared with the standard impact response spectrum of the final peak saw-tooth wave (peak acceleration: +20g, pulse width; 11ms), which is the input shock condition for the simulation, as shown in Figure 4.
Impact response spectrum of test point 4

Impact response spectrum of test point 5
Impact response spectrum of test point 6

**Figure 4.** Comparison curve of measured shock and simulated shock.

Among them, the blue curve is the standard impact response spectrum of final peak saw-tooth wave, the red curve is the various axial impact response spectrum of the test points of the cabin. It can be seen from the comparative curve that the input shock conditions of the simulation calculation cover the actual shock conditions of the experiment.

4. Simulation analysis

4.1. Constraints and loads

The launcher was mounted on the upper surface of the race, so a mass point was added to simulate the mass of the launcher and was coupled with the upper surface of the race, as shown in Figure 5. The cabin frame body was mounted on the chassis girder through six contact surfaces of its bottom to accept the impact transmitted by the chassis. Therefore, transient impact load was loaded on the six contact surfaces, as shown in Figure 6. The cabin frame body was connected to the chassis girder. The vibration caused by the impact is attenuated under the action of the suspension. Therefore, a spring restraint was applied to the fixed position of cabin and chassis, and the spring damping and the overall structural damping are set, as shown in Figure 7.
4.2. Calculation of single axial impact

4.2.1. Longitudinal impact
The impact waveform was input in accordance with (0,0), (11,20), and the unit step size was 0.5ms for a total of 100 steps. The waveform of the stress change over time is shown in Figure 8, which excludes the concentrated stress at the constraint positions, with a maximum stress of 135MPa.

4.2.2. Vertical impact
In addition to the direction, the loading conditions are the same as above. The waveform of the stress change over time is shown in Figure 9, which excludes the concentrated stress at the constraint positions, with a maximum stress of 70MPa.

4.2.3. Horizontal impact
In addition to the direction, the loading conditions are the same as above. The waveform of the stress change over time is shown in Figure 10, which excludes the concentrated stress at the constraint positions, with a maximum stress of 127MPa.
**Figure 8.** The stress-time curve of longitudinal impact.

**Figure 9.** The stress-time curve of vertical impact.

**Figure 10.** The stress-time curve of horizontal impact.
4.3. Calculation of three axial impacts under extreme conditions

All the above calculations are based on a single axial impact load. Considering the possibility of simultaneous occurrence of bumping (vertical), turning (horizontal), braking (longitudinal) and other driving conditions in the actual driving process, the extreme condition, that is the cabin is subjected to impact force in three axial directions at the same time, is analyzed. The results are shown in Figure 11, and the maximum stress is 154MPa.

Figure 11. Stress nephogram of three axial loads at the same time

In conclusion, the simulation results of single axial impact and three axial impact are summarized and compared with the material strength and weld strength, as shown in Table 2 (According to the standard in mechanical design manual, the strength of fillet weld of 5A06 aluminum is not less than 0.65 times of the base metal).

4.4. Simulation results

Table 2. Comparison of simulation and design under various impact conditions.

| Conditions               | Maximum stress (MPa) | Yield strength of 5A06 aluminum (MPa) | Yield strength of weld (0.65 times of the base metal) (MPa) |
|--------------------------|----------------------|--------------------------------------|----------------------------------------------------------|
| Single axial impact       |                      |                                      |                                                          |
| longitudinal             | 135                  |                                      |                                                          |
| vertical                 | 70                   |                                      |                                                          |
| horizontal               | 127                  |                                      |                                                          |
| Simultaneous impact of three axial directions | 154                  |                                      |                                                          |
It can be seen from the above comparison that the maximum stress of the cabin under extreme condition is 154MPa. The cabin frame is made of 5A06, its yield strength is 315MPa and allowable stress is 210MPa. According to the standard in mechanical design manual, the yield strength of fillet weld of 5A06 aluminum is 205MPa, its allowable stress is 180MPa. They are all greater than the maximum stress of 154MPa. Therefore, the design strength of the cabin frame and the weld joint meet the requirements.

5. Conclusion
In this paper, the cabin of a certain launch vehicle was analyzed, which bears not only the weight of the launcher, but also the transient impact caused by the uneven road transmitted by the chassis. The structural strength of this cabin was simulated and calculated by using the finite element analysis method to verify the reliability of the cabin frame structure. Combined with the driving experiment, the impact conditions are studied and it is concluded that:

1) Through the experiment, the correctness of the input shock conditions of the simulation is verified. That is the final peak saw-tooth wave can cover the actual impact situation of the driving experiment;

2) Through the finite element simulation, it is verified that the strength and reliability of the cabin structure of a certain launch vehicle meets the requirements.

References
[1] Miao D H, Ma Z J and Li J S 2018 Simulation research on the stability of the hoisting and loading launch vehicle J. Air Defense.1(4) 58-64
[2] Zhang L L and Lv C 2016 Transient dynamic analysis of vehicle under the impact of road impact load J. Equipment Manufacturing Technology. 1132-33+47
[3] Zhang D Z, Xue H R, Qin G and Yu J X 2009 Structural design of combined helicopter launcher frame J. Journal of Missiles, rockets and guidance.1-4
[4] Zhi P P,Li Y H,Chen B Z and Shi S S 2020 Bounds-based structure reliability analysis of bogie frame under variable load cases J. Engineering Failure Analysis.114
[5] Jensen H A, Mayorga F and Valdebenito M A 2020 On the reliability of structures equipped with a class of friction-based devices under stochastic excitation J. Computer Methods in Applied Mechanics and Engineering.364
[6] Hu H Y, Huang Z L and Jiang C 2017 Research on optimal design of reliability of vehicle structure strength J.Computer Simulation.34(03) 118-122
[7] Dong L X, Zhou X F, Liu L L and Dai G Z 2006 Impact simulation of construction vehicle falling object protection structureJ. Modern machinery. 0453-54+71
[8] Liu G, Chi H W, Wang Y P, Guo S and Yuan H Y 2014 Finite element simulation of impact response of flight simulator platform structure J. System simulation technology.0216-20
[9] GJB 150.18A-2009 2009 Environmental test method for military equipment laboratory section 18: Impact tests
[10] GB/T 7031-2005 2005 Mechanical vibration road pavement spectrum measurement data
reports