

Analysis and Optimization of Impact Condition of Power Conditioning Inverter Based on ANSYS

Guilin Chen¹, Yong Wu¹, Qijun Pan², Fuli Ning³ and Peng Jiang¹

¹School Of Automation, Wuhan University of Technology, 430070 Wuhan, China

²Daqo Group Co. Ltd., Yang Zhong, 212211 Jiangsu, China

³Wuhan Aonesoft Technology Co.Ltd., 430062 Wuhan, China

Abstract. This paper uses the professional finite element analysis software ANSYS to process a certain type of power regulating inverter. The overall finite element model of the inverter is established then the three-way impact condition is analysed according to the requirements of GJB 1060.1-91. Through the modal analysis in ANSYS software, the natural frequency and natural vibration mode of the inverter are calculated. Based on modal analysis of the inverter, the DDAM shock spectrum analysis function is used to analyse the impact condition of the inverter. Through the calculation of the impact condition, the deformation and stress state of the inverter structure under various working conditions are obtained. By analysing the calculation results, the weak position of the inverter structure is found and the corresponding optimization scheme is proposed to improve the dynamic performance and improve the electron. The overall dynamic performance of the inverter cabinet also provides an important reference for the design of subsequent electronic cabinets.

1 Introduction

On electrical equipment for ships, anti-vibration and anti-shock capability is one of the most important standards for electronic equipment. However, because experimental conditions are limited, it is very difficult to use the product for experiments. Therefore, it is very suitable to use computer simulation software to simulate the impact resistance[1]. With the development of computer technology, the structure simulation software has become more and more graphical from the beginning, and the functions are more and more integrated. Up to now, it has been able to model the product directly in software and can calculate the simulation of structure, fluid, electromagnetic field, sound field, etc. And in simulation, it can quickly adjust the model and parameters to achieve the purpose of design and experiment. Based on finite element principle, this paper simulates the structure of inverter using ANSYS software and analyse the dynamic performance. The results can provide beneficial design proposal for electrical equipment.

2 Inverter model establishment and modal calculation

2.1. Model establishment

Inverter is an electronic device that converts DC voltage into AC voltage and can adjust output voltage. The basic structure size of the analysed inverter is

3000mm×1200mm×2000mm (W×D×H) and its weight is nearly four tons.

Finite element analysis method requires model division grid. The original model needs to be simplified before mesh grid partition and the calculation[2]. The main idea of model simplification is to retain the main frame structure, large components and other load-bearing structures. As for small components, decorative parts or external devices that do not support the structure, they can be removed. The aim of the model simplification and geometrically process are that the model is easy to divide the mesh grid.

The inverter model established in ANSYS software is simplified by SCDM as shown in Figure 1.

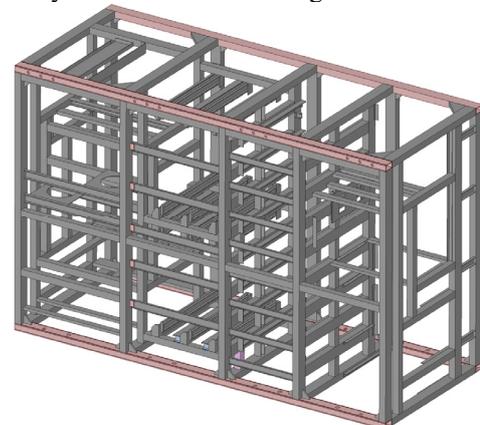


Figure 1. Simplified inverter model.

2.2. Meshing and spring settings

* Corresponding author: ^a 1559972750@qq.com

In ANSYS software, modal analysis is calculated based on meshing and setting boundary conditions[3]. The requirements for meshing include the frame with load bearing and the device needs a relatively small mesh which is about 15mm. But for the large devices, because there is no significant impact on the carrying capacity of the structure. So there is no need to refine the mesh and the size is set about 50mm. Most of the structures in this paper use multi-domain hexahedral meshing. Compared with tetrahedral meshing, it has better calculation accuracy. The same size uses less mesh, and the calculation scale is relatively small, but the geometric adaptability is poor. It takes more time to simplify and cut the geometry.

Due to the large number of components in the inverter model, a connection relationship needs to be established among the parts. With the ANSYS WORKBENCH contacts automatic detection function, while importing the geometric model, the program will automatically establish the contact pair connection among the parts. The inverter model establishes 1335 binding contact pairs to ensure the force transmission among the various components.

Eight shock absorbers are used on the bottom of the inverter vibration isolation system, and four shock absorbers are installed on the back. The damper is simulated using the spring damping unit "Spring" in the ANSYS software. In the numerical simulation, each damper is set with 3 Spring units. The impact stiffness setting is used for impact analysis, and the impact stiffness parameters are shown in Table 1.

Table 1. Impact stiffness parameters.

Impact stiffness N/mm			
parameters	Z	X	Y
Bottom shock absorber	317	136	136
Back shock absorber	570	245	245

After the meshing and boundary conditions are set, the model can start the modal calculation. The divided main frame mesh and the set spring are shown in Figure 2.

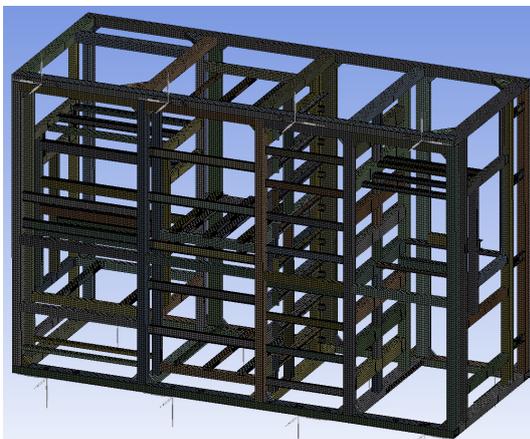


Figure 2. Meshing and spring settings.

2.3. Modal calculation

The modal calculation needs to be performed in imperial units before the DDAM analysis[4]. The first 12 modal of the inverter are calculated. The numerical results of the 12 modal are calculated as shown in Table 2.

Table 2. Modal order and corresponding frequency value.

Mode	Frequency[Hz]	Mode	Frequency[Hz]
1	3.80	7	36.09
2	4.27	8	36.90
3	5.47	9	45.26
4	7.87	10	47.71
5	13.51	11	49.59
6	14.15	12	51.27

From the modal analysis results, it can be seen that the frequency increases sharply between the 6th order and 7th order from 14.15Hz to 36.09Hz, because of its higher order, its vibration form will be more complicated. However, the order that needs to be paid more attention is generally the previous modal order because the lower frequencies are more easily to produce vibration. From table 2 above, the maximum frequency of the first 6 modal does not exceed 15 Hz.

3 Impact condition setting

This paper uses Dynamic Design Analysis Method (DDAM)[5]to analyse impact response of the inverter. It needs APDL command in ANSYS software to realise.

According to the analysis requirements of DDAM method, the impact design analysis conditions are surface ship equipment, equipment impact grade (Class A), equipment installation position (hull part installation), and plastic deformation is not allowed for elastic analysis. According to the obtained spectral value calculation constant expression and the direction coefficient table of various installation positions of the surface ship[6], the formulas of the impact acceleration spectrum value and the velocity spectrum value under various modes are obtained as follows:

$$A = AF \times 20 \times \frac{(37.5 + \omega_i)(12 + \omega_i)}{(6 + \omega_i)^2} \quad (1)$$

$$V = VF \times 60 \times \frac{(12 + \omega_i)}{(6 + \omega_i)} \quad (2)$$

Where VF and AF are the coefficient, ω_i is the modal quality, A is the spectral acceleration and its unit is g ($g=386\text{in/s}^2$), V is the spectral velocity and its unit is in/s[7].

According to the obtained impact spectrum, the impact design acceleration of the frame system in three directions is obtained listed in Table 3. The impact calculation can be performed on the three directions of the frame respectively.

Table 3. Design impact spectrum values.

Impact direction	Aa	Va	Remarks
Vertical	1.0A ₀	1.0V ₀	Z direction
Landscape	0.4A ₀	0.4V ₀	Y direction
Portrait	0.2A ₀	0.2V ₀	X direction

4 Analysis and improvement of impact simulation calculation results

The calculation results of the impact condition are listed in Table 4.

Table 4. Impact analysis calculation results.

direction	Displacement result			Equivalent stress maximum /MPa
	Maximum /mm	Minimum value /mm	Relative change value /mm	
X	109.4	96.9	12.5	616
Y	103.6	66.4	37.2	560
Z	67.9	58.3	9.6	694

It can be seen from the values in the table 4 that both the displacement result and the equivalent stress result exceed the yield strength of the main frame material.

The three-dimensional stress cloud diagram are shown in Figure 3, Figure 4 and Figure 5.

Although the stress value is relatively large, it can be seen from the stress cloud diagrams in three directions that the actual excess of the yield stress is relatively small, and there is a small range of stress concentration. It can be seen from the three-direction stress cloud diagram that the stress exceeding the yielding position is mainly concentrated on the vertical beam of the power unit frame, the part of the mounting beam, the connection of the vibration isolator and the connection hole of the vibration isolator. The position that occurs in all three directions is on the vertical beam of the frame portion where the power unit is located.

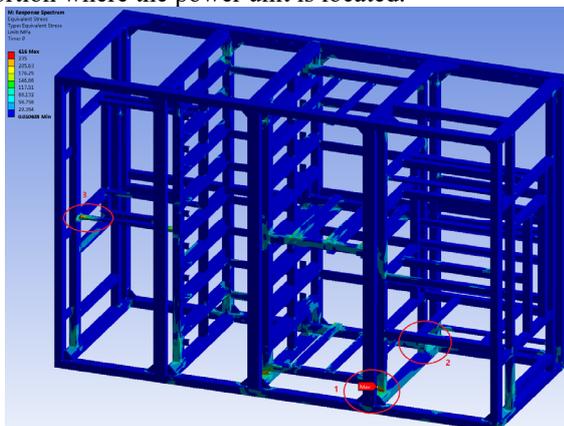


Figure 3. X-direction stress cloud.

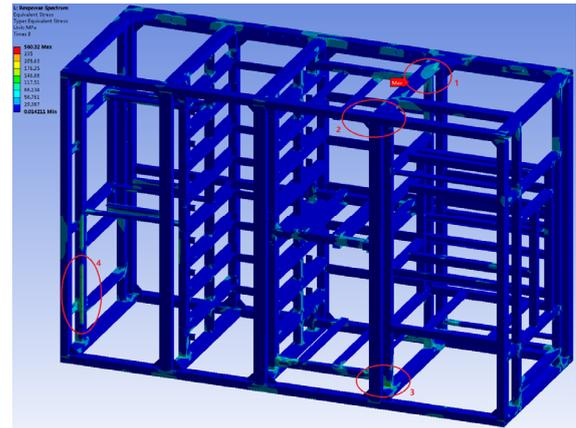


Figure 4. Y-direction stress cloud.

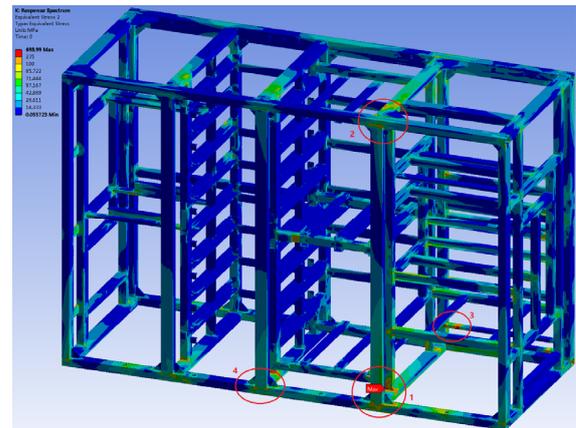


Figure 5. Z-direction stress cloud.

Through finite element analysis in ANSYS software, under the impact condition of the main frame of the inverter, the overall stress level is mostly within the allowable range of material strength and the overall structural design of the frame is basically reasonable. But the local position stress value exceeds the yield strength. This local location needs to be appropriately enhanced and optimized. The optimization program mainly has the following contents:

(1) From the analysis results, the vertical beam and the lower beam where the power unit is installed are the parts with large stress values under the impact condition. It is recommended that the position should be properly strengthened. The triangular ribs can be added at the upper and lower root positions. Strengthen the rigidity of the root position to make the stress transition more smooth;

(2) It is recommended to strengthen the beam where the damper is connected. Although the stress level is not the highest, it has more parts close to the yield stress, and the beam is directly applied to the impact. Therefore, it is recommended to use higher yield strength steel to improve its impact resistance. The framework structure enhancement proposal is shown in Figure 6.

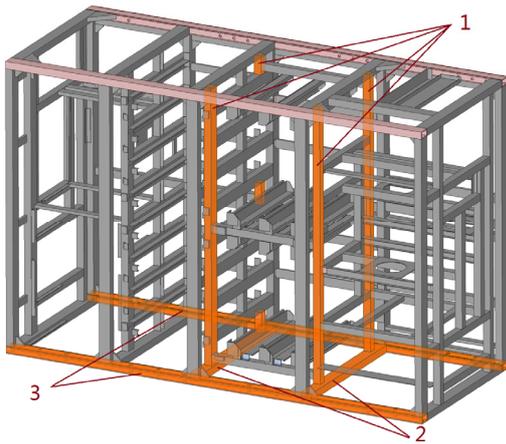


Figure 6. Framework structure enhancement proposal.

7. W. Liu, N. Ramp, Computation of shock resistance for rocket launcher using ddam. *Mine Warfare & Ship Self-Defence*. (2017)

5 Conclusion

According to the calculation results and the optimization scheme mentioned above, the framework model for calculation is changed and the calculation of the optimized model is performed. After changing the model, the impact of the DDAM analysis in three directions is simulated and the calculation results are shown in Table 5.

Table 5. Impact analysis calculation results.

Direction	Pre-optimal equivalent stress maximum /MPa	Optimized equivalent stress maximum /MPa
X direction	616	446
Y direction	560	450
Z direction	694	412

From the results in table 5 above, the maximum stress reduction after optimization is obvious.

With the development of computer technology, it is possible to simulate the impact through computer software calculation, and play a guiding role in the design of electronic equipment structure.

References

1. GX. Zhong, JY. Wu, JY. Luo, Dynamical simulation research of valve impact for drilling pump based on ansys/lr-dyna, *J. Machine Design*(2017)
2. W. Wang, YL. Cheng, B. Wang, Impact Simulation Analysis of Solid Wood Chair Based on ANSYS-DYNA, *J. Northwest Forestry University*(2018)
3. X. Jia, M. Zhu, P. Zhang, X. Zhang, Finite element analysis of impact on elevator landing door based on ansys. *Machine Design & Research*(2016)
4. Y. Wang, HX. Hua, *Ship modern impact theory and application*(Science Press, Beijing,2005)
5. X. Wang, Ansys-ddam study for the shock response of shipboard equipments. *Traffic Engineering & Technology for National Defence*. (2013)
6. PU. Jun, BK. Shi, Shock resistance analysis on certain ship lift based on ddam. *Ship Science & Technology*. (2017)