Structural Optimization of Explosion-proof Electrical Module

Jian Rui[1, a], Qing Zhang[2, b]

[1] School of Mechanical Engineering, Tianjin University, Tianjin 300072, China;

[2] School of Mechanical Engineering, Tianjin University, Tianjin 300072, China.

[a]tjuruijian@126.com, [b]zqtju@263.net

Keywords: Electrical module, explosion load, dynamic response, ANSYS optimization.

Abstract. Electrical module is the control unit of oil and gas processing, so it must have the ability to withstand the impact of the explosion. By transient dynamic module of ANSYS, dynamic response of electrical module under explosion load is analyzed. The results show that the maximum stress and displacement of electrical module are larger. Through ANSYS structural optimization, the maximum stress of electrical module decreases from 336 MPa to 269 MPa, the maximum displacement reduces to 16.9 mm from 22.7 mm.

Introduction

It is important for industrial buildings with explosion danger to have an explosion-proof design, which ensures the safety of buildings and people[1]. Related research has been carried out in the 1970s abroad. Such studies started relatively late in China. But with the development of the marine oil and gas, the safety of oil and gas production has more and more gained the attention. Currently, explosion-proof design of electrical module is based on engineering experience at home and abroad[2, 3]. In this paper, based on software ANSYS, electrical module is analyzed under explosion load. According to the analysis results, the structure of electrical module is optimized. As a result, it provides a powerful theory support for the explosion-proof design of electrical module. As the petrochemical industry pays more and more attention to how to safeguard the safe operation of the equipment in the process of oil and gas production and people's life, it has a good market prospect[4].

Finite Element Model

Electrical module is divided into three parts: the ground structure, the middle structure and the roof structure. Square-steels are connected vertically between the ground structure and middle structure, the middle structure and roof structure. The length of the whole electrical module is 9600, the width is 9600 mm, and the height is 9700 mm. Figure 1 is finite element model of electrical module.

The elastic modulus of steel is 205000 N/mm2, the Poisson's ratio is 0.3, and the density is 850 kg/m^3. The yield strength of steel is 345 MPa.

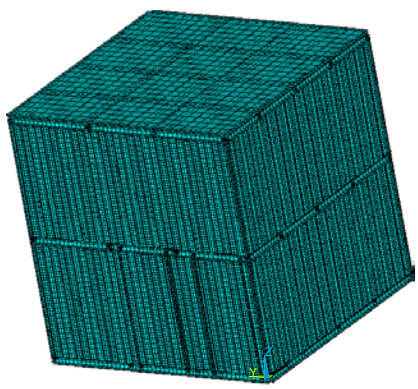


Fig. 1 Finite element model of electrical module

Analysis Results

In this paper, wave incident peak overpressure P_{so} of shock wave shock is determined as 10 kPa, positive pressure duration time t_d is 200 ms. The pressure-time curve is expressed in an ideal linear booster and attenuation stage. Figure 2 and 3 respectively are explosion load of the front wall and explosion load of the side and roof wall[5, 6].

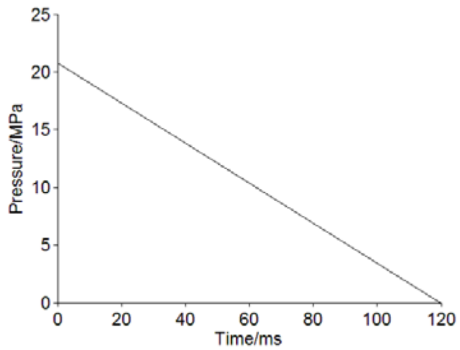


Fig. 2 Explosion load of the front wall

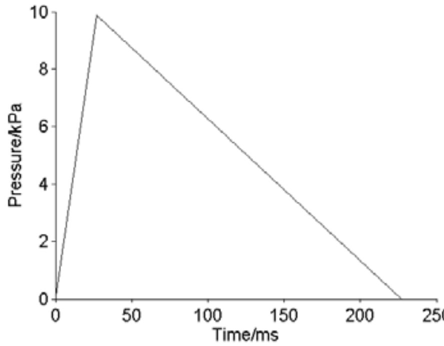


Fig. 3 Explosion load of the side and roof wall

Analysis results show that the maximum displacement of electrical module under explosion load is 22.7mm and the maximum stress is 336MPa, as shown in figure 4 and 5. The stress and displacement of electrical module are larger,so the structure optimization of electrical module is needed.

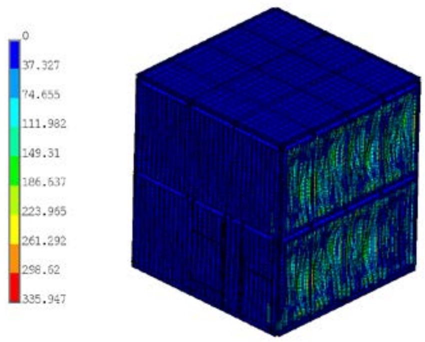


Fig. 4 Maximum von mises stress contour

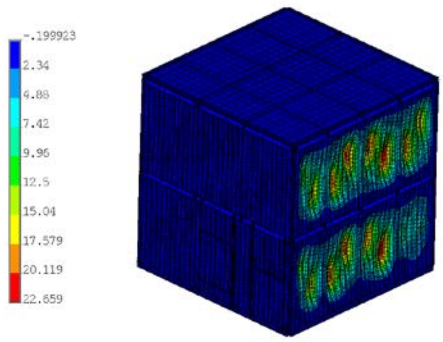


Fig. 5 Maximum displacement contour

ANSYS Optimization

ANSYS optimization steps include

(1) Optimization model is set up;

(2) The cycle analysis file is generated, it must include the entire analysis process, and meet the following conditions: parametric model, solving, extracting, specifying the state variables and objective functions;

(3) Parameters in correspond with analysis file variables in ANSYS database is established;

(4) Enter OPT, and specify the analysis file;

(5) Statement optimization variables;

(6) Select the optimization tool or optimization method;

(7) Specify the optimization loop control mode[7, 8].

Because the framework of electrical module is composed of I-beams, by optimizing the distance between the I-beams, it can improve the overall explosion-proof performance of electrical module. Due to each layer's I-beam layout of the electrical module is same, number of design variables reduces. A total of seven design variables are determined, as shown in Figure 6.

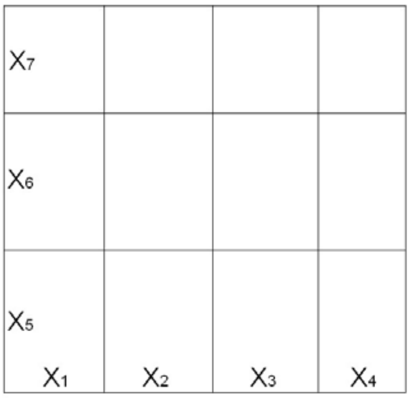


Fig. 6 Design variables

In the process of ANSYS optimization, a goal function only can be set. Maximum displacement on the module is selected as the objective function.

According to certain performance of the structure or design requirement, the performance constraint is established. Because the material is used in the Q345, mathematical expression of the state variable constraint function is shown blew

$$g_1(X) = \sigma - 345 \le 0 \tag{1}$$

Boundary constraints are used to limit a range of design variables and the relative relationship between them, usually expressed as the following form

$$X_{i\min} \le X_i \le X_{i\max} \quad i = 1,2,\dots,N \tag{2}$$

Considering the actual situation, boundary constraint functions of design variables are defined as

$$g_2(X) = -X_1 \le 0 \tag{3}$$

$$g_3(X) = X_1 - 2300 \le 0 \tag{4}$$

$$g_4(X) = -X_2 \le 0 \tag{5}$$

$$g_5(X) = X_2 - 2500 \le 0 \tag{6}$$

$$g_6(X) = -X_3 \le 0 \tag{7}$$

$$g_7(X) = X_3 - 2500 \le 0 \tag{8}$$

$$g_8(X) = X_1 + X_2 + X_3 + X_4 = 9300 \tag{9}$$

$$g_9(X) = -X_5 \le 0 \tag{10}$$

$$g_{10}(X) = X_5 - 3300 \le 0 \tag{11}$$

$$g_{11}(X) = -X_6 \le 0 \tag{12}$$

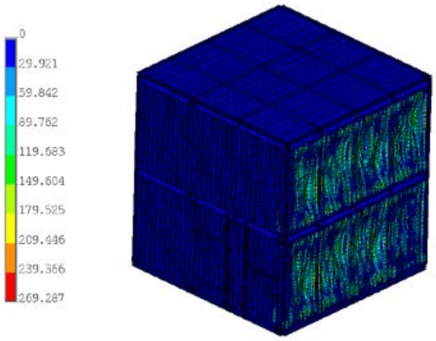
$$g_{12}(X) = X6 - 3000 \le 0 \tag{13}$$

$$g_{13}(X) = X_5 + X_6 + X_7 = 9300 \tag{14}$$

Through the ANSYS optimization design, the maximum displacement of electrical module under explosion load decreases from 22.7 mm to 16.9 mm, the maximum stress is 269.3 MPa ,as shown in Figure 7 and 8. The initial and optimal value of design variables are shown as Table 1.

Table 1 Optimization results of structural parameters

Variables	X_1	X_2	X_3	X_4	X_5	X_6	X_7
Initial Value(mm)	2000	2200	2400	2700	3200	2700	3400
Optimal Value(mm)	2110	2370	2300	2520	3050	2880	3370



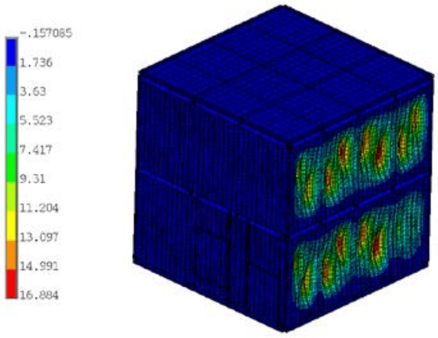


Fig. 7 Maximum von mises stress contour Fig. 8 Maximum displacement contour

Conclusions

In this paper, dynamic response of electrical module under explosion load is analyzed. The displacement and stress of electrical module is larger. Based on finite element optimization theory, Optimization mathematical model of electrical module is built. Through the ANSYS optimization, the maximum displacement of electrical module is 16.9 mm, the stress is 269.3MPa.

Acknowledgements

The authors wish to acknowledge the Key Laboratory of Mechanism Theory, Equipment Design of Ministry of Education, Tianjin hydraulic cylinder Engineering Center and the financial support by the Innovation and development of regional marine economy demonstration projects including Deep-sea oil and gas fields to build large structures critical equipment industry(No. CXSF2014-01), offshore oil and gas construction of large modules and complete sets of non-skid equipment industry engineering (No. CXSF2014-13).

References

[1] C Z Huang, J P Tan, Research on finite element mechanics analysis methods about large steel structure equipment, J. Machinery Design and Manufacture.6 69-71.(2005)

[2] J X Xu, J C Zhao, Dynamic response analysis of offshore platform under explosion, J. Journal of Disaster Prevention and Mitigation Engineering. 33 (2) 389-393. (2013)

[3] Y Y Wang, N He, Simulation test of rescue chamber under the explosive load based on the AUTODYN and LS-DYNA model, J. Journal of Safety and Environment. 14 (2) 9-12. (2014,)

[4] H. Du, R. Q. Du, Numerical simulation of the distribution of explosion pressure on buildings, J. Shanxi Architecture. 37 (11) 32-33. (2011)

[5] Z. X. Li, Z. X. Liu, Dynamic response analysis of steel structure under explosion load, J. Building Structure. 36 (6) 98-101. (2006)

[6] Y. J. Song, J. G. Li, J. H. Zheng, Calculational analysis of full-containment LNG storage tank under external explosion load, J. Petroleum Engineer Construction. 39(4) 7-10. (2013)

[7] L W Zhang, W Wang, Finite element optimize technology based on parameterize modding, J. Hydraulic Design. 2 18-20. (2008)

[8] S Y Cheng, Y Lu and X J Zhao, Application of the optimum technology in multi-stor light-weight steel framework by ANSYS software, J. Optimization of Capital Construction. 27(1) 93-99. (2006)