Journal of Physics: Conference Series

PAPER • **OPEN ACCESS**

Optimization design and finite element analysis of welding robot base based on ANSYS Workbench

To cite this article: Xiao-jie Li and Jian Liang 2022 *J. Phys.: Conf. Ser.* **2383** 012073

Optimization design and finite element analysis of welding robot base based on ANSYS Workbench

Xiao-jie Li[1a*], Jian Liang[1b*]

[1]School of Intelligent Manufacturing and Electrical Engineering, Guangzhou Institute of Science and Technology, Guangzhou, 510540, China

[a*]44451823@qq.com, [b*]67489622@qq.com

Abstract—A welding robot is a standard industrial robot, and its base is the essential component of the robot, which needs to ensure its safety, stability, and reliability. Using Solid Works software, taking a robot base in an automobile workshop as an example, a three-dimensional model of the robot base structure was established. The model was then directly imported into ANSYS Workbench without error. The finite element analysis method was used to carry out static analysis, modal analysis, and harmonious response analysis of the base model. According to the finite element calculation, the equivalent stress and deformation distribution cloud diagram of the robot base structure model is obtained. The structure size is optimized, and the original model is compared with the improved and optimized design scheme for finite element analysis. The solution's size is reduced, but the dynamic and static performance is also more advantageous, which achieves the expected structural optimization goal. The finite element simulation experiment verified that the welding robot base could be optimized in size to make its strength and working space better meet the design requirements. The simulation experiment analysis results are accurate and the design scheme is feasible.

1. Introduction

The automobile welding production line is an integral part of the automatic production of new model research and development. With the improvement of the automobile market and the technical requirements of automobile production, the application of industrial robots in the automobile body welding production line is becoming more and more extensive, effectively satisfying the mass production of the automobile industry. Therefore, how to increase the number of robots in the welding workshop has become one of the complex problems in measuring the automation rate of the welding line. In the early stage of the automobile welding production line design [1], it is necessary to make a reasonable layout of the robots in the welding workshop within a specific production planning and layout and design an automated automobile production line that meets the production capacity [2].

As the carrier of the robot, the robot base is the main load-bearing component. The general diameter of the robot base in the welding workshop is 1,650mm. The size of the base is too large to limit the placement space of the equipment in the process layout. Therefore, on the premise of ensuring the mechanical performance of the base, rationally improving the structure and size of the robot base can effectively improve the layout of the welding workshop, increase the number of robots, and improve the automation rate of the welding production line. This paper takes the base structure of a welding robot in a welding workshop as the research object. It uses the finite element method to analyze and optimize its structure size, which provides a sufficient basis for the rational design of the structure size of the welding robot base.

2. Static analysis of the original base structure of the welding robot

2.1. Original base structure and geometric modeling

The original base structure of a welding robot is cylindrical. The 3D modeling software Solidworks is used to establish the geometric model of the robot base. The geometric model was imported into the finite element analysis software Ansys Workbench, as shown in Figure 1.

2.2. Material parameters

The material of the base structure is Q235. Its elastic modulus is E=2.1×106Pa. The poisson's ratio is μ=0.3. The density is ρ=7, 900kg/m3.

2.3. Mesh the model

As shown in Figure 2, the base model has meshed with 15 mm tetrahedral elements. The total number of elements is 69472, the total number of nodes is 120816, the mesh skewness is 0.4, and the mesh quality is good.

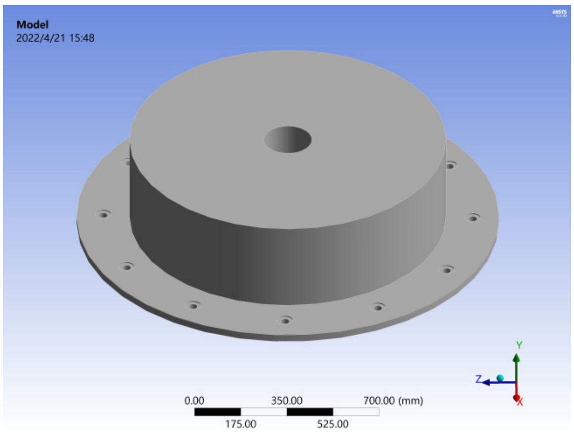


Fig.1 The finite element model of the original base structure of the welding robot

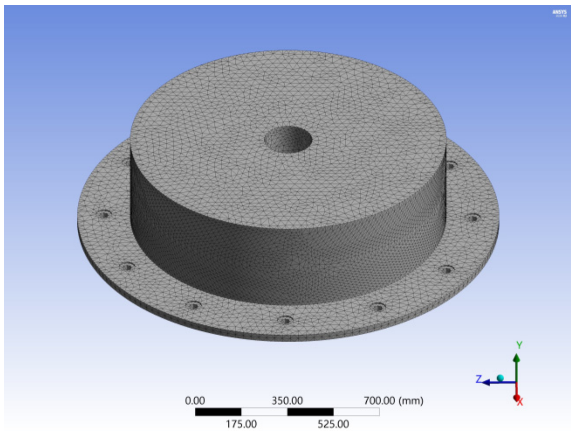
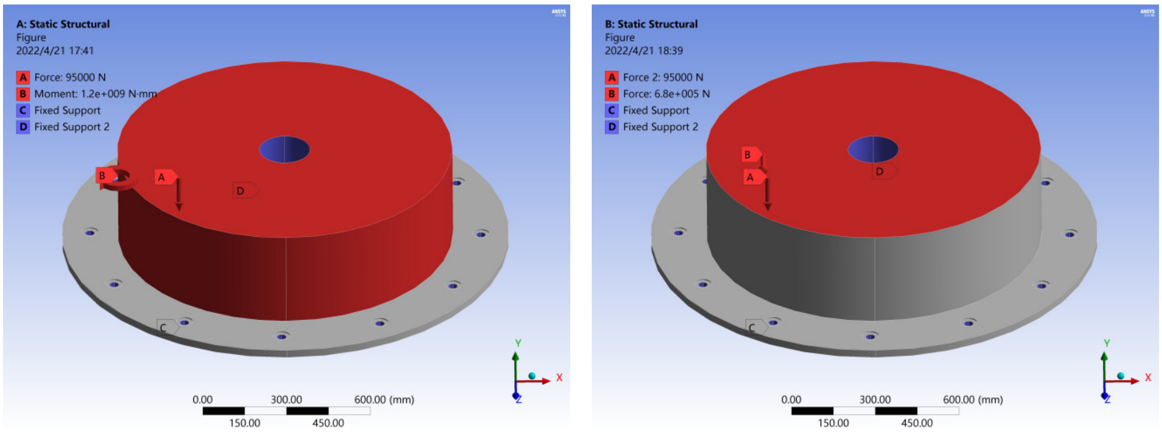


Fig.2 Schematic diagram of the meshing of the finite element model of the base

2.4. Boundary conditions and load application

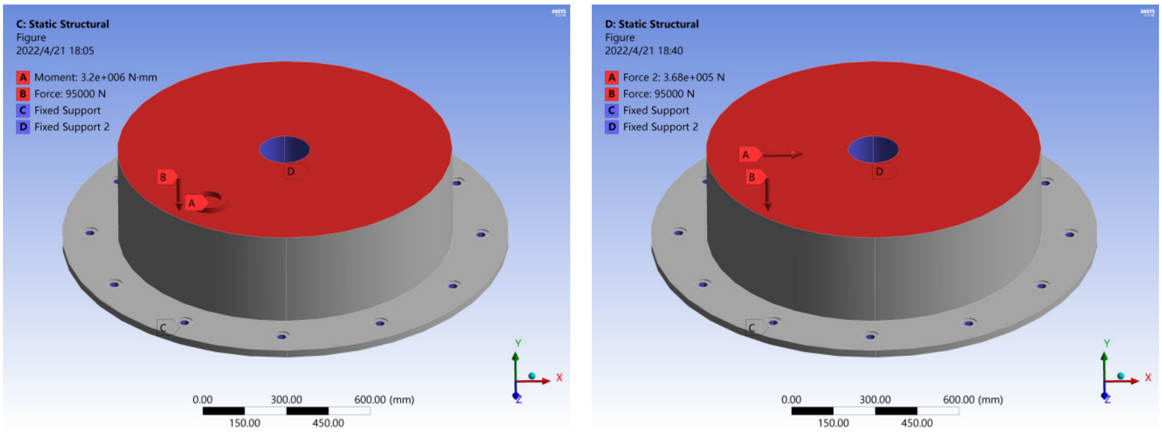
There are 12 bolt holes on the base plate of the base model, and the base is fixed on the ground by bolt connection, so the inner cylindrical surface of the bolt holes of the lower base plate adopts fixed constraints to limit all the freedom of translation and rotation.

The robot body is installed on the base, and the base is not only under the action of gravity of the body but also under the action of working load and inertial force [3]. The vertical inertia moment and inertia forces are generated when the robot body is tilted forward and the emergency stop. The horizontal inertia moment and inertia force generated when a rotating emergency stop is selected as the main working conditions acting on the base. The specific load distribution is shown in Figure 3 and Figure 4, respectively.



(a) Under the action of the vertical moment of inertia (b) Under the action of vertical force

Fig.3 Boundary constraints and load distribution during forward leaning and emergency stop



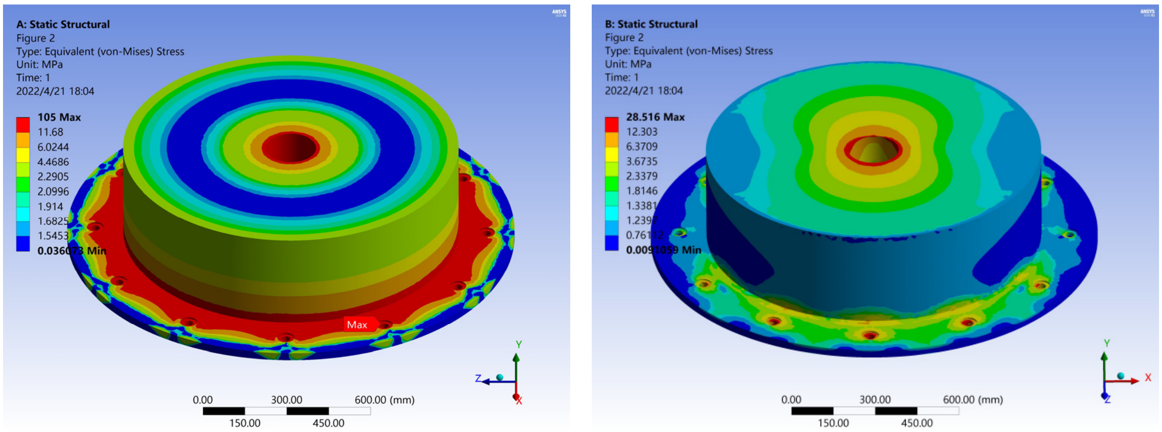
(a) Under the action of the horizontal moment of inertia (b) Under the action of horizontal inertial force

Fig.4 Boundary constraint and load distribution during rotation emergency stop

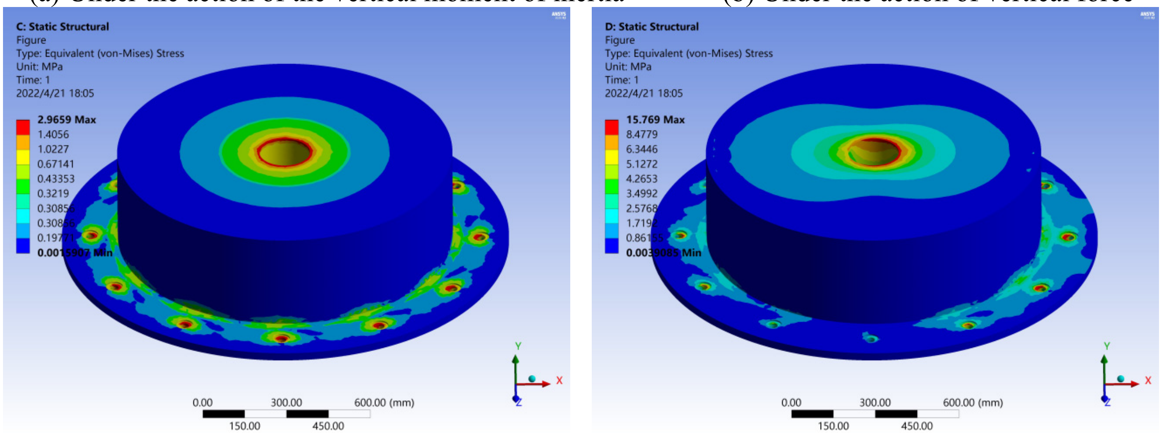
2.5. Results of finite element analysis

As the carrier of the robot body, the base is required to withstand various forms of loads and have stable performance. Therefore, it must first meet the strength requirements, that is, the maximum stress of the base when it is stressed is lower than the allowable stress of the material; secondly, it should meet the static stiffness requirements, that is, the overall The deformation should be as small as possible; in addition to a specific static performance, the base structure should also have good dynamic performance, which is generally the low-order modal characteristics of the structure.

The von-Mises stress, total deformation and first-order mode shape results of the original base structure under different loading conditions are shown in Fig. 5, Fig. 6 and Fig. 7. It can be seen from the figure that the overall equivalent stress and total deformations are the largest under the condition of sheer inertia moment when the robot body is tilted forward and emergency stop. The maximum von-Mises stress is 105MPa, which occurs at a bolt hole in the lower bottom plate, and the maximum total deformation is 0.0368mm, which occurs at a specific load-bearing place on the upper roof. The first-order modal frequency of the base is 703.95Hz. The yield strength of Q235 is 235MPa, and the safety factor is 1.2. According to the finite element analysis results, the maximum stress of the base is less than the allowable stress of 195MPa, which meets the strength requirements.

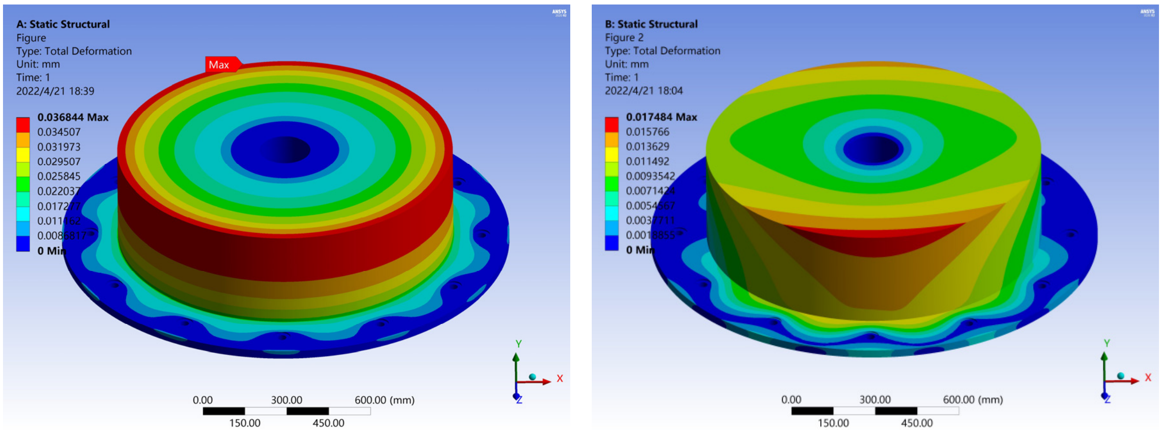


(a) Under the action of the vertical moment of inertia (b) Under the action of vertical force

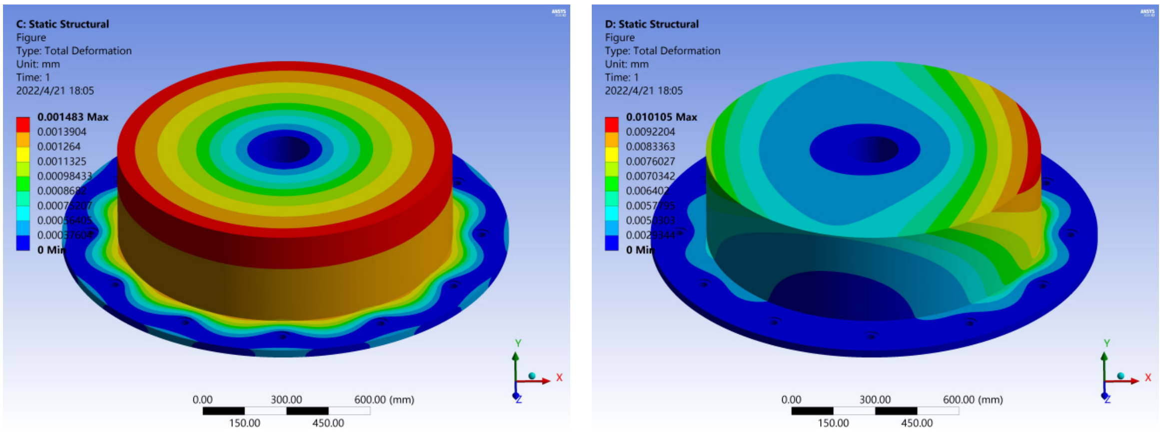


(c) Under the action of the horizontal moment of inertia (d) Under the action of horizontal inertial force

Fig.5 Overall von-Mises stress cloud diagram of the base under different working conditions



(a) Under the action of the vertical moment of inertia (b) Under the action of vertical force



(c) Under the action of the horizontal moment of inertia (d) Under the action of horizontal inertial force
Fig.6 Cloud diagram of the overall total deformation of the base under different working conditions

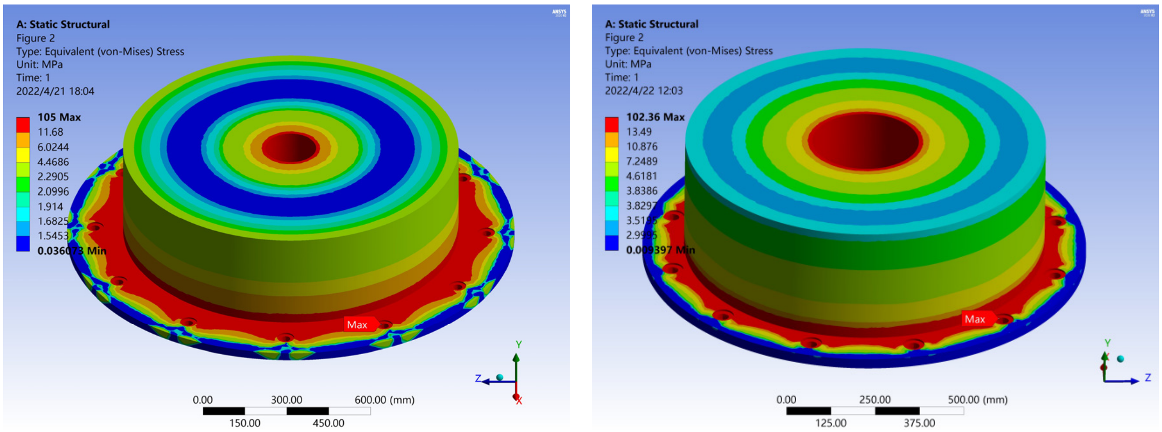
3. Optimum design and analysis of base structure size

3.1. Optimum design of base structure size

From the finite element analysis results of 2.5, it can be seen that the maximum stress of the base is minor in most working conditions. Although the stress and deformation are significant under the sheer inertia moment load, the stress is mainly concentrated in the bolt holes of the lower base plate. To improve the layout of the welding workshop, increase the number of robots, and improve the automation rate of the welding line, the original structure size can be optimized on this basis, and the outer diameter of the cylinder wall and the lower bottom plate can be reduced accordingly. In addition, according to the structural characteristics of the cylinder wall and stress distribution, which reduces its thickness, the diameter of the bottom plate of the improved base is 1,180mm.

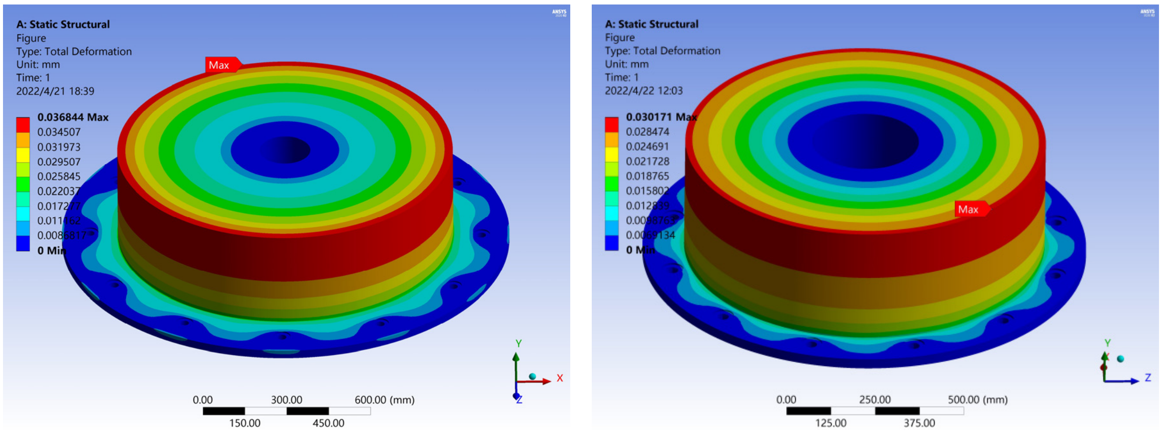
3.2. Finite element analysis results of the new base structure

The same finite element analysis is performed on the new base structure. Figures 7 and 8 are the equivalent stress cloud diagram and total deformation cloud diagram of the base structure before and after the improvement under the action of the vertical inertia moment. It can be seen from the figure that the stress distribution of the improved base under the action of the vertical inertia moment is close to that of the original base, and the position where the maximum stress occurs is the same but reduced to 102.36MPa; the total deformation of the improved base is also close to that of the original base, and the overall deformation is reduced. Small, the position where the maximum deformation occurs is the same, and the maximum value is 0.0302mm. In addition, the low-order modal frequency of the improved base has been increased, the first-order natural frequency has increased to 1348.4Hz, and its mode shape is the same as that of the original base. The comparison of dynamic and static performance before and after optimization is shown in Table 1. The structure size of the robot base is reduced, among which the size of the lower bottom plate has the most significant relative change, and the bolt connection position of the base fixed to the ground also changes relatively. At the same time, the total deformation under different working conditions is slight. There is a reduction, but the intensity and first-order natural frequency are not reduced. It can be seen from the above analysis that the larger the structure size of the robot base is not, the better, and the overall performance of the structure can be improved by rationally designing the structure size and the fixed position of the bolt connection.



(a) Original plan (b) Optimization plan

Fig.7 Overall von-Mises stress cloud diagram of the base under the action of vertical inertia moment



(a) Original plan (b) Optimization plan

Fig.8 Cloud diagram of the total deformation of the base under the action of the vertical inertia moment

Tab.1 The main dimensions and dynamic and static performance of the base structure before and after optimization

Project	Bottom plate size (mm)	Cylinder wall size (mm)	Distance from base plate bolt to center (mm)	Maximumvon-Mises stress (MPa)	Maximum total deformation (mm)	First order frequency (Hz)
Original plan	1650	1200	720	105	0.0368	703.95
Optimization	1180	1012	560	102.36	0.0302	1348.4

4. Conclusion

This paper takes a robot base in a welding workshop as the research object, uses Ansys workbench software to carry out finite element analysis, and proposes an improvement plan for the size of the base structure. Based on meeting the design requirements, the relative size of the base structure is reduced, and the relative dynamic and static performance are better. The research results provide a specific basis for the rational design of the structure size of the welding robot base.

Acknowledgments
This work was financially supported by the Applied Research Project of Youth Innovative Talents of Guangdong Province (2019KQNCX217).

References
[1] Pang Zen. (2004) Finite element analysis and application. Tsinghua University Press, China.
[2] Pengyun Duan, Weibing Wang. (2019) Optimal design of robot base in welding workshop based on finite element analysis.J. China Water Transport, pp.117-118.
[3] Gueta L B, Chiba R, Ota J. Coordinated motion control of a robot arm and a positioning table with arrangement of multiple goals. C.IEEE International conference on robotics and automation, pp.2252-2258.
[4] Suli Li, Zhengying Wei, Bingheng Lu. (2014) Ansys-based stress analysis of spot selding assembly of body flexible parts. Journal of Welding, pp.55-58.
[5] Rui Zen, Lihao Pin. (2007) Workbench-based multi-learning development of scientific simulation analysis platform. Mechatronics Conference.
[6] Taidong Cha, Ping Yang. (2012) Based on ansys workbench optimized design of the fixed bracket. coal mining machinery, pp.28-30.
[7] Chang D, Son D, Lee J. (2012) A new seam-tracking algorithm through characteristic-point detection for a portable welding robot. J. Robotics and Computer-Integrated manufacturing, pp.1-13.
[8] Tianmiao Wang, Yong Tao,Hui Liu. (2018) Current researches and future development trend of intelligent robot.J.International journal of automation and computing, pp.525-546.
[9] Yong Ye, Yanhua Hao, Changhan Zhang. (2004) Structural reliability analysis based on ansys. J. Mechanical engineering and dynamic, pp.63-65.