Finite Element Analysis and Structural Design of Vehicular Manipulator

Yong Gan¹, Qiufeng Zhang², Xu Meng³, Yongsen Wen⁴, Weiqiang Zhu⁵

(School of mechanical and electrical engineering, Guilin University of Electronic Technology, Guilin, China)

*Corresponding author e-mail: 925253380@qq.com

Abstract: Based on the requirement of flexible operation of intelligent car in narrow welding environment, a vehicle-mounted manipulator which is capable of carrying spray gun was designed. In order to make the manipulator have reasonable structure and dynamic characteristics, SolidWorks software is used to carry out three-dimensional modeling, static stress analysis and modal analysis of the vehicle-mounted manipulator. Based on the experimental data, it is concluded that the end of the manipulator is easy to be destroyed. At the same the optimization design of the arm is carried out, such as lightweight, reinforcement of the load-bearing part and so on. The experimental results show that the maximum stress of the load-bearing parts is reduced by about 34.26% after optimization, while the maximum stress of the arm is reduced by 30.3% after optimization, and the mass is reduced by 53.3%. The performance is improved obviously, which provides a theoretical basis for the manufacture of intelligent car platform with thermal stress relief at weld joint.

1. Introduction

The existing thermal stress relief methods are heat treatment, vibration ageing, hammering and shot peening. Shot peening can not eliminate the thermal stress in the weld of components[1]. The application scope of hammering method is limited, the cost of vibration ageing method is high[2], the stress relief annealing operation is simple and the cost is relatively low[3], but the heat treatment furnace used for large components are very heavy because of the large ships at present. Because of the long period, low efficiency and operational risk of stress relief at welded joints of large building structures, the electromagnetic adsorption wall climbing robot based on thermal stress relief at welded joints can be used to improve the structural performance of welded joints and effectively reduce the risk...In summary, a kind of intelligent car based on welding joint thermal stress is designed, which can generate enough adsorption force and maintain steady motion, has broad work coverage, can cross obstacles and climb the wall in complex working environment. It is of great significance to control the temperature by automatic fixed point injection to eliminate the welding thermal stress. In this paper, the mechanical arm is used in the welding thermal stress relief device. The end of the manipulator needs a spray gun of the same quality to spray and burn, which has a wide range of action. In order to make the manipulator work normally under load, the strength and stiffness of the manipulator are tested by static stress analysis and modal analysis, and the optimization design is carried out to further improve the structural stability and motion performance of the whole machine.

2. Structural design of mechanical arm

The structure of the electromagnetic adsorption robot includes matrix adsorption chassis, magnetic...
wheel, telescopic bracket, vacuum sucker and manipulator arm. According to the actual needs, a vehicle-borne mechanical arm is designed, whose base is fixed on the chassis of the car, and the support plate, motor, first arm, secondary arm, elbow joint and bearing parts are connected on the base. The spray gun used to eliminate the thermal stress of the welding joint is mounted on the bearing parts. The intelligent car moves near the weld joint, and the manipulator adjusts the rotation angle according to the working requirements, and reaches the position of the weld joint for fuel injection. The mechanical arm of the original scheme does not have enough strength and stiffness for the mounting of the spray gun. In the structure design of the vehicle-mounted manipulator, the material used in the manipulator must have good yield strength, meet the load and strength at the same time, have certain flexibility and can work normally at high temperature. Therefore, it is proposed to optimize the reinforcement of the load-bearing component.

3. Establishing finite element model

3.1 Establishing 3D model

In this paper, SolidWorks software is used to model the vehicle arm, and the assembly of the vehicle arm is shown in Figure 1,2.

![Figure 1: the structure of Intelligent car](image1)

![Figure 2: Mechanic arm Assembly body](image2)

3.2 Definition of materials

The base of the vehicle-mounted manipulator is fixed on the smart car. The material is defined as the density of aluminum alloy 2700 kg/m³, elastic modulus 72 GPa and Poisson's ratio 0.33. According to the mechanism and theoretical analysis of the manipulator, the arm is the main load component, and the end load component is directly in contact with the spray gun, which is also a sensitive component in the analysis. Therefore, in the case of its own gravity and load, the manipulator is in dangerous condition. The vertical attitude of the arm and the horizontal attitude of the end bearing member are taken as the starting point to study the manipulator. After modeling with SolidWorks, open its own Simulation plug-in for finite element analysis.

According to the actual working conditions, the quality of the spray gun loaded by the vehicle-mounted manipulator is one of the important conditions for analysis, but its stress and deformation are relatively unimportant. At this time, the spray gun can be easily processed into a remote mass and rigidly connected to the bearing surface of the end of the manipulator.

3.3 Grid partition

After applying the material parameters to the three-dimensional model, the upper arm is first subdivided into 3 mm meshes with a ratio of 1.5. Then the total number of nodes is 137171 and the total number of elements is 69658.

3.4 Constrained geometry

Since the solid model of each part of the vehicle-mounted manipulator is rigidly connected, the simulation module automatically adds gravity to each part after applying gravitational load. The force diagram of the vehicle manipulator is shown in Figure 3. The 4 bolts of the base are fixed.
4. Finite element analysis

4.1 static analysis
After solving by simulation finite element analysis, the stress nephogram (Figure 4-a) and strain nephogram (Figure 4-b) of the vehicle-mounted manipulator as shown in the figure are obtained.

(a) Stress nephogram                 (b) Displace nephogram
Figure 4 Stress and Displace nephogram

According to the above figure, the stress of the arm and the bearing parts is larger, the maximum value is about 86.25 Mpa, and the deformation displacement of the bearing parts is about 7.9 mm. Therefore, the bearing parts of the end of the arm are the most easily deformed parts, followed by the end of the arm. This shows that there is still room to improve the strength of the manipulator. Next, using the relationship between stiffness and natural frequency, the stiffness of the manipulator is analyzed and verified to obtain the optimal structure design.

4.2 Modality analysis
The three-dimensional model of the vehicle-mounted manipulator is solved by applying boundary conditions and meshing the same conditions according to the conditions in the static stress example. The frequencies of the first five modes are shown in Figure 5.
According to the above finite element analysis results, the deformation of the frequency at 36Hz is large, and the amplitude at 19Hz is the largest. In order to meet the working needs, the above frequency range should be avoided when the manipulator works. According to the practical experience, the end bearing parts of the vehicle-mounted manipulator are easy to be destroyed, so it is necessary to analyze and optimize them separately.

5. Structure analysis and optimization of secondary arm and loader

5.1 Optimization solution

Through static stress analysis and modal analysis, it can be seen that the right angle position of the end bearing parts and the bearing surface are easily deformed. According to material mechanics, all the parts designed are brittle materials, and the effect of stress concentration should be considered. The local increase of stress caused by the sharp change of section is called stress concentration, and its degree is expressed by stress concentration factor $K$.

$$K = \frac{\sigma_{\text{max}}}{\sigma_n}, \quad \sigma_n = \frac{F}{(b-d)\delta}$$  \hspace{1cm} (1)
It can be seen that the smaller the radius of the fillet, the greater the stress concentration factor, and the stress concentration phenomenon is easy to occur at the right angle. The moment \( M \) is equal to the product of force and arm. The longer the arm is, the greater the moment is. In the original scheme, the bolt hole is far from the center of gravity of the whole arm, which easily causes greater stress to bend the arm, and the utilization of ladder shape space with narrow upper and lower bottom and wide bottom is low. According to the actual processing situation, the optimization scheme for the end bearing parts and the small arms is proposed. The bearing surface of load-bearing part is changed from bottom to top, and the position of right-angle stress concentration is strengthened by reinforcing bars to form a triangular frame shape, so as to increase the strength of the joint surface and make the spray gun more stable. For the arm, the position of the two bolt holes causes greater moment, which is easy to cause stress concentration and deformation, which is not conducive to the activity of the manipulator. The distance between the short force and the center of gravity and the volume of the forearm are reduced. The shape of lower arm bottom width and narrower upper arm bottom is not conducive to acceptance, so it is changed into ellipse shape, which makes the mechanism more compact and improves the space utilization rate. The solid solution treatment and artificial aging of raw materials make its performance more superior.

5.2 Experimental Verification
A new static stress example of the optimized structure is built to solve the problem under the same boundary conditions. As shown in Figure 6,7. The results show that the maximum stress value of the load-bearing part is 56.7 Mpa. As table 2 present that load-bearing part is about 71.65% lower than before, while the maximum stress value of the arm after optimization is 30.3% lower than before, and the weight is reduced by 53.3%.

![Figure 6. Stress after optimization](image6.png)

![Figure 7. Stress after optimization](image7.png)

| Parts     | Original solution/Mpa | After optimization/Mpa | Reduction/% | Scheme 5 |
|-----------|------------------------|------------------------|-------------|---------|
| Loader    | 86.25                  | 56.70                  | 34.26       | 123     |
| Secondary | 39.00                  | 27.61                  | 30.30       | 644     |

5.3 Comparison of Rigidity before and after optimization
According to the research of YangZhijun\(^\text{[4]}\), the equivalent rigidity \( K \) of the structure is composed of bending rigidity and tension rigidity.

\[
K = K_B + K_T = \frac{F}{W_F} + \frac{c \pi^2 N}{L}
\]

The corresponding natural frequency of the vibration system is equivalent to that of the bending mass under the bending mode.

\[
f = \frac{1}{2\pi} \sqrt{\frac{K_B + K_T}{c \rho A L}} = \frac{1}{2\pi} \sqrt{\frac{K_B}{c \rho A L} + \frac{\pi^2 N}{\rho A L}}
\]
Since the dynamic stiffness is proportional to the quadratic of natural frequency\(^5\), the load on the bearing parts is relatively small. Only the lower order modes need to be considered. The proportion of higher order modal energy is too low to affect the vibration of the whole structure, and the external excitation is difficult to reach the higher order natural frequency. Therefore, the stiffness change of the end bearing member can be verified by the natural frequency. The multimodal analysis of the model before and after optimization is compared with that shown in table 3.

| Vibration mode | Original solution/Hz | After optimization/Hz | Reduction/% |
|----------------|----------------------|-----------------------|-------------|
| 1              | 173.027716           | 12084.6049            | 68.84201826 |
| 2              | 456.121449           | 21051.1081            | 45.15241872 |
| 3              | 456.121449           | 176005.4209           | 20.23218861 |
| 4              | 63837.0756           | 7775174.56            | 120.7971608 |
| 5              | 377204.7889          | 23380126.09           | 60.98258023 |
| 6              | 3920400              | 35797485.61           | 8.131079892 |

By comparing the natural frequencies before and after optimization, the Rigidity of the optimized structure is improved obviously.

6. Conclusion
After the structural design of the vehicle-mounted manipulator is completed, the static stress analysis and modal analysis are carried out by using finite element analysis. Aiming at the reinforcement treatment of bearing parts and the shape optimization of secondary arm structure, and verified by strength and stiffness check, the maximum stress value of the optimized bearing parts is reduced by about 71.65% compared with the original scheme, and the structural stiffness is obviously improved, while the maximum stress value of the secondary arm after optimization is reduced by 30.3% compared with that before, and the lightweight requirement is achieved, which shows that the structure of the manipulator arm has good performance. The results show that the structure of the manipulator has good stability. As an intelligent car carrying module, it provides a necessary precondition for the thermal stress relief of the weld joint.

Acknowledgments
Thank my tutor for his guidance in this project, and my classmates for their help and encouragement in the experiment. This work was financially supported by National Nature Fund (51665008) Research on Nondestructive Measurement and reconstruction of mechanical parts based on grid slice volume measurement and supported by Undergraduate Innovation and entrepreneurship training program project (201810595013) (201810595193). It belongs to Innovation Project of GUET Graduate Education 2018YJ CX10.

References
[1] Hou Guoqing, Lu Wei, Qin Jian, Li Jiang Li, Wang Guoan. Application of vibration aging and ultrasonic shock in eliminating welding residual stress in front frame of loader [J]. Equipment Manufacturing Technology, 2017 (09): 99-101.
[2] Sun Youping, Zhang Yangyang, Tu Yao-yao, Han Jun, Li Wangzhen. Effect of stress removal annealing on residual stress and Mechanical Properties of welded joints of 310S Steel [J]. Welding Technology, 2015N 44 (02): 34-37 6.
[3] Yuan Hongzhi, Feng Xiaoyu. Study on the process parameters of annealing without stress [J]. Heat treatment Technology and equipment, 2014N 35 (02): 48-50.
[4] Zhijun Yang, Youdun Bai, Xin Chen, et al. Approximate analytical solution of equivalent stiffness and natural frequency of prestressed beams [J].China Science: Physical Mechanics Astronomy, 2015, 45 (07): 98-104.
[5] Shuwan Cui, Jianjun Wei. Finite element analysis and structural optimization of welding robot [J]. Manufacturing automation, 2014, 36 (20): 67-70.