Modeling and Simulation of Stress and Strain of Chassis Structure for Wheeled Robot

Lan-lan YUAN¹, Yuan-jin FANG², Xiao-xiao LU³, Ang CHEN⁴, Zhi-chao REN⁵, Yong-bo ZHONG⁶, Ye LANG⁷, Bin WANG⁸, Jia-zheng ZHAO⁹, Chao WU¹⁰ and Lu-han MA¹¹

Mechanical and Electronic Engineering Department of China University of Mining and Technology (Beijing), Beijing, China

Keywords: Robot, Chassis structure, Stress, Strain, Simulation.

Abstract. The three-dimensional solid model of the chassis which is the support structure for the wheeled robot's load and power parts is established. Based on the theory of elasticity, the relation equations of the three direction stress and strain of the chassis structure are given. The calculation analysis and simulation are completed by the finite element method, and the structural stress and strain of the chassis under loads and driving forces are obtained. Through simulating and analyzing the stress, strain and deformation values and distribution on the chassis structure, it makes a basis for the design and research of the chassis with light weighted and appropriate stress and deformation.

Introduction

On the chassis of the wheeled robot installed many driving units, such as the forward, backward, steering parts, and so on, and the chassis almost bears the weight of the whole robot. It is an important part of the robot. The stress and deformation of the chassis structure under loads determines its strength and stiffness, thus affecting the structural shape, volume and weight of the chassis. Many domestic and foreign scholars have studied this issue. Yunxia Wang, the Xi'an Jiaotong University, carried out the stiffness analysis of the researched robot chassis, but only extracted the total deformation image and the vertical direction deformation image. She did not pay attention to the stress and strain situation [1]. Huihui Gao of Shanghai University has analyzed the stability of the studied robot chassis under three working conditions. By extracting the stress and deformation cloud charts of the chassis, the dangerous position of the structure was found and the improvement suggestions was put forward, but the concrete improvement measures were not given [2]. The Hai Zhou of Nanjing Institute of Technology has carried out the finite element static analysis to the chassis of the harvester. By observing the stress and deformation cloud charts, the structure are light weighted, but no specific improvement processes were given[3]. In this paper, the three-dimensional solid model of the robot chassis is set up first, and the stress, strain and deformation are simulated by elasticity and finite element method to provide the basis for the design of the chassis structure.

The Three-dimensional Modeling and Force Analysis of the Robot Chassis

A three-dimensional model of the chassis structure is set up, shown in Figure 1.

Figure 1. The three-dimensional model of the chassis.

Figure 2. Force diagram.
According to the actual working condition of the chassis, boundary conditions are found. The loads are the gravity, the pressure from all parts above the chassis, the bearing loads at the two bearing seat holes, and the bearing load at the universal wheel support hole. The constraints are limitations of six degrees of freedom from bearings and the universal wheel to the chassis respectively. The forces on the chassis are shown in Figure 2.

**Elasticity Equations of Structural Stress and Deformation**

According to the basic theory of elasticity, all points in the robot chassis structure satisfy the equilibrium equation, geometric equation and physical equation along the axis x, y, and z, as shown in formula (1) (2) (3)[4].

\[ A\sigma + \bar{f} = 0 \]  

Where, \( A \) is the differential operator, \( \bar{f} \) is the volume force vector, \( \sigma \) is the stress vector.

\[
A = \begin{bmatrix}
\frac{\partial}{\partial x} & 0 & 0 & \frac{\partial}{\partial y} & 0 & \frac{\partial}{\partial z} \\
0 & \frac{\partial}{\partial y} & 0 & \frac{\partial}{\partial x} & \frac{\partial}{\partial z} & 0 \\
0 & 0 & \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial y} & \frac{\partial}{\partial x}
\end{bmatrix}
\]

\[
\sigma = \begin{bmatrix}
\sigma_x \\
\sigma_y \\
\sigma_z \\
\tau_{xy} \\
\tau_{yz} \\
\tau_{zx}
\end{bmatrix}
\]

\[
\bar{f}^T = \begin{bmatrix}
f_x \\
f_y \\
f_z
\end{bmatrix}
\]

\[
\varepsilon = Lu
\]

Where, \( \varepsilon \) is the strain vector, \( L \) is the differential operator, \( u \) is the displacement vector.

\[
\varepsilon = \begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\varepsilon_z \\
\gamma_{xy} \\
\gamma_{yx} \\
\gamma_{yz} \\
\gamma_{zx}
\end{bmatrix}
\]

\[
\varepsilon = \begin{bmatrix}
\varepsilon_x & \varepsilon_y & \varepsilon_z & \gamma_{xy} & \gamma_{yx} & \gamma_{yz} & \gamma_{zx}
\end{bmatrix}^T
\]

\[
u = \begin{bmatrix} u \\ v \\ w \end{bmatrix}
\]

\[
L = A^T
\]

\[
\sigma = D\varepsilon
\]

In this paper, Q235 steel is used as the material for the robot chassis. Its modulus of elasticity \( E=211\text{MPa} \) and Poisson ratio \( \nu=0.3 \)[5]. The elastic matrix in the formula (3) is as follows:
Finite Element Calculation

It is difficult to solve the equations of elasticity, so the finite element method is used. The chassis structure is discretized. 48593 nodes and 18572 elements are obtained. The result is shown in Figure 3. As shown in figure 3, eight nodes hexahedron element is used in the position of the supporting plate of the chassis and the supporting side of the bearing seat, and the four node tetrahedron element is adopted in other parts.

The displacement matrix of the four node tetrahedral element is as follows[4]:

\[
\mathbf{u} = \mathbf{N} \mathbf{a}^e
\]

Where, \(\mathbf{N}\) is the interpolated function matrix, \(\mathbf{a}^e\) is the displacement matrix of unit nodes.

\[
\begin{align*}
N_i &= \frac{1}{6V}(a_i + b_i x + c_i y + d_i z) \\
N_j &= -\frac{1}{6V}(a_j + b_j x + c_j y + d_j z) \\
N_m &= \frac{1}{6V}(a_m + b_m x + c_m y + d_m z) \\
N_l &= -\frac{1}{6V}(a_l + b_l x + c_l y + d_l z)
\end{align*}
\]

\[
\mathbf{a}^e = \begin{bmatrix}
u_i & v_i & w_i & u_j & v_j & w_j & u_m & v_m & w_m & u_l & v_l & w_l
\end{bmatrix}
\]

The strain matrix of the four node tetrahedral element is as follows[4]:

\[
\mathbf{\varepsilon} = \mathbf{B} \mathbf{a}^e = \begin{bmatrix}
\mathbf{B}_i & -\mathbf{B}_j & \mathbf{B}_m & -\mathbf{B}_l
\end{bmatrix} \mathbf{a}^e
\]

The stiffness matrix of the four node tetrahedral element is as follows[4]:

\[
\mathbf{K}^e = \int_V \mathbf{B}^T \mathbf{D} \mathbf{B} dV = \mathbf{B}^T \mathbf{D} \mathbf{B} = \begin{bmatrix}
\mathbf{K}_{ii} & -\mathbf{K}_{ij} & \mathbf{K}_{im} & -\mathbf{K}_{il} \\
-\mathbf{K}_{ji} & \mathbf{K}_{jj} & -\mathbf{K}_{jm} & \mathbf{K}_{jl} \\
\mathbf{K}_{mi} & -\mathbf{K}_{mj} & \mathbf{K}_{mm} & -\mathbf{K}_{ml} \\
-\mathbf{K}_{li} & \mathbf{K}_{lj} & -\mathbf{K}_{lm} & \mathbf{K}_{ll}
\end{bmatrix}
\]

Among them, any block can be calculated by the next formula:
\[ \mathbf{K}_{rs} = \mathbf{B}_r^T \mathbf{D} \mathbf{B}_s V = \frac{E(1-\nu)}{36V(1+\nu)(1-2\nu)} \begin{bmatrix} K_1 & K_4 & K_7 \\ K_2 & K_5 & K_8 \\ K_3 & K_6 & K_9 \end{bmatrix}, \quad (r, s = i, j, m, l) \]

Where,
\[ K_1 = b_s b_s + A_2(c_s c_s + d_s d_s), \quad K_2 = A_1(b_s c_s + A_2 b_s c_s), \quad K_3 = A_1 d_s b_s + A_2 b_s d_s, \quad K_4 = A_1 b_s c_s + A_2 c_s b_s \]
\[ K_5 = c_s c_s + A_2(b_s b_s + d_s d_s), \quad K_6 = A_1 d_s c_s + A_2 c_s d_s, \quad K_7 = A_1 b_s d_s + A_2 d_s b_s, \quad K_8 = A_1 c_s d_s + A_2 d_s c_s \]
\[ K_9 = d_s d_s + A_2(b_s b_s + c_s c_s), \quad A_1 = \frac{\nu}{1-\nu} = \frac{3}{7}, \quad A_2 = \frac{1-2\nu}{2(1-\nu)} = \frac{2}{7}. \]

The element displacement matrix, the element strain matrix, the element stiffness matrix of the eight nodes hexahedral element are similar to those of the four nodes tetrahedral element. This paper is not to give unnecessary details.

The integration of the overall stiffness matrix of the structure:
\[ \mathbf{K} = \sum_e \mathbf{G}^T \mathbf{K}^e \mathbf{G} \] (7)

Where, \( \mathbf{G} = \mathbf{a}^T \mathbf{a}^{-1} \).

The integration of the overall load array of the structure:
\[ \mathbf{P} = \sum_e \mathbf{G}^T \mathbf{P}^e \] (8)

The finite element solution equation based on the minimum potential energy principle is as follows:
\[ \mathbf{K} \mathbf{a} = \mathbf{P} \] (9)

The displacement boundary condition is introduced. According to the force analysis of the previous section, it is known that the displacement boundary conditions of the robot chassis are the six degree of freedom full constraints on the two bearing seat holes and the universal wheel bracket holes. That is, displacements of all the nodes in these parts are set to be zero \( a_i = 0 \). We set up \( a_i \) as the undetermined node displacement, \( a_k \) for the known node displacement. Recombining the finite element equation established by the minimum potential energy principle.
\[
\begin{bmatrix}
K_{aa} & K_{ab} \\
K_{ba} & K_{bb}
\end{bmatrix}
\begin{bmatrix}
a_a \\
a_b
\end{bmatrix}
=
\begin{bmatrix}
P_a \\
P_b
\end{bmatrix},
\]
(10)

Plugging \( a_b = 0 \), the equation's order is descended as follows:
\[
\begin{bmatrix}
K_{aa} & a_a = P_a \\
K_{ba} & a_a = P_b
\end{bmatrix}
\]
(11)

The load boundary condition is introduced. According to the force analysis carried out in the previous section, there are 5 loads on the robot chassis.

The gravity load is considered. The weight of the robot chassis is about 5Kg, so the overall gravity is about 49N. Known the element number, the gravity on each element is about 2.64e-3N, that is
\[ \mathbf{P}_f = [0 \quad 0 \quad 2.64e-3]^T. \]
Considering the pressure 254N of all parts on the top of the chassis. There are 2441 elements on the loading surface of the chassis board, and the average force on each element is about 0.104N. That is, the load array of all the loading elements is \( \mathbf{P} = \begin{bmatrix} 0 & 0 & 0.02884 \end{bmatrix}^T \).

Consider the bearing loads 90N on the two bearing holes and the bracket hole of the universal wheel. In ANSYS Workbench, when the bearing load on the surface of the cylinder is applied, the radial load component is obtained by distributing the pressure according to the projection area, and the axial load component is gotten by evenly distributing the pressure along the circle. Setting the load array of all loading elements as \( \mathbf{P}_s \).

The overall load array of the robot chassis is as follows:

\[
\mathbf{P} = \sum \mathbf{G}^T (\mathbf{P}_f + \mathbf{P}_s + \mathbf{P}_s')
\]

(12)

Simulation of the Stress and Strain of the Chassis

The equivalent stress, equivalent strain and total deformation cloud charts are extracted from the results of the finite element static analysis. They are shown in Figure 3.

![Figure 3. The cloud charts of the three parameters.](image)

Observing Figure 4, we can see that the maximum equivalent stress, the maximum equivalent strain and the maximum total deformation of the robot chassis are 24.032MPa, 0.00012016 and 0.13862mm respectively, which are all in the allowable range. The stress, strain and deformation on the bearing seat holes are very small and redundant, so the thickness of this part is reduced from 27mm to 18mm preliminarily. In order to analyze the stress change on the disc structure, several sections were selected at key locations. Its stress and deformation are shown in Figure 4.

The finite element analysis is carried out for the new structure after weight reduction. The mesh method and the boundary condition are kept unchanged. The maximum equivalent stress and the maximum total deformation of the improved robot chassis are 28.1MPa and 0.53mm respectively. Under the condition of satisfying the performance requirements, the weight is reduced by about 26%.

203
Summary

(1) Aiming at the three-dimensional solid model of the robot chassis structure, the relation equations of stress, strain, displacement and force are established based on the elasticity. And the element displacement, strain and stiffness matrix of the finite element are also established.

(2) The finite element method is used to calculate, analyze and simulate the stress, strain and deformation of the chassis structure of the robot. Then the redundancy of the original structure is found, and a new structure design scheme is proposed.

(3) The simulation analysis of the new chassis structure shows that the maximum equivalent stress and the maximum total deformation are 28.1MPa and 0.53mm respectively. Under the condition of satisfying the performance requirements, the weight is reduced by about 26%.

Acknowledgement

This research is supported by "Foundation of Central Universities of Education Department of China" No. 2010YJ04. Authors would like to acknowledge and express their thanks for the support.

References


