Strength analysis and method research of a light truck powertrain mounting system

Yudong Lai¹³, Shiping Sun²*, Xianzhong Yu¹³, Hexing Wu¹³, Huang Hui¹³, Xing Qiu¹³

¹ Product development center, Jiangling motors Co., Ltd, Nanchang, Jiangxi, 330013 China;
² School of Aeronautical Manufacturing Engineering, Nanchang Hangkong University, Nanchang, Jiangxi, 330063 China;
³ Jiangxi Province Key Laboratory of Vehicle Noise and Vibration (Jiangling Motors Co., Ltd), Nanchang, Jiangxi, 330013, China;
*Corresponding author’s e-mail: shipingsun@163.com

Abstract. This paper presents an integral strength checking method to solve the problem that the step-by-step strength checking method cannot check the strength of metal parts in the rubber mounting system of light truck powertrain. By establishing an integral finite element model including powertrain and rubber mount system and considering the rubber bearing and force transfer mode, the motion coordination and overall analysis of mount system and powertrain are realized. The calculation results show that the proposed method can easily and accurately complete the strength check of metal parts of the mounting system, which provides a reference for the structural design of the mounting system.

1. Introduction
The mounting system of light truck powertrain (PT) is one of the key components of the vehicle. The rubber mounting scheme composed of metal parts and rubber parts is widely used, in which the metal parts play the role of support and limit to ensure vehicle safety, while the rubber parts play the role of vibration and energy absorption and contribute to vehicle comfort. The mounting system is generally arranged with multiple mounting points, commonly 3-point or 4-point arrangement, and each mounting point includes the active side connecting the PT and the passive side connecting the frame. During the actual use of the vehicle, severe load conditions may lead to wear or fracture failure of metal parts or rubber, which requires the mounting system to have sufficient strength performance.

However, at present, the research on rubber mounting system mainly focuses on NVH performance[1-2] and rubber strength durability[3-4]. The research on the strength performance of metal parts mostly takes the mounting bracket as the research object[5-6], while there are few reports on the strength performance of metal parts of the mounting itself. The reason is that the current strength analysis method of mounting system based on CAE needs to first obtain the load force at the elastic centre of the mounting point[7], and take the elastic centre point as the loading point to check the strength of the active and passive side components of each mounting point respectively. When the mounting system works, nonlinear mechanical behaviours such as limit contact and large deformation often occur. At this time, the active and passive side components are checked respectively, and the analysis results cannot effectively reflect the stress distribution of the mounting parts (metal and
rubber) when the limit occurs, so its strength performance cannot be checked accurately. To solve this problem, a strength finite element analysis (FEA) method of metal parts of integral mounting system including PT and mounting system is proposed in this paper. This method fully considers the material, geometry and boundary nonlinear factors in the work process of the mounting system. Through the integral modelling method, the motion coordination between the PT and the mounting system is realized, and the analysis results are more real and reliable.

2. Integral analysis method

2.1. Integral analysis method overview
The load of the mounting system mainly comes from the PT’s dynamic work load and the inertial force load during vehicle driving under acceleration and deceleration, turning, road bumps, collision and so on. The strength check condition system in engineering practice has covered the above inertial force load and self-dynamic load[7]. Taking the front mounting point of a light truck 4-point rubber mounting system as an example, its structure is shown in Figure 1. The active side 4 of the mounting is connected with the mounting support 2, the passive side 5 is connected with the mounting support 3, and the active side and the passive side are connected through vulcanized rubber 6.

Figure 1. Structural diagram of a light truck PT mounting system

According to the general step-by-step analysis method, the strength check of the mounting system needs to be completed in two steps. As shown in Figure 2. Firstly, the multibody dynamics model is used to extract the working condition load, and then the finite element analysis models of the active and passive sides are established respectively, and finally the strength check is completed. In fact, the existing strength check conditions can be applied in multibody dynamics and finite element simulation. Therefore, the steps of extracting the load from multi-body dynamics can be omitted, the integral analysis model can be established directly in the finite element analysis, and various factors affecting the analysis results can be taken into account by using the nonlinear advantages of the finite element method, realize the strength check of the metal parts of the mounting system.

Figure 2. Mounting system analysis process: a) step by step method, b) integrated method
2.2. Modeling idea of integral analytical model
The integral analysis model consists of the PT and mounting system. As the main research object, the mounting system would establish a detailed finite element model according to the design scheme. The PT has high stiffness and large mass. It is simplified as a mass point located at the mass centre of the PT and rigidly connected with the suspension system. Thus, the integral analysis model is obtained to realize the motion coordination between the PT and the mounting system.

Meanwhile, the material nonlinearity of the metal part is considered, and the rubber material is set as the hyperelastic constitutive model. In order to fully consider the force transfer between rubber and rubber, rubber and metal parts when the suspension system is limited under extreme working conditions, the contact algorithm is adopted for the whole model. Therefore, the material, geometry and boundary nonlinear factors of the mounting system in the work process are considered in this model, which can more accurately reflect the stress and deformation state of metal parts and rubber.

2.3. Rubber constitutive model
Rubber material is a hyperelastic material with nonlinear, isotropic and nearly incompressible properties. Its stress-strain relationship is very complex[8]. Establishing an appropriate rubber constitutive model to describe the mechanical behaviour of rubber is very important for the accuracy of simulation results. Mooney-Rivlin constitutive model is a widely used constitutive model of rubber materials in engineering. And a large number of engineering practice also proves the validity and accuracy of the model[9-10].The expression is:

\[ W = C_{10}(I_1 - 3) + C_{01}(I_2 - 3) \]  

Where \( W \) is the strain energy density function, \( I_1 \) and \( I_2 \) are the deformation tensor invariants, \( C_{10} \) and \( C_{01} \) are the rubber material constants. The relationship between rubber material constant and elastic modulus \( E \) is:

\[ E = 6C_{10}(1 + C_{01}/C_{10}) \]  

The key to the application of this model is to determine the rubber material constants \( C_{10} \) and \( C_{01} \). Usually, these two parameters need to be calculated through the data of multiaxial tensile test. This process has long cycle and high cost, while the measurement of rubber hardness is relatively easy and fast. The research shows that, rubber hardness has significant correlation with elastic modulus and material constant, and the fitting formula is given[11]:

\[ \log E = 0.0198H_r - 0.5432 \]  

Where \( E \) represents the elastic modulus of rubber material and \( H_r \) represents the IRHD hardness of rubber. It can be seen that \( C_{10} \) and \( C_{01} \) can be calculated after the ratio \( C_{10}/C_{01} \) of rubber material constant is determined. According to the conclusion of reference 11, the finite element results are in good agreement with the experimental results when \( C_{10}/C_{01} = 0.05 \).

3. Establishment of finite element model
Taking the strength analysis of a light truck PT mounting system as an example, an integral FEA model is established, as shown in Figure 3. The metal part and rubber are modeled by hexahedron element. The rubber and metal part elements share nodes to simulate the connection relationship between rubber and metal. The element size is 3mm. Other parts are modeled by shell element with the element size of 8mm. The PT weighs 403kg and is simplified as a mass point, which is located at the center of mass of the PT. Taking the mass point as the independent node, the center of the mounting hole on the four mounting points and the torque output point of the PT as the dependent node, a rigid element is established to form an integral analysis model.
Figure 3. Schematic diagram of integral FEA model of mounting system

The IRHD hardness of rubber is 60, and the elastic modulus $E=4.4$ MPa can be calculated by entering formula (3). Assuming $C_{10}/C_{01}=0.05$, bring it into formula (2) to calculate $C_{10}=0.7$ and $C_{01}=0.035$. The material brand of metal parts is SAPH400, its yield strength is 225 MPa, tensile strength is 550 MPa, and the stress-strain curve is shown in the Figure 4. The general contact is defined for the mounting system, the friction coefficient between metal and rubber is 0.7, the general contact between the passive side of the mount and the mount support is defined, and the friction coefficient is 0.15.

Figure 4. Stress-strain curve of SAPH400 material.

Fixed the bolt holes connecting the mount support and the frame, and simulate the installation state of the mount relative to the frame, as shown in Figure 3. Among the 28 load conditions listed in reference 7, 8 analysis load conditions are selected according to the severity of each direction, as shown in Table 1. For 5 normal load conditions, the stress is less than the material yield strength as the target, and for 3 extreme load conditions, the maximum equivalent plastic strain (PEEQ) is less than 0.01 as the target.

Table 1. List of strength analysis conditions of mounting system.

| Gravity load /g | Engine output torque /N.mm | Load type | Target               |
|-----------------|---------------------------|-----------|---------------------|
| X               | Y                         | Z         |                     |
| CASE1           | 0.6                       | -1        | 3138975             |
| CASE2           | 1                         | -1        | 3138975             |
| CASE3           | -3                        | 3138975   | Normal              |
| CASE4           | -4.5                      | 3138975   | Stress<225MPa       |
| CASE5           | -3                        | -1        |                      |
| CASE6           | -11                       | -1        |                      |
| CASE7           | 3                         | -6        | Extreme              |
| CASE8           | -1                        | 4932675   | PEEQ<0.01           |

4. Finite element analysis and results

4.1. Integrated model verification

Using ABAQUS implicit solver, taking CASE6 and CASE8 as examples, serious contact behavior occurred at the active and passive sides of the mount, as shown in Figure 5. CASE6 mainly focuses on the inertia force in - X direction. The PT will produce large displacement movement in - X direction.
Because the passive side is fixed, the active side will move in the -X direction with the PT. It can be seen from the figure that the x-direction limit designed at the passive side has worked and the rubber in the contact area is extruded and deformed. CASE8 is mainly based on the maximum output torque load of the engine. The PT will rotate in the direction of the crankshaft axis. At this time, the active side will rotate with the PT in the direction of the arrow in the figure, and the rubber in the contact area has been extruded and deformed. It can be seen that the integral analysis method can well describe the limiting effect of the mounting system and the protective effect of rubber on the metal part in the limiting process. At the same time, it also shows that the results obtained by the integral method are closer to the actual state.

Figure 5. Limit contact area of Mount 2 under CASE6 and CASE8

4.2. Strength analysis results of metal parts

It can be seen from Table 2 that in normal conditions CASE1-5, the maximum stress of the metal part at the four mounting points is less than the yield strength of the material. In extreme conditions CASE6-8, CASE7 and 8 meet the target. Under CASE6, the PEEQ at the active side of the front mount reaches 0.025, exceeding the target value of 0.01. As shown in Figure 6(a), it occurs in the x-direction contact area between the active side and the passive side. Because it is a fillet area, it belongs to the transition area with characteristic changes. In fact, there is a high probability of cracking here. In view of this risk, it is proposed to upgrade the material brand from SAPH400 to SAPH440. After replacing the material, the PEEQ in this area decreased to 0.08, as shown in Figure 6(b), meeting the target and eliminating the risk of cracking. After the implementation of the optimization scheme, the sample vehicle successfully passed the comprehensive road durability test of 50000 km.

| CASE  | Mount 1 Stress | PEEQ | Mount 2 Stress | PEEQ | Mount 3 Stress | PEEQ | Mount 4 Stress | PEEQ | Target |
|-------|----------------|------|----------------|------|----------------|------|----------------|------|--------|
| CASE1 | 199.4          | 146  | 192.7          | 100.3|                |      |                |      | Stress<225MPa |
| CASE2 | 204.6          | 147.4| 205.9          | 105.5|                |      |                |      |        |
| CASE3 | 233.8          | 17.6 | 234            | 65.4 |                |      |                |      |        |
| CASE4 | 161.2          | 197.1| 218.5          | 204.4|                |      |                |      |        |
| CASE5 | 158.8          | 153.3| 155.6          | 162.1|                |      |                |      |        |
| CASE6 | 0.022          | 0.025| 0.005          | 0.006|                |      |                |      | PEEQ<0.01 |
| CASE7 | 0.00007        | 0.004| 0.002          | 0    |                |      |                |      |        |
| CASE8 | 0.0004         | 0.0042| 0             | 0    |                |      |                |      |        |
6. Conclusion
Taking the strength analysis of a light truck PT mounting system as an example, an integral finite
element modelling and integral verification analysis method of PT mounting system which can more
truly reflect the stress and deformation state of the mounting system is proposed. The strength analysis
results consistent with the working condition setting are obtained, the strength risk is eliminated, and
the durability road test is passed. Compared with the step-by-step method, the integral method has the
advantages of high analysis accuracy and comprehensive investigation, and has higher guiding
significance for the structural design of the mounting system.

Acknowledgments
This work was supported by the National natural science foundation of China (Grant Number
11862015)

References
[1] Pan, G., Fu, B., Wang, G. et al. (2021) Optimization design of a powertrain mount-system based
on the TPA and genetic algorithm. Journal of Vibration and Shock, 40: 279-286.
[2] Sun, Y., Duan, P., Liu, F. et al. (2020) Design and optimization of nonlinear stiffness for
powertrain mount System. Noise and Vibration Control, 40: 148-153.
[3] Ye, B., Duan, X., Huang, X. et al. (2012) Structure fatigue design on rubber mount of automobile
power train. Computer Aided Engineering, 21: 61-64.
[4] Wang, W., Shangguan, W., Duan, X. et al. (2012) Study on prediction of fatigue life of rubber
mount based on linear cumulative fatigue damage theory. Journal of Mechanical Engineering,
48: 56-65.
[5] Zhao, L., Zhou, Z., Huang, J. et al. (2016) Performance calculation and structure design for
vehicle Powertrain mounting bracket. Journal of Shenyang Aerospace University, 33: 38-46.
[6] Zhang, L., Zhao, Q., Zhang, H. et al. (2017) Multi-objective topology optimization for the mount
bracket of vehicle powertrain. Automotive Engineering, 39: 551-555.
[7] Guo, S., Li, J., Xiong, F. et al. (2016) A research on structure optimization of engine mount
bracket based on misuse working condition. Automotive Engineering, 38: 1220-1227.
[8] Yang, X. (1999) Finite element method for nonlinear rubber materials. Petroleum Press, Beijing.
[9] Li, X., Yang, X. (2005) A review of elastic constitutive model for rubber materials. Elastomerics,
15: 50-58.
[10] Zheng, M., Xie, J. (2001) Finite element analysis of large deformation of compressed rubber
component. Journal of Northern Jiaotong University, 25: 76-79.