Simulation Analysis of Thermoplastic Composites on Suspension Brackets of Electric Commercial Vehicles

Haijun Zhao 1 *, Haixia Wang2 *, Naibin Zhai 1, Hongliang Li 3, Zhiqiang Wang 1, Feng Wang 1
1 College of Automobile and Transportation, Tianjin University of Technology and Education, Tianjin, 300222, China
2 Changshu Institute of Technology, Changshu, Jiangsu, 215500
3 Automotive Engineering Research Institute, China Automotive technology & Research Center, Tianjin, 300300, China
* Corresponding author’s e-mail: hjzhaoly@lit.edu.cn

Abstract. Lightweight is more important in electric commercial vehicles, the static simulation analysis is first carried out on the suspension system bracket using the material of steel under three working conditions in this paper. Taking the suspension system bracket connection plate as an example, the bracket connecting plate model of the suspension system is established with thermoplastic composite materials, which is nine-layer three-dimensional model. The pre-processing, solving and post-processing procedures of the stress analysis of the part are introduced, and the stress distribution of different layers is obtained. Results show that the maximum stress of the steel bracket of the suspension system under three working conditions of triple gravity, acceleration of gravity adding steering inertia, acceleration of gravity adding braking inertia are 189.22, 92.48, and 156.9Mpa for steel, which are all less than the ultimate stress of 235Mpa. The maximum stress of each layer of the thermoplastic composite connecting plate meets the design requirements. In the case of the same ply angle, the bottom layer will bear less and less stress, the base layer will bear very little stress, and the main stress will be borne by the reinforcing material, support is provided for the application of thermoplastic composite materials on automobile chassis.

1. Introduction
Body weight of traditional energy vehicles consumes about 70% of fuel. If vehicle weight is reduced by 10%, fuel consumption can be reduced by 6%-8% [1]. Lightweight is an important method to improve fuel economy and reduce pollutants and carbon emissions. Driven by national policies and regulations and the fierce market competition environment, lightweight design of automobile has become an inevitable trend in automobile development. The body weight reduction design can be realized through three dimensions, which are structural optimization design, the use of advanced manufacturing technology and new lightweight materials. It is very limited for automotive structural design to reduce weight through structural optimization, and weight reduction effect is significant using composite material body [2]. Thermoplastic composite materials not only have the advantages of light weight and high strength, composite materials can be integrated, which greatly reduces the number of parts and fasteners. Thermoplastic composite materials are used in automobile bodies to improve collision energy absorption, vibration damping and fatigue resistance [3]. Future vehicles are
green cars that adapt to environmental protection. Therefore, it is inevitable to mention the
environmental protection awareness of composite materials. Thermoplastic composite materials have
characteristics, low emissions during the production process, and in addition to the advantages of
general new composite materials. It has the advantage of being recyclable, so it is in line with the
future development of environmentally friendly cars [4]. Due to the pressure of environmental
protection, motor vehicles in almost all cities, including counties, are restricted to driving only in some
time. At present, the distribution of goods in cities is mainly realized by electric commercial vehicles.
The cruising range of electric vehicles is directly related to the quality of the whole vehicle. Therefore,
the meaning is greater.

In order to reduce the weight and maintain the performance of electric commercial vehicles, the
static simulation analysis is first carried out in this paper on the suspension system bracket using steel
material under three working conditions. Taking the suspension system bracket connection plate as an
example, and the system support connecting plate is modelled using the thermoplastic composite
material for suspension, and the processes are introduced in detail on the pre-processing, solving, and
post-processing of the three-dimensional finite element simulation analysis, and results are obtained
on the stress distribution of different layers.

2. Model establishment and working conditions

2.1. Building a model
In the field of finite element, Hypermesh is a convenient meshing software, and Ansys Workbench is a
multi-physics coupling analysis software with excellent performance. Its powerful calculation
accuracy and multi-physics coupling are favoured by most automakers. The combination of the strong
and the strong is currently the mainstream analysis method for most CAE analysis. Import the
suspension bracket digital model into Hypermesh for mesh division. The mesh is required to use a
hexahedral mesh, try to avoid mesh distortion, and the part of the mesh that involves a large curvature
change is appropriately refined, and the divided mesh is imported. The three-dimensional finite
element model shown in the figure 1 in Ansys Workbench has an average mesh quality of 0.94 and a
full score of 1, which meets the requirements for meshing quality.

![Figure 1. Three-dimensional finite element model after meshing.](image)

3. Static analysis of steel materials

3.1. Building a model
The static analysis of the mounting system bracket simulates and calculates three working conditions
respectively. The first working condition simulates the force when 3 times the acceleration of gravity,
the negative direction of the Z axis in the vehicle coordinate system, and the second working condition
simulates the force when the car is turning. That is, acceleration of gravity adding steering inertia, the
third working condition simulates the force situation when the car is braking, that is, acceleration of
gravity adding braking inertia. The first working condition, a uniform load of 350×3 =1029N
perpendicular to the body downward is applied at A and B respectively to simulate the weight of the
motor under 3 times the gravitational acceleration, and 3 times the gravitational acceleration is applied. That is, 9806.6mm/s²×3=29420mm/s², fixed with 8 bolt holes.

The second working condition is that applying 350N uniformly distributed loads perpendicular to the vehicle body downwards at A and B to simulate the weight of the motor, respectively apply a gravity acceleration of 9806.6 mm/s² and 9806.6 in the Y direction of the vehicle coordinate system. The acceleration of mm/s² is used to simulate the steering inertia, and the fixed 8 bolt holes are similar to the point A in the figure 2. The third working condition is that applying 350N uniformly distributed loads perpendicular to the vehicle body downwards at A and B to simulate the weight of the motor, respectively apply a gravity acceleration of 9806.6 mm/s² and 9806.6 in the X direction of the vehicle coordinate system. The acceleration of mm/s² is used to simulate the braking inertia, and the fixed 8 bolt holes are similar to the point A.

4. Static analysis of steel materials

4.1. Static analysis on the first working condition
After finishing editing the boundary conditions, perform static analysis through Workbench. The stress cloud diagram under the first working condition is shown in the figure 2. The maximum stress is 189.22MPa, which is located at the mounting hole of the body connecting plate. MAX shows the position in the figure. The maximum stress of the position is less than the yield limit of the selected material 235MPa, which meets the design requirements.

![Figure 2. Static analysis cloud diagram under the first working condition](image)

4.2. Static analysis on the second working condition
The stress cloud diagram of the suspension bracket under the second working condition is shown in the figure 3, where the maximum stress is 92.483MPa, which is located at the mounting hole of the body connecting plate, and the maximum stress at the dangerous position is less than the yield limit of the selected material 235MPa, which meets the design requirements.

![Figure 3. Static analysis cloud diagram under the second working condition](image)

4.3. Static analysis on the third working condition

The stress cloud diagram of the suspension bracket under the third working condition is shown in the figure 4, where the maximum stress is 156.9MPa, which is located at the mounting hole of the body connecting plate, and the maximum stress at the dangerous position is less than the yield limit of the selected material 235MPa, which meets the design requirements.

![Figure 4. Static analysis cloud diagram under the third working condition](image)

Results show that the maximum stresses of the suspension system support steel in the three working conditions of triple gravity, gravitational acceleration adding steering inertia, gravitational acceleration adding braking inertia are 189.22, 92.48, and 156.9Mpa respectively, which are all less than the ultimate stress is 235Mpa, and the margin is relatively large. The currently selected materials meet the strength design requirements. In order to ensure driving safety, higher-grade bolts can be used for the installation of the motor lower beam, or to enhance the strength and precision of the body link plate. Methods such as the design of higher-strength composite materials.

5. Simulation analysis for composite material

Through static analysis, it can be concluded that under the three working conditions, the maximum stress of the suspension system is concentrated at the mounting hole of the body connecting plate. In order to improve the safety factor and ensure driving safety, the use of composite materials can meet the requirements of high strength and light weight. The purpose of quantification.

5.1. Analysis program

The 3D simulation analysis process of the thermoplastic composite material of the bracket connecting plate of the suspension system is the same as other materials, including the pre-processing part, solution, and post-processing to view the results. Due to the material processing properties, the pre-processing and post-processing processes are much more complicated. The pre-processing is roughly divided into 23 steps, the solution is simple, click the solve button to solve it directly, and post-processing is also more complicated, and it includes 8 steps.

5.2. Composite plies

The lay-up method of thermoplastic composite materials is basically processed in accordance with the processing process, and at the same time according to certain rules [5], it is divided into nine layers, the sixth layer is the matrix with a thickness of 2mm, and the rest are fibre reinforced layers with a thickness of 0.2mm, due to the properties of the material, the maximum tensile stress and maximum compressive stress of the matrix and the fibre reinforced layer are very different. Due to the different layup angles, the maximum tensile stress and maximum compressive stress of the fibre-reinforced layer with the same thickness are also different.
5.3. Boundary conditions
Through static analysis, the magnitude of the force on the connecting plate in all directions can be obtained. Import the connecting plate model into Ansys ACP, as shown in Figure 9, apply a force of -133.3N in the X direction, 399.06N in the Y direction, and -1218.4N in the Z direction to constrain all the degrees of freedom of the four mounting holes.

5.4. Result analysis
The stress cloud diagram of the sixth layer is shown in the figure 5. This layer is a matrix material, where the maximum tensile stress is 0.34Mpa less than the maximum allowable tensile stress of 1.6MPa, and the maximum compressive stress is 0.64Mpa less than the maximum allowable compressive stress of 1.1MPa. Meet the design requirements.

![Stress cloud map of the sixth layer](image)

The first layer of stress cloud is shown in the figure 6. This layer is made of UD-T700 laminated at -45°, and the direction of the layer is shown by the arrow in the figure. The maximum tensile stress is 993.19Mpa which is less than the allowable tensile. The maximum stress is 1500MPa, and the maximum compressive stress is 363.69Mpa, which is less than the maximum allowable compressive stress of 700MPa, which meets the design requirements. The second layer is made of UD-T700 layered at 45°. The maximum tensile stress is 892.81 MPa, which is less than the maximum allowable tensile stress of 1500 MPa, and the maximum compressive stress is 327.99 MPa, which is less than the maximum allowable compressive stress of 700 MPa, which meets the design requirements. The third layer is made of UD-T700 layered at 90°. The maximum tensile stress is 405.12MPa, which is less than the maximum allowable tensile stress of 1500MPa, and the maximum compressive stress is 255.66MPa, which is less than the maximum allowable compressive stress of 700MPa, which meets the design requirements. The fourth layer is made of UD-T700 layered at -45°. The direction of the layering is as shown by the arrow in the figure. The maximum tensile stress is 448.53MPa, which is less than the allowable maximum tensile stress of 1500MPa, and the maximum compressive stress is 231.73 MPa. The maximum compressive stress is less than 700 MPa, which meets the design requirements. The other four lays all meet the design requirements too.
Figure 6. Stress cloud of the first layer

The analysis conclusions are as follows:

1. The maximum stress of each layer of the composite material meets the design requirements.
2. Under the action of working condition 1, the paving angle of 90 degrees (the third, seventh, eighth, and ninth floors) has better stress and lower stress.
   - (3) By comparing the static analysis cloud images of the seventh, eighth, and ninth floors, under the same ply angle, the lower the bottom level bears less and less stress.
   - (4) The stress borne by the base layer is very small, and the main stress is borne by the reinforcing material.

6. Conclusions

1. The maximum stress of the suspension system bracket steel under three working conditions of triple gravity, gravitational acceleration adding steering inertia, and gravitational acceleration adding braking inertia are 189.22, 92.48, and 156.9Mpa respectively, which are all less than the ultimate stress of 235Mpa.
2. A nine-layer three-dimensional model of the thermoplastic composite material suspension connecting plate is established, and the pre-processing, solving and post-processing processes of the thermoplastic composite material simulation analysis are introduced.
3. The maximum stress of each layer of the thermoplastic composite material meets the design requirements. In the case of the same ply angle, the lower the layer is subjected to less and less stress.
4. The stress borne by the thermoplastic composite matrix layer is very small, and the main stress is borne by the reinforcing material.

Acknowledgments

This research was financially supported by National Natural Science Foundation of China (U1604141) and China Scholarship Council (CSC). The authors would like to acknowledge suggestion from the colleges in the past laboratory and several electric car companies.

References

[1] Xiong F., Wang D.F., et al. Lightweight optimization of the side structure of automobile body using combined grey relational and principal component analysis. Struct Multidisc Optim. 2018, 57: 441-461.
[2] Tetsuya Yamamoto, Katsumasa Uematsu, Toshihira Irisawa, Yasuhiro Tanabe. A polymer colloidal technique for enhancing bending properties of carbon fiber-reinforced thermoplastics using nylon modifier[J]. Composites Parts, 2018, (112):250-254.
[3] Mayer D, Bein T, Buff H, et al. Enhanced lightweight design by composites-results of the EU project ENLIGHT[J]. Journal of Reinforced Plastics and Composites. 2018, 37(9):1217-1224.
[4] Kang Q., Gu P.Y., Zuo S.G.. A research on design principles for electric drive assembly mounts of pure electric vehicles. Automotive Engineering. 2019, 41(11):1236-1242.

[5] Zhao H.J., Zhu W.C., Li H.L., etal. (2018) Determination and performance analysis on miniature and electric commercial vehicle frames of carbon fiber materials in the laying mode. 4th International Conference on Energy Equipment Science and Engineering. Xi’an. pp. 1-6.