Force Analysis and Optimization of BSC Brake Pedal Based on ANSYS

Shuheng Wang¹, Xueyuan Liu¹*, Wengang Chen¹, Jie Tang¹, Qiping Liu¹, Taiyu Ning¹, Haibing Yao¹

¹College of Mechanical engineering and Transportation, Southwest Forestry University, Kunming, Yunnan, 650224, China
*Corresponding author’s e-mail: xl@swfu.edu.cn

Abstract. In the BSC rules of the Chinese Society of Automotive Engineering, it is stipulated that the brake pedal should be designed to withstand a maximum brake pedal force of 450 LBF (2000 N). In order to ensure that the pedal design conforms to the rules of the event, it is necessary to use ANSYS finite element analysis software to conduct force analysis on the pedal design to ensure that it meets both the rules of the event and the safety design as well as the lightweight design. Therefore, through the finite element analysis of the pedal model in ANSYS environment, the structural stress and deformation data under the pedal force were obtained, and according to the stress and deformation cloud map, the pedal structure was optimized and the optimal material was selected, so as to obtain the brake pedal with high comprehensive performance and meet the competition rules.

1. Introduction
According to the regulations of the Chinese Society of Automotive Engineering baja, the car must be equipped with a hydraulic brake system that acts on all wheels and is operated by a single pedal. The pedal is rigidly connected to directly push the brake master cylinder; The braking system must be able to lock and lock all four wheels at static and road and unpaved speeds; The brake pedal shall be designed to withstand a maximum brake pedal force of 450 LBF (2000 N).[1] At the same time, as the finite element modeling and analysis method of continuous development and mature, finite element analysis technology has been widely applied in aviation, machinery, automobile, shipbuilding, construction and other engineering fields, finite element analysis can be used to calculate the shape and loading mode of complex structural static and dynamic characteristics, such as stress and strain, safety coefficient, total stress deformation, etc.[2] Therefore, this paper uses the finite element analysis software ANSYS to carry out statics and dynamics simulation analysis on the pedal, study the stress, strain and deformation of the pedal when it is subjected to pedal force, carry out reasonable design of the brake pedal, and analyze its influencing factors, and provide reference for further optimization design of the pedal. Finally, the practice of pedal design was tested in combination with Baja racing car of Lingyun team of Southwest Forestry University.

2. ANSYS brake pedal analysis target
Load a certain pedal force to the pedal through the ANSYS software analysis module, and obtain the analysis report of the pedal structure unit. By interpreting the analysis results, the distribution of stress and deformation in each area when the pedal is stressed is obtained, and the pedal structure is optimized based on theoretical data. And through the material stress analysis module in ANSYS,
different materials and combinations of materials are selected for the pedals, and the strength, hardness, and rigidity of the materials are considered to select the lightest weight and meet the braking requirements of racing cars.

2.1. Analysis of Factors Affecting Braking Feeling
Through consulting the data affecting the braking sensation, it can be seen that in the process of racing braking, the braking sensation is mainly composed of dynamic characteristics between the pedal force applied by the racer, the pedal stroke and the braking deceleration feedback from the racer.[3] Therefore, in the design of pedal structure, the above factors can be taken into account to gradually increase the comfort of the control movement of the racer, so as to achieve better results.

2.2. Brake pedal structure design and model establishment
Due to the compactness of the car head and the necessary placement of the front differential in four-wheel drive, the placement of the braking system has certain limitations. Therefore, the rational layout design of the braking system in the front cabin is very important, and the braking system needs to meet the test of the endurance race for up to four hours. Therefore, in order to ensure that the driver is comfortable and dynamic in the endurance race, the design process of the pedal needs to fully consider the dynamic comfort of the pedal, the lever ratio of the pedal and other factors.

According to the parameters of the car, the rules of the race and the simple and lightweight design principles of the braking system, the Bach braking system does not have an air booster, but uses the principle of the lever of physics. It is assumed that the pedal force at the surface of the pedal is the power, and the force pushing the oil pump process is the resistance, and the distance proportion between the power arm and the resistance arm is reasonable to meet the braking force required by the racing car. Taking the U6 racing car of Lingyun Team of Southwest Forestry University as an example, the distance ratio between the power arm and the resistance arm was set as 5:1, and the pedal force was increased by five times. According to the braking parameters of the racing car and the space and layout requirements of the front cabin of the racing car, the brake pedal structure as shown in the left figure below was designed. And according to the stress analysis of the brake pedal by ANSYS, the optimized structure of the brake pedal as shown in the figure on the right below is planned.

2.3. Meshing of finite element model
In ANSYS, reasonable mesh division determines the accuracy and operation speed of the solution results to a certain extent, so it is necessary to accurately select the mesh type and mesh size of different structures, and carry out local mesh encryption at the stress concentration to improve the quality of mesh division. Therefore, firstly, the brake pedal is divided into three parts: pedal surface, pedal arm and fixed column by using the Boolean operation function in ANSYS. Then, the mapping mesh method was used to generate regular hexahedral mesh for the tread surface and pedal arm, and the sweeping mesh method was used for the fixed column. The quadrilateral mesh was encrypted before partition to ensure that the hexahedral mesh was generated after sweeping. At the same time, local mesh division is carried out at the stress concentration, and the mesh quality is improved by increasing the number of mesh and encrypting the mesh size, and the mixed mesh division method is used to improve the mesh quality and obtain a higher solution accuracy.[4]

In the process of numerical simulation, the stress deviation (SDSG) of brake pedal is estimated by finite element error estimation module in ANSYS. The local mesh size was encrypted by the stress deviation cloud map, and the mesh independence was verified by increasing the number of nodes and the mesh size. Through the continuous debugging of grid size, the error range is controlled within 3%, and higher calculation accuracy is obtained.[5]
2.4. ANSYS import and simulation construction of brake pedal model

UG 3D drawing software is used to build the ideal model of brake pedal. With the advantage of ANSYS finite element analysis of material mechanics simulation, UG model is converted into a STP file, and then imported into ANSYS Workbench. Import the modeled braking system, and modify the constraint relationship between components, so that it can carry out the simulation experiment normally, and get the correct simulation results. A braking force perpendicular to the pedal surface is established in the static structural. In a simple experiment, it is known that for an adult of 60kg, the leg force applied in the sitting position is 200N. Therefore, in the case of emergency braking, the simulation braking force value is increased to 250N to ensure the stability of pedal structure in case of emergency. Therefore, 250N braking force perpendicular to the pedal face down is applied on the tread surface, and a force expanded five times by the pedal lever principle is applied on the piston connected to the brake master cylinder. At the same time, the solution conditions of the total force and total deformation of the pedal are inserted into the solution plate and solved by solving. The following data reports are obtained when the structural material is structural steel.

3. Pedal simulation optimization

3.1. Simulation results analysis and quasi-optimization

| Object Name | Total Deformation | Equivalent Stress |
|-------------|-------------------|-------------------|
| Minimum     | 0. mm             | 0. MPa            |
| Maximum     | 1.1333 mm         | 85.368 MPa        |
| Average     | 0.26135 mm        | 4.2334 MPa        |

It can be seen from the stress cloud diagram of the simulation results in figure 2 that: the stress on the symmetry axis of pedal arm is the minimum, and the stress on both sides of the symmetry axis gradually increases, and there is obvious stress concentration at the connection between pedal arm and the fixed column and the hanging lug of the fixed column. The stress at the connection reaches a maximum of 85Mpa, and the overall stress of the pedal surface component is relatively small.
According to the physical properties of the materials in the table below, the safety factor of the pedal made of 45 structural steel is solved according to Formula (1):

\[ n = \frac{\sigma_b}{\sigma} \quad (1) \]

Type: \( \sigma_b \) - Ultimate strength of material at fracture, Mpa; \( n \) - Safety factors of material components; \( \sigma \) - Material stress, Mpa.

Table 2. 45 Physical Properties of Steel and 7075 Aeronautical Aluminum

| Material     | Density/(g/cm³) | Modulus of elasticity/GPa | Poisson ratio | Tensile strength/(MPa) | Yield strength /(MPa) | Elongation% | Quality/(kg) |
|--------------|-----------------|---------------------------|---------------|------------------------|-----------------------|-------------|--------------|
| 45 Steel     | 7.85            | 210                       | 0.31          | 600                    | 355                   | 16          | 0.95         |
| 7075 Aluminum| 2.82            | 71                        | 0.33          | 570                    | 505                   | 11          | 0.26         |

Therefore, in order to reduce weight, the use of structural steel can be gradually reduced under the premise that the brake pedal is made or processed from steel or aluminum, and the pedal material can be replaced with a mixture of steel and aluminum or high-performance aluminum material. However, there are many problems in the welding between steel and aluminum. First of all, there are great differences in chemical and physical properties between steel and aluminum. For example, the melting point of steel is about 1500°C and the melting point of aluminum is about 660°C, the difference between the two is about 900°C; [7] Second in the welding process, the aluminum alloy directly exposed to the air will be particularly prone to refractory 3 oxidation 2 aluminium oxide film, its melting point 2550°C directly affect the welding quality, and aluminum in steel is easy to form very crispy IMP, and along with the increase in heat input, generate IMP phase is, the more serious damage to the brittle phase coupling of the strength of the static and dynamic. [8] Therefore, when welding aluminum metal, there are strict requirements on the welding working environment and welding equipment. Therefore, under the comprehensive consideration of economy and practicality, lightweight aluminum material can be used on the central axis of pedal symmetry of Baja racing car, or weight reduction holes can be made on the central axis. Pedal surface material can be completely replaced by lighter material, through mortise and tenon structure pedal surface and pedal arm assembly as a whole, and the use of rubber cover on the surface of the pedal surface to increase friction, not only increase the driver system dynamic, but also make the vehicle design further in line with the concept of lightweight.

3.2. Structural optimization of 45 steel

Under the loading force condition of 250N tread surface, although the pedal can meet the normal braking according to the practical test results, and the analysis results can also meet the requirements of strength and hardness, and meet the range of safety factor. However, the pedal mass is relatively heavy, so it is planned to optimize and appropriately reduce the thickness of the pedal. According to the rules of Bach competition system of China Automotive Engineering Society, the pedal should bear a force of 2000N. Therefore, the optimized 45 steel pedal is analyzed in ANSYS for deformation and stress strain, and the following data results are obtained:

Fig 3. Optimized cloud diagram of pedal deformation under 45 steel
Table 3. Stress-strain and deformation data table of 45 steel after optimization

| Object Name | Total Deformation | Equivalent Stress | Equivalent Elastic Strain |
|-------------|-------------------|-------------------|---------------------------|
| **Minimum** | 0. mm             | 2.2423e-004 MPa   | 1.19e-009 mm/mm           |
| **Maximum** | 13.729 mm         | 2014.5 MPa        | 1.0792-002 mm/mm          |
| **Average** | 4.0907 mm         | 73.496 MPa        | 3.8292e-004 mm/mm         |

After the optimization of reducing the thickness of pedal arm and pedal surface, the pedal weight decreased from 0.9kg to 0.481kg when the pedal was subjected to the pedal force required by the competition system. However, according to the stress and deformation cloud of pedal, the pedal stress distribution remains unchanged, while the maximum stress of pedal element stress increases from 85.36mpa to 1786.4mpa, which is greater than the ultimate stress of material fracture. The safety factor of the pedal becomes 0.21, which is not within the safety factor range of 1.5-2.0 of plastic materials. At this time, the pedal parts are no longer safety parts. Moreover, the total deformation of the pedal increases from 1.13mm to 13.695mm. The pedal deformation is too large, and the pedal may cause more or less irreversible deformation in the case of frequent emergency braking.[9]

Therefore, it can be concluded that when the pedal material is 45 steel and the structure is not optimized, the pedal quality can meet the requirements of the competition, with a high safety factor, but the weight is biased. After optimizing the structure, the stress is too concentrated and the rigidity and safety can not meet the requirements, which will lead to safety problems of pedals and do not conform to the competition rules.

3.3. Pedal optimization design of 7075 aviation aluminum

It was known that the pedal of 45 steel structure did not meet the production requirements of the competition, and aluminum material with high hardness, strength and low density was selected to obtain the brake pedal of high strength and light weight, so the material was replaced with 7075 aluminum. According to the pedal stress distribution diagram of 45 steel, more precise optimization was carried out on the structure. Under the condition of low stress strength at the pedal central axis, weight-reducing holes were designed on the pedal central axis to reduce the pedal quality, as shown in figure 4 below.[10] According to the theoretical knowledge of material mechanics, chamfering design is carried out at the joint to reduce the concentrated stress and make the pedal stress distribution more reasonable. In addition, considering the economy, reasonable chamfering design is carried out for the part shown in figure 5 to complete the optimization design of pedal structure and material selection.

![Fig4. Pedal weight reduction hole design](image1)

![Fig5. Chart chamfering design](image2)

After the structure is optimized and the pedal material is replaced, the pedal force that meets the competition rules is applied to the pedal, and the following finite element analysis results are obtained:
According to the above analysis results, after the structural optimization and the replacement of the pedal material with 7075 aluminum, the mass of the pedal decreases to 0.26kg, 0.64kg lighter than the non-optimized brake pedal, which is an ideal brake pedal. At the same time, the maximum stress borne by the brake pedal is 364.14Mpa, which is still less than the ultimate stress when the material breaks. According to Formula (1), the safety factor of 7075 aluminum pedal is 1.57, which is within the safety factor range of 1.5-2.0 for plastic materials and meets the braking requirements.\[11\]The stress cloud figure 7 shows that the stress is mainly concentrated at the parts connected to the balance bar, the force on the pedal arm is moderate, and the maximum stress of the pedal is still less than the tensile strength of 7075 aluminum 570MPa. The maximum pedal shape variable is 8.58mm, and the average type variable is 2.64mm. It can be seen from the pedal deformation cloud figure 6 that the largest part of the shape variable is on the pedal surface, but the pedal surface is only subjected to the force of the racer's feet, which is far less than 400N, which is not enough to make the pedal surface deformation to a large extent. According to the above analysis results, the pedal structure stress and deformation table in Table 5 is obtained, and the analysis data of three kinds of pedals are directly obtained. Through data comparison, the pedal with the best structure is obtained.

**Table 4. Stress and deformation data table of 7075 aero aluminum pedal after optimization**

| Object Name                  | Total Deformation | Equivalent Stress | Equivalent Elastic Strain |
|------------------------------|-------------------|-------------------|---------------------------|
| Minimum                      | 0. mm             | 2.2941e-004 MPa   | 5.4326e-009 mm/mm         |
| Maximum                      | 8.5831 mm         | 361.14 MPa        | 5.0585-002 mm/mm          |
| Average                      | 2.6401 mm         | 28.204 MPa        | 4.0247e-004 mm/mm         |

**Table 5. Optimization results**

| material                                    | maximum stress/MPa | mean stress/MPa | Maximum shape change/mm | Mean shape change/mm |
|---------------------------------------------|--------------------|-----------------|-------------------------|---------------------|
| Unoptimized 45 steel structure brake pedal  | 85.36              | 4.23            | 1.13                    | 0.26                |
| Structure-optimized 45-steel brake pedal     | 1786.40            | 76.36           | 13.70                   | 3.93                |
| Optimized 7075 aluminum moving pedal        | 364.14             | 28.20           | 8.58                    | 2.64                |
Therefore, the conclusions can be drawn as follows: According to the analysis results, the design of weight reduction hole in the position of pedal stress is low, such as the central axis, and the design of chamfering and fillet in the place of stress concentration, such as the connection between pedal arm and fixed column, is carried out by the connection technology. The pedal material is replaced with high-performance aluminum alloy from 45 structural steel, and the pedal material is 7075 aluminum by comparison. And 7075 aluminum quality brake pedal load in the ANSYS analysis software to meet format rules pedal force, the factor of safety within the scope of the security, make the design of the brake pedal should both comply with the lightweight design of the car brake pedal, to fulfill the requirements of the format rules again at the same time, guarantee the brake pedal in the tournament game can work normally.

4. Conclusion
According to the rules of Baha Competition of China Society of Automotive Engineering and based on the design concept of lightweight and safety, 3D modeling of brake pedal was carried out by NX10.0. In addition, finite element analysis was used to analyze the loading pedal force of 2000N for brake pedals with different materials and structures by ANSYS software, and the following conclusions were obtained:

Under the condition of 45 steel brake pedal which is not optimized, ANSYS finite element analysis is carried out on it, and the maximum stress is 85.36MPa, the maximum deformation is 1.13mm, and the safety factor is 4.12. Although the working performance can meet the working pressure of the brake pedal according to the competition rules, the weight of a single brake pedal is as high as 0.95kg, which is a relatively large brake part in the Baja racing braking system and does not conform to the lightweight design of Baja racing.

According to the stress cloud map and deformation cloud map in ANSYS analysis report, the structure optimization design of stress concentration is carried out. The design method of fillet and chamfering is adopted to avoid stress concentration. The weight reduction hole is designed in the area with low stress, such as the spool in the pedal arm, to reduce the pedal weight; Select different pedal materials according to the rules, through the performance comparison between different materials, at the same time with the help of ANSYS finite element analysis, obtain the pedal performance of different materials, to select the appropriate pedal material.

Through the same parametric analysis of pedals of different materials, the working performance of 7075 aluminum is not lower than that of 45 steel. Moreover, the weight of the brake pedal decreased to 0.26kg, 0.64kg lighter than that of the non-optimized Baja brake pedal, in line with the lightweight design. Therefore, the data show that the 7075 aluminum material pedal can not only meet the work requirements of the brake pedal, greatly reduce the weight of the brake system, but also gradually improve the theoretical analysis of the Baja racing brake system, to ensure the safety of personnel in the process of the race.

Acknowledgments
Thank you to my family, teachers and classmates who have been silently supporting me.

Project funds: Yunnan University Students Innovation training project, Fund number: 202110677010.
Project funds: National innovation training program, Fund number: 202010677104.
Project funds: National innovation training program, Fund number: 202010677162.

References
[1] China Society of Automotive Engineering Baja Competition Rules (2020) [M]. Beijing.Automobile Society of China. 2020.
[2] YAN Yun. Structural Optimization Design Based on ANSYS Parametric Design Language [J].Journal of East China Jiaotong University,2004(04):52-55.
[3] Xiong Ziyuan, Wei Yiyiing, Dai Zhanwei, Bai Fan, Chen Zhijun. Automotive Brake Pedal Feeling Simulation and Optimization Based on AMESim [J]. Automobile Technology, 2020(03):50-57.

[4] YUAN Guoyong. Analysis of ANSYS Mesh Partition Method [J]. Modern Machinery, 2009(06):59-60.

[5] Zhang Hanwen, Si Yonghong. Finite Element Analysis Software ANSYS Calculation Speed Research [J]. Computer Knowledge and Technology, 2021, 17(05):246-247.

[6] YAN Wei, SHAO Yiming, ZHU Xiayi. Simulation Test Research on Braking Performance of Truck Based on Virtual Prototyping Technology [J]. Journal of Shandong Jiaotong University, 2013, 21(01): 1-5.

[7] YANG Renhua. Simulation Analysis of Brake Caliper Based on ANSYS [J]. Science Technology and Innovation, 2018(19): 1.

[8] Li Yajiang, Wu Na. Research Status of Steel/Aluminum Dissimilar Metal Welding [J]. Welding, 2010(03): 5-12.

[9] Zhang Rongyun, Song Rencai, Hong Zhuangzhuang, et al. Finite Element Analysis of Brake Connector and Brake of Baja Racing Car [J]. Journal of Chaohu University, 2018, 20(06): 75-83.

[10] Zhang Hongge, Zhang Songhong. Structural Optimization Analysis of Vehicle Bucket Lifting Mechanism Based on ANSYS [J]. Machinery Design & Manufacture, 2021(02): 204-208.

[11] XU Dan. Finite Element Analysis of Disc Brake Structure Based on ANSYS[J]. Forestry Machinery and Woodworking Equipment, 2020, 48(09): 35-39 + 44.

[12] Liu Kelong, Yao Weixing, Yu Xiongqing, et al. Journal of Nanjing University of Aeronautics and Astronautics, 2004(01): 22-28. (in Chinese)