Parametric Modeling and Finite Element Analysis of the Brake Drum Based on ANSYS APDL

Bin ZHENG\textsuperscript{1,2}, Jing-dong ZHANG\textsuperscript{1,2} and Tang Li\textsuperscript{1,2}

\textsuperscript{1}School of Transportation and Automobile Engineering, Panzhihua University, Panzhihua 617000, Sichuan, China
\textsuperscript{2}Institute of Automobile Engineering Technology of Panzhihua, Panzhihua 617000, Sichuan, China

Keywords: Brake drum, Parametric modeling, Finite element analysis, APDL.

Abstract. The brake drum is accompanied with deformation while operation process, so it is particularly important to study its static characteristics. The static analysis of the brake drum was implemented and the static deformation and equivalent stress of drum were computed based on ANSYS APDL. The maximum deformation and equivalent stress distribution were obtained. In addition, the design hidden troubles were pointed out. These results can offer references for the brake drum design and manufacture as well as improved the quality of the brake system.

Introduction

The braking system is a key part of automobile and the braking performance directly affects the safety of automobile driving. More and more researchers pay attention to the automobile brake system with the increasing of the automobiles. The static analysis of the brake drum has significance to improve the braking efficiency and stability of the automobile braking system.

Xi Chuanpeng studied a kind of wheel brake of a heavy vehicle by virtual prototype technology. The virtual prototype model of wheel brake is simulated and analyzed with the ADAMS software, obtained total braking torque are consistent with the results from theory calculation. Then, the surface stress of each friction plate is analyzed by using the finite element software ANSYS\textsuperscript{[2]}. Ma Xun set up a three dimensional geometric and finite element model of a drum brake by means of ANSYS software. The simulation results are corresponding to the experiment’s, the results of contact pressure distribution between the brake drum and brake shoes are obtained as well as the equivalent stresses of the brake\textsuperscript{[3]}. Furthermore, many domestic experts and scholars conducted related research on brake drum\textsuperscript{[4-9]}.

From the above literatures, it concluded that the brake drum are designed, optimized and finite element analysis, but few parametric modeling and static analysis of the drum are researched. Because the static analysis of the brake drum has an important role, there is a need to establish parametric model by using APDL and carried out the simulation.

The static deformation and equivalent stress of the drum were carried out by using ANSYS APDL static analysis which provided a theoretical basis for drum deformation research. Therefore, this paper mainly focused upon the structure deformation and equivalent of the drum and discussed the static analysis results.

Parametric Modeling of the Drum Using APDL

The braking system is very important to a car. If the brake system don not operate properly, the result can be disastrous. Brakes are actually energy conversion devices, which convert the kinetic energy of the car into thermal energy. There are two main types of brakes: drum brake and disc brake.

The typical drum brake consists of five main parts, that is the brake, the two brake shoes (leading shoe and trailing shoe) and two friction linings. The brake process is that the leading shoe and trailing shoe can rotate with its axis of the eccentric shaft axis so that the friction lining contact with the inner of the brake drum and cause the wheel to stop.
The physical picture of the drum is shown as Figure 1. The specifications of the drum are described below.

- Mass of the automobile with laden is about 7500 kg.
- Mass of the automobile without laden is about 3500 kg.
- Internal diameter of the brake drum is 394 mm.
- External diameter of the brake drum is 440 mm.

In the following, the corresponding APDL command flow is given for the solid modeling of drum brake drum brake, and the specific command flow is as follows:

```plaintext
num=10       /prep7
k,,0,120,0    k,,0,175,0
k,,50,210,0   k,,140,210,0
k,,140,220,0  k,,240,220,0
k,,240,205,0  k,,50,205,0
k,,15,175,0   k,,15,120,0
k,,0,0,0      k,,10,0,0
asel,none     a,1,2,3,4,5,6,7,8,9,10
cm,a-mid,area vsel,none
vrotat,a-mid,,,,,,11,12,360 cm,v-mid,volu
vsel,none     wprota,,,-90 ...
```

The above ANSYS APDL command stream can be edited or modified by the text editing software, or it can be executed directly in ANSYS software. The figure 2 gives the drum model that established by ANSYS APDL.
Static Analysis of the Drum

While carrying out the mesh division of drum, the 3D solid element of 187 tetrahedron unit is used for creating finite element model, which includes 10 nodes. By using the free mesh division method to mesh the drum, the finite element model of the drum is created as shown in figure 3.

The function of static analysis is to solve the displacement and stress of the structure under the action of static load. The static analysis calculates the effect of a structure under a fixed load without considering the effects of inertia and damping. While implement static analysis of the drum, the boundary conditions is that ten screwed holes contacted to the base are fixed to prevent movement.

Displacement Analysis of the Drum

The Overall Displacement Distribution of Drum. From the figure 4, it can be seen that the overall displacement is 0.078664mm. It shows that the deformation of the drum is relatively large. Therefore, the following optimization of the drum should be taken into account the edge deformation of the drum.

Displacement Distribution Along X Axis of Drum. From the figure 5, it can be seen that the maximum displacement of the drum is 0.012119mm, which indicate that the distribution along X axis is small.

Displacement Distribution Along Y Axis of Drum. From the figure 6, it can be seen that the maximum displacement is 0.77946mm, the displacement distribution along Y axis is relatively large. During the braking process, the friction between the brake shoe and the deformation of drum is large. In the design process of the brake drum, it should consider the reasonable structure to avoid large deformation and to ensure the service life of the brake and braking performance.

Displacement Distribution Along Z Axis of Drum. From the figure 7, it can be seen that the maximum displacement is 0.77983mm, the displacement distribution along Z axis is similar to Y axis. During the braking process, the friction between the brake shoe and the deformation of drum is large. From the results of static analysis, it can be found that the contact effect easy to deform, that is, it also to cause its deformation.
The Stress Analysis of the Drum

Overall Equivalent Stress Distribution of the Drum. Based upon the results of the stress analysis, in the process of braking, the friction lining and drum near the position of the first contact is the maximum stress in the place where the maximum stress is 49.6MPa, which occurs relatively at the upper side of the drum. The stress distribution is equally distributed on the surface of the drum as the drum is symmetrical structure.

Equivalent Stress Distribution Along X Axis of the Drum. The equivalent stress distribution along X axis of the drum is shown as the figure 9. It can be seen that the maximum stress along X axis is 20MPa, which occurs relatively centrally at the upper bottom of the drum. The stress distribution is equally distributed on the surface of the drum as the drum is symmetrical structure.

Equivalent Stress Distribution Along Y Axis of the Drum. The equivalent stress distribution along Y axis of the drum is shown as the figure 10. It can be seen that the maximum stress along Y axis is 3.74MPa, which occurs relatively centrally at the bottom of the brake drum. Because the brake drum is symmetrical structure, its stress distribution is equally distributed on the surface of the drum.

Equivalent Stress Distribution Along Z Axis of the Drum. The equivalent stress distribution along Z axis is shown as the figure 11. It can be seen that the maximum equivalent stress along Z axis is also 3.74MPa, which occurs relatively centrally at the bottom of the brake drum from figure 11. Because the brake drum is symmetrical structure, it can be seen that there is no big difference along Z axis. The equivalent stress distribution along Z axis is very similar to the equivalent stress distribution along Y axis.
Conclusions

In this paper, the parametric modeling by using ANSYS APDL and finite element static analysis of the brake drum are carried out, the deformation and equivalent stress are obtained and the following conclusions are concluded. The parametric model of drum is completed by using ANSYS APDL parametric design. While the dimensions of the drum changed, it only need to modify the relevant parameters of the APDL to establish the new drum model. Then, the static analysis of the drum is carried out also by using ANSYS APDL and the deformation and equivalent stress are obtained. The results of finite element analysis can provide reference for the design of brake drum and select of reasonable parameter, make the design of the structure of drum more reasonable. It is significant to improve the work efficiency and shorten the design cycle by using ANSYS APDL.

References

[1] Xi Chuan-peng, Wang Kai, Li Zhong-xun. Simulation analysis of a new type wheel brake of heavy duty truck[J]. Journal of Mechanical Transmission, 2017, 41(2):171-176.

[2] Zhang Jingdong, Zheng Bin, Yao Jie, et al. Modal analysis and optimization design on brake drum for a heavy-duty automobile[J]. Bus & Coach Technology and Research, 2016, (2): 1-3.

[3] Ma Xun, Cheng Ming-dong, Zhao Xu. Research of finite element analysis of drum brakes[J]. Machinery Design & Manufacture, 2012, 6(6):217-219.

[4] Ibrahim Ahmed. “Modeling of automobile drum brake for contact analysis using ANSYS”, SAE 2012-01-1810, 2012.

[5] Liu li. Study on the braking efficiency and optimal design of the drum brake in engineering automobile[J]. Journal of Mechanical Transmission, 2015, 32(5):101-104.

[6] Liu Kun, Peng Meichun, Lin Yiqing, et al. Finite element analysis of drum brake stamping-welding shoes based on ANSYS Workbench[J]. Journal of Guangdong University of Technology, 2013, 30(1):92-96.

[7] Li Hanwu, Dong Hongwei, Du Honglei, et al. Modal analysis of brake drum of automobile drum type brakes[J]. Forestry Machinery & Wood Working Equipment, 2011, 39(6):31-33.

[8] Ibrahim Ahmed, Shawky Aboul-Seoud. “On the analysis of drum brake squeal using finite element methods technique”, SAE 2006-01-3467, 2000.

[9] Ma Xun, Cheng Ming-dong, Zhao Xu. Research of finite element analysis of drum brakes[J]. Machinery Design & Manufacture[J], 2012, (6):217-219.