Structural Strength Analysis of Gearbox Casing Based on ABAQUS

Xiaohan Tang
School of mechanical and civil engineering and architecture, Northwestern Polytechnical University, Xi’an, China
17791814788@163.com

Abstract. In order to improve the transmission accuracy, static and dynamic characteristics of the transfer gearbox of a test bench, a numerical model is built by finite element software ABAQUS to analyze its static characteristics. The static analysis under typical working conditions are investigated to obtain the stress and the strain, and then the static reliability of the structure is checked, which provides theoretical basis and guidance for the structural optimization design of the gearbox. The research shows that the gearbox structure is reliable and has greater space for structure optimization, which can improve the transmission accuracy and static characteristics.

1. Introduction
As a common part in mechanical equipment, gearbox plays an important role in manufacturing [1]. There are three traditional types of many kinds of gearboxes, cylindrical gearboxes, planetary gearboxes and the combination of those two gearboxes. Gearbox is composed of front box, back box, driving gear, driven gear, bearing and some related accessories [2]. The front box and the back box are the devices which assemble each part into an integral unit to realize the high speed small torque input to low speed large torque output conversion [3]. Lightweight and long service life of the gearbox has become the prime requirements of the modern power transmission systems [4]. The gearbox is driven by the gear to change the speed, and it can produce collision and impact in the working process and then transmit to the box by the gear shaft or the bearing, which causing vibration or noise in the gearbox casing. Once the gearbox malfunctions, it will bring incalculable losses to enterprises and even countries. Therefore, it is vital to analyze the structural strength of the gearbox in the design stage.

3D design softwares commonly used are AutoCAD, Pro/Engineer, UG, SOLIDWORKS, RHINO, CATIA and CIMATRON, etc. And there are many commonly used finite element analysis softwares in engineering, such as ANSYS, which is dedicated to linear analysis and offers a complete set of tools for automatic mesh generation [5]; ABAQUS, which focuses on analyzing more complex and in-depth engineering problems; NASTRAN, which is special for structural analysis of the aerospace industry for NASA; and ADINA, which is mainly used to mechanical analysis. Yibin et al. [6] have used SOLIDWORKS to reducer box structure finite element analysis. Wen et al. [7] have established the dynamic finite element analysis models of the marine gearbox's housing and the gear-shaft-bearing-housing coupled system respectively by using ANSYS. Huang and Yan [8] have established One-
Armed Planet Carrier of Wind Turbine Gearbox three-dimensional model in UG and accomplished static stress analysis by the finite element method. Zhao et al. [9] have built finite element analysis models of the SG135 automobile gearbox based on correlation theory in the ANSYS software environment. Zhang et al. [10] have built a 3-D model of a certain heavy vehicle gearbox by SOLIDWORKS, then analyzed the model strain state by ANSYS and got a perfect structural design. Patil and Kumar [11] have used Solid Edge and Pro/E for designing gearbox assembly and used ANSYS 14.5 finite element analysis simulation for working on meshing of geometry.

ABAQUS has more types of unit types, contact and connection types, which can provide more options and a deeper reflection of microstructural phenomena and differences between phenomena. SOLIDWORKS has a more user-friendly operation interface and the numerical simulation with powerful function, has greater advantages in mechanical model design. Therefore, is used to establish the model of gearbox, which then simplified and imported into for analysis and calculation. This article is organized as follows: in Section 2 and 3 the preprocessing and establishment of finite element model are elaborated. Numerical and experimental results are compared in Section 4. Concluding remarks are presented in Section 5.

2. Finite element model processing
Slightly different from the solid model applied in practical engineering, the establishment of finite element analysis model needs to ignore some small structural features of the structure, such as bolt holes, rounded corners, chamfering, etc. In the process of finite element analysis, it is necessary to eliminate the small geometric features that have little influence on the analysis results, and only retain the main features and general framework that can accurately describe the structural characteristics of the model. It may cause problems such as mesh failure and solution error in finite element analysis if there are too many small structures in the finite element model. Therefore, the following model simplification must be performed on the analysis objects before importing the models in:

1) The model should not have defects such as structural dislocation and superposition when importing the model established by SOLIDWORKS into ABAQUS;
2) The chamfering, fillet and other features in the solid model that do not affect the calculation results are ignored, and the convex body that does not affect the analysis results is removed from the surface of the box;
3) The key parts in the non-frame of the model are removed, such as the bolts, screws and nuts in contact with the fixed surface;
4) The threaded parts are removed from the remaining screws and thread holes and simplify them into polished rods.

3. Establishment of finite element model of the gearbox
Import the gearbox 3D model built by SOLIDWORKS into ABAQUS.
3.1. Setting the material properties
Due to the excellent shock absorption, low notch sensitivity and high abrasion resistance, gray cast iron is used as the material of the gearbox. Elastic modulus of gray cast iron is $E = 140 \text{GPa}$, Poisson's ratio is $\nu = 0.25$, and density is $\rho = 6.6 \times 10^3 \text{kg/m}^3$. The material properties is added to the gearbox model.

3.2. Finite element mesh division
All manuscripts must be in English, also the table and figure texts, otherwise we cannot publish your paper. Please keep a second copy of your manuscript in your office. When receiving the paper, we assume that the corresponding authors grant us the copyright to use the paper for the book or journal in question. Should authors use tables or figures from other Publications, they must ask the corresponding publishers to grant them the right to publish this material in their paper.

3.3. Defining loading loads and boundary conditions
The gearbox body mainly plays the role of supporting and absorbing vibration in the working process, and the load it receives mainly comes from the radial force and axial force that the gear working load makes the bearing receive. Therefore, considering the load size and direction of each bearing installation position, the load on the gearbox can be applied on the joint surface of the gear bearing and the gearbox body. The front and back boxes are connected by bolts, so the contact relationship between the two boxes is set as binding contact. Since the gearbox body is connected to the base through the anchor bolt, and the anchor bolt makes the gearbox body contact with the base through the action of pre-tightening force, a fixed constraint is added on the bottom surface of the box seat. The gearbox body and the hub part are connected by binding constraints so that there is no displacement between the gearbox body and the hub. The axial load of spur gear is zero, so only the radial load acting on the hub of the gearbox body is considered. In the practical state, the load acting on the bearing can be regarded as uniformly distributed load on the gearbox body. The finite element model is shown in Fig. 2.

![Figure 2. Finite element models of the gearbox](image)

4. Analysis Results and Discussion
After establishing the finite element model, the calculation is submitted. Through the image analysis and data summarization of the static analysis results of gearbox, the static distribution of the initial design of the gearbox structure can be obtained, such as deformation distribution and stress distribution of the gearbox structure, and then the magnitude and direction of maximum deformation and equivalent stress can be obtained. The deformation cloud diagram of the gearbox in X, Y and Z directions and the total deformation cloud diagram are shown in Fig. 3.
The results show that the deformation of the gear box is concentrated at the mounting point of the output shaft bearing. The maximum deformation of the gearbox body in the X-direction is located on the left and right side of the output bearing installation, and the maximum deformation is 2.31 \( \mu \)m. The minimum displacement occurs at the bottom plate of the gearbox. The maximum deformation of the gearbox body in the Y-direction is located on the upper and lower sides of the output bearing installation, and the maximum deformation is 2.21 \( \mu \)m. The maximum deformation of the gearbox body in the Z-direction is located outside the output shaft, and the maximum deformation is 2.43 \( \mu \)m. The displacement on the bottom and left side of the gearbox is minimal. It was found that the maximum structure deformation is 3.07 \( \mu \)m on the contact surface between the output shaft bearing and the gearbox. The bottom plate of the gearbox and the top of the box have the minimum deformation.

To sum up, the influence weight of deformations in the X, Y and Z directions on the total deformation of the gearbox is almost the same, which verifies the rationality of the structural design of the gearbox.

According to Fig. 4, the maximum value of the first main stress of the gearbox is 9.4MPa and the maximum value of equivalent stress is 12.7MPa. The maximum stress values of both are in the contact position between the output shaft bearing and the gearbox body. The structural material of the gearbox body studied in this paper is gray cast iron, so the maximum equivalent stress is far less than the allowable stress of the material. Therefore, the strength of the gearbox body meets the requirements.

5. Conclusion
In this paper, the solid model of the gearbox is established by SOLIDWORKS software and the force under rated working conditions is analyzed, then the simplified geometric model is imported into
ABAQUS software for finite element statics analysis. It can be found that the maximum stress of the gearbox is much less than the yield strength of the material. Therefore, the rationality of the designed gearbox is verified. According to the analysis results, the gearbox can be analyzed and evaluated to find the structure to be improved and optimized.

References
[1] Leite G N P, Araújo M A, Rosas P A C. Prognostic techniques applied to maintenance of wind turbines: a concise and specific review. Renewable and Sustainable Energy Reviews, 2018: 1917-1925.
[2] Brar J S, Bansal R K. A Text Book of Theory of Machines. Firewall Media, 2004.
[3] Liang X H, Zuo M J, Feng Z P. Dynamic modeling of gearbox faults: A review. Mechanical Systems and Signal Processing. 2018, 98: 852–876.
[4] Maruti P, Ramkumar P, Shankar K. Multi-objective optimization of the two-stage helical gearbox with tribological constraints. Mechanism and Machine Theory, 2019, 138: 38-57.
[5] Malgar D, Udupa N G S, Venkatram N. Investigation of fatigue behaviour of traction gear box gears. Research & Technology in the Coming Decades. IET, Ujire, India, 27-28 Sept. 2013.
[6] He Y B, Yang B K, Zhang Y, et al. Lightweight Study on Gear Reducer Box Structure Based on Finite Element Analysis. Advanced Materials Research, 2013, 823: 20-23.
[7] Liu W, Lin T J, Peng Q C. Modal Analysis and Experimental Research of Marine Gearbox. Applied Mechanics and Materials, 2014, 607: 405-408.
[8] Huang Bo, Yan YuTao. The Structural Optimization of One-Armed Planet Carrier of Wind Turbine Gearbox Based on Lightweight Structure.3rd Annual International Conference on Mechanics and Mechanical Engineering (MME), Chengdu, PEOPLES R CHINA, DEC 16-18, 2016.
[9] Zhao G Y, Fu P, Zhou S W, et al. Dynamic Mechanical Analysis of Automotive Gearbox Casing. Advanced Materials Research, 2011, 230-232: 539-543.
[10] Zhang H Z, Zhang X X, Feng S L. Finite Element Analysis of a Certain Heavy Vehicle. Applied Mechanics and Materials, 2013, 454: 19-22.
[11] Patil P P, Kumar A. Dynamic Structural and Thermal Characteristics Analysis of Oil-Lubricated Multi-speed Transmission Gearbox: Variation of Load, Rotational Speed and Convection Heat Transfer. Iranian Journal of Science and Technology-Transactions of Mechanical Engineering, 2017, 41 (4): 281-291.