The Finite Element Modelling and Dynamic Characteristics Analysis about One Kind of Armoured Vehicles’ Fuel Tanks

Yang Gao¹, Zhishang Ge², Weihao Zhai³, Shiwang Tan¹ and Feng Zhang¹

¹. School of Mechanics, Civil Engineering and Architecture, Northwestern Polytechnical University, Xi’an 710129, P. R. China
². Shanghai Electro-Mechanical Engineering Institute, Shanghai 201109, P. R. China
³. Shanghai Aerospace Equipment Manufacturer, Shanghai 200245, P. R. China
Email: 849916464@qq.com

Abstract. The static and dynamic characteristics of fuel tank are studied for the armoured vehicle in this paper. The CATIA software is applied to build the CAD model of the armoured vehicles’ fuel tank, and the finite element model is established in ANSYS Workbench. The finite element method is carried out to analyze the static and dynamic mechanical properties of the fuel tank, and the first six orders of mode shapes and their frequencies are also computed and given in the paper, then the stress distribution diagram and the high stress areas are obtained. The results of the research provide some references to the fuel tanks’ design improvement, and give some guidance for the installation of the fuel tanks on armoured vehicles, and help to improve the properties and the service life of this kind of armoured vehicles’ fuel tanks.

1. Introduction

The armoured vehicles’ fuel tank is the most important energy reserves device, and the chronic jolt and vibration at work acts on the fuel tank in the form of fatigue load, resulting in the failure of its structure due to fatigue fracture [1]. Once the fuel tank fault occurs, such as cracks, damage and other failures, it will cause fuel leakage, which restricts the sphere of activities of armoured vehicles. In addition, the hidden troubles caused by fuel leakage are also quite serious. Therefore, it is necessary to carry out the static and dynamic analysis for armoured vehicles’ fuel tank, and provide guidance for the design and improvement of fuel tank, and then improve the life of fuel tank, which is one of the important tasks to improve the overall performance of armoured vehicles.

The static strength of the fuel tank must conform to the design requirements, it is the prime condition to decide whether it can be used or not, and there has been many research results on static strength analysis about ordinary vehicles’ fuel tank at home and abroad. Chen carried out static strength analysis for heavy-duty trucks’ fuel tank by using finite element method and experimental study [2]. Enoki used shell element to model an automobile fuel tank, carried out the static and dynamic analysis of model structure, and compared the two results, and proved that the shell element is reasonable for finite element analysis of automobile fuel tank [3].

In addition, the analysis of the dynamic strength and vibration characteristics of the fuel tank is an important approach to study its fatigue characteristics and effective life, and modal analysis is usually used for such research work. Fu calculated the modal of a kind of medium-sized truck and its component, analyzed the vibration characteristics, and identified the cause of the abnormal vibration [4]. Zhang carried out the modal analysis for the automobile fuel tank, analyzed the natural frequency and the vibration pattern of the first three order modals, and found the reasons of the fuel tank
cracking [5]. Fichera applied the modal analysis method and the nonlinear finite element method to the analysis of the automobile fuel tank, and the finite element analysis of the fuel tank is carried out by using the multi-body dynamics method, the relevant parameters of the fuel tank are optimized [6].

Based on the existing research on ordinary vehicles’ fuel tank, in this paper, the corresponding research results and conclusions are applied to the research and analysis of armoured vehicles’ fuel tank, and the static and dynamic strength analysis are carried out by using the finite element method, which can provide reference for the improvement of the fuel tank design and the installation and fixation on the vehicle body.

2. The Establishment of Fuel Tank Model
The structure of actual armoured vehicles’ fuel tank is quite complex, in order to effectively improve the efficiency of finite element calculation and analysis, and to guarantee the consistency between the fuel tank model and the actual structure, the model of fuel tank is simplified and assumed as follows[7,8]:

(1) Ignore the holes on the fuel tank and the internal pipelines, including the mounting holes of fuel pump, the fuel filler, and the fuel outlet and so on.

(2) The influence of the weld seam between plates on the finite element calculation is not considered, the whole fuel tank and the welding edge are regarded as an integral enclosure with uniform thickness.

(3) Remove the fixed support on the fuel tank; the fixed constraint load is applied directly to the finite element model at the corresponding location of the fixed support.

In this paper, the CAD model of fuel tank is established by CATIA software. The fuel tank is designed according to the layout requirements of the interior space of armoured vehicle, and it is a regular rectangular shape. The interior of the fuel tank has two frames; the distance between the two frames and their respective adjacent side panels is 0.333m. The outer plate is designed to be four pieces, the long side is welded with two L plates, the two sides are blocked with two rectangular plates, the front, the back, the left and the right sides have predetermined grooves respectively generated by the stamping technology, and the thickness of all plates is 2mm. The two L plates have a R5 bend, the fuel tank model is shown in figure 1.

![Figure 1. CAD model of fuel tank in CATIA](image)

3. Static Analysis of Fuel Tank
Since the fuel tank uses the thin-walled structure, so the general shell element is selected as the calculation unit [9]. The Shell63 shell element has both bending capacity and membrane forces, so it can withstand in-plane loads and normal loads, each node of the element has six degrees of freedom (The translation along the nodal coordinate system X, Y, Z directions and the rotation along the axis X, Y, Z of nodal coordinate system), stress stiffening and large deformation capacity have been taken into account. Comprehensive comparison of the conditions and characteristics of various shell units, the Shell63 shell element is more suitable for the force analysis requirement of the fuel tank. Therefore, the Shell63 shell element is selected as the unit for the finite element analysis of the fuel tank.
The material selected for the fuel tank is structural steel, its density is 7850kg/m$^3$, the elastic modulus is $2\times10^9$Pa, the bulk modulus is $1.6667\times10^9$Pa, the shear modulus is $7.6923\times10^8$Pa, and the Poisson ratio is 0.3.

The finite element mesh of the fuel tank can be automatically generated by means of the ANSYS Workbench meshing tool. For some areas with high precision requirement, in the process of meshing, the meshing tool will automatically adjust the mesh density and mesh size, so that the generated mesh shape and characteristics are better, and the high quality of the grid is guaranteed [10]. The divided mesh number of the fuel tank model is 40794, and the number of nodes is 83692. The result of division is shown in Figure 2, where the minimum size of the mesh is 2.2093mm, the maximum surface size is 11.046mm, and the minimum edge length is 1.4575mm. Compared with the overall size of the fuel tank, the maximum size of the mesh is relatively small, which meets the requirement of the calculation accuracy.

Figure 2. Finite element model of fuel tank

The main fixed form of the fuel tank is to clamp the two sides of the fuel tank with a fixture; the fixture is located on the fuel tank mounting base of armoured vehicle’s body. In the course of work, the fuel tank is fixed relative to the fixture, so the applied boundary condition is to fix two side of the fuel tank.

According to the industry standard of automobile fuel tank, it should work under the requirement of 0.7MPa working pressure, and the fuel tank stress must meet the strength requirement. Taking into account the special work environment of the armoured vehicle’s fuel tank, and in order to meet the structural strength requirement of the armoured vehicle’s fuel tank, so the force model added to the fuel tank is that all internal surfaces are subjected to the 1.4MP distributed normal compressive outwardly stresses.

After applying boundary conditions to the finite element model of the fuel tank, ANSYS Workbench solver is used to analyze and solve the model, and the results are shown in figure 3 and figure 4.
The results show that regions of high stress exist at the upper and lower surfaces, the middle area of the front and back surfaces, and both sides of the front and back surfaces of fuel tank, the stress in the area connected to the bulkhead is a little bit low, the maximum stress is at the middle of the connection between the bulkhead and the box, the value of maximum stress is $1.1977 \times 10^9 \text{Pa}$. Therefore, it is necessary to reduce the high stress zone and improve the overall stiffness of the fuel tank by improving the structure and installation location of the internal bulkhead, so as to ensure that the structural strength meets the work requirements.

4. Dynamic Characteristics Analysis of Fuel Tank
Based on the finite element model established by the static strength analysis of the fuel tank, the modal analysis of the fuel tank is carried out and its dynamic characteristics are studied. Modal analysis is an important part of structural dynamics analysis. Based on the theory of structural dynamics analysis, and combined with the finite element modal analysis method, the dynamic characteristics of the fuel tank are analyzed by using ANSYS Workbench finite element analysis software. The fuel tank can be redesigned and improved according to the analysis results, in order to guarantee good working performance and safe reliability of fuel tank [11].
The boundary condition of the fuel tank has significant influence on its modal vibration and frequency. In free modal analysis, the boundary condition is determined as free boundary that is without any restraint. However, the free modal analysis is only a general analysis for the dynamic characteristics of the fuel tank. If the comprehensive and accurate dynamic characteristics of the fuel tank is desired, then it better to carry out the constrained modal analysis. Therefore, this paper leaves out the analysis process of free modal, and the constrained modal analysis of the fuel tank is carried out directly. After exerting the boundary condition with fixed constraint on the two side of the fuel tank, considering that the low order modals have great influence on the fuel tank structure, so the first six order modals of the fuel tank is calculated and used to analyse [12].

The calculated natural frequency of the first six order modals respectively are 64.762Hz, 65.838Hz, 97.905Hz, 105.3Hz, 127.5Hz and 129.37Hz. The first six order modals vibration are shown in figure 5(The Max and Min in the figure represent the location of maximum amplitude and minimum amplitude).

Figure 5. The first six orders modal of fuel tank
The results of modal analysis show that each order natural frequency distribution range of the fuel tank is dispersive; therefore, it may cause the resonance of the fuel tank body under the load of the actual road condition. The natural frequency of the first and second order modal of the fuel tank is very close, there is no modal vibration. The third-order modal vibration shows that there is a single peak fluctuation vibration, and the location is the middle area of the connection between the upper surface of the case and rear surface of the case, the amplitude gradually decreases from the junction to the outside. Obvious deformation areas can be seen in the fourth-order modal and fifth-order modal, there are three peaks fluctuation in the middle area of the front and rear surface and the two areas adjacent to two sides. The area of greater amplitude is mainly focused on the four front, rears, up and down plates. The area in the front and rear plates between two frames is the main vibration area. In the sixth-order modal, the amplitude of the fuel tank body decreases gradually, and the maximum amplitude area is at the internal bulkhead of fuel tank. The weakness of the fuel tank is at the larger amplitude of the modal vibration, that is, the area between the two bulkheads on the fuel tank and the two bulkheads. Therefore, it is necessary to enhance the stiffness of the tank body, the bulkhead and the connection between the bulkhead and the tank body, so as to improve the vibration characteristics of the fuel tank.

5. Conclusion
By analyzing the static and dynamic characteristics of the fuel tank model, it can be obtained that the structural design of the fuel tank meets the requirements of static strength, and structural damage will not occur, and the high stress zones are focused on the bulkhead and edge. Therefore, the high stress area should be dispersed by reasonable design of the structural shape and installation location of internal bulkhead. And in order to make the overall force of fuel tank at low level, it is necessary to increase the damping effect by enhancing the stiffness of the tank at the same time. In addition, the welding technology between components should be improved, so as to improve the performance of welds at components’ connection. The L plate of the fuel tank should use a large radius circle cutter when bending plates, which prevents stress concentration. And sufficient welding areas should be reserved, so as to make sure fully overlap between weld components when adopting the lap welding method, and the connection strength meets the requirements.

6. Acknowledgements
Authors gratefully acknowledge the support of China Scholarship Council, the fundamental research funds for the central universities (NPU-FFR-3102015BJ (II) JL04), natural science foundation of Shaanxi province (2016JQ5109), and the seed foundation of innovation and creation for graduate students in Northwestern Polytechnical University (Z2017116).

7. References
[1] Yang T, Wang HY. Research on fatigue crack growth life prediction technique for the armoured hull [J]. Acte Armamentarll, 2010, (02): 129-134.
[2] Chen C. Reserch on finite element static characteristics of a heavy-duty truck frame and experiment [D]. Hefei: HeFei University of Technology, 2009
[3] Enoki S. Automotive fuel tank dynamic and static strength analysis [J]. Computers And Structures, 2000,23(3): 65-70
[4] Fu CH. Study on vibration problem and solution of truck [D]. Hefei: HeFei University of Technology, 2010
[5] Zhang Y. Dynamic and static strength analysis of automotive fuel tank [D]. Liuzhou: Guangxi University of Science and technology,2013
[6] Fichera G, Lacagnina M. Modeling and finite element analysis of automotive fuel tank by using multibody method. Multibody System Dynamics, 2005; 27(4): 30-36
[7] Wang Y. Study on automobile fuel tank structure design and forming quality control [D]. Chongqing: Chongqing University, 2004
[8] Du PJ. FEM analysis and gas pressure intensity test of FL-61 wind tunnel.[J] Science Technology and Engineering, 2017,17(08): 303-306
[9] Xie YP, Feng G. Study of ANSYS modelling on the combination of 3D entity unit and plate shell unit [J]. Journal of Machine Design, 2009, (04): 5-7
[10] Mou WJ, Yang JZ, Chen JY, et al. Finite element analysis for power car skeleton based on ANSYS Workbench[J]. Science Technology and Engineering, 2010,(22): 5592-5594
[11] Cao YY, Zhao DF. Finite Element modal analysis theory and application [J]. Mechanical Engineer and Automation, 2007, (01): 73-74.
[12] Wang JL, Feng BL, Zhang CM, et al. Dynamic characteristics analyses of weighing sensor fixture based on finite element method [J]. Science Technology and Engineering, 2017; (20): 28-34