U.P.B. Sci. Bull., Series D, Vol. 80, Iss. 2, 2018 ISSN 1454-2358

DYNAMIC CHARACTERISTICS ANALYSIS OF ENGINE VALVE TAPPET

Wu WENJIANG[1], Guo BIN[2], Zheng MINGJUN[2], Gao ZHANFENG[1]*, Tian YUNFENG[3]

The tappet is one of the important parts of the engine. Its performance directly affects the working performance of the valve train. Therefore, it is of great significance to study the dynamic characteristics of the tappet. Taking a diesel engine as the research object, the structural strength, modal and contact stress of diesel engine's tappets which control inlet and exhaust are carried out in this paper by means of finite element analysis method. The static strength analysis result shows that the static strength model engine's intake and exhaust valve tappets all meet the designed requirements. The result of modal analysis shows that intake and exhaust valve tappets will not resonate during its working process. According to the results of contact stress analysis, the dangerous parts of the intake and exhaust valve can be determined, which provide an important reference for the further optimization design of the tappets' structure.

Keywords: Engine Tappet; Strength Analysis; Modal Analysis; Contact Stress Analysis

1. Introduction

Valve mechanism is an important part of the engine. According to the order of the cylinder ignition, it can close and open to achieve the engine ventilation process based on a certain time law [1, 2]. When the engine is working normally, the parts of the valve mechanism are frictional in contact with each other and different degree of elastic deformation will occur at different parts. The consequence will cause the engine system to vibrate violently. In serious cases, the engine shakes will cause the valve seat rebounding, the valve fast disengaged, and so on. The above phenomenon will result in internal failure of the engine. And the engine cannot perform continuous operation normally [3, 4]. Therefore, whether the design and optimization of the valve mechanism is rational will directly affect the engine's reliability, economy, stability and durability [5]. In this paper, the three-dimensional model of the intake and exhaust tappet of the engine

[1] Engineering training center, Shijiazhuang Tiedao University, Shijiazhuang, P.R. China
*Email: gaozhanfeng88@126.com.
[2] College of Mechanical Engineering, Shijiazhuang Tiedao University, Shijiazhuang, P.R. China
[3] Technology Research Institute, Great Wall Motor Co., Ltd, Baoding, P.R. China.

valve mechanism is established. The static strength analysis, modal analysis and contact stress analysis of the tappets are carried out by using the finite element software MSC.Patran and MSC.Nastran, respectively. It is of great significance to optimize the structure and predict fatigue life of the engine.

2. Establishment of finite element model

The geometric model of the tappet can be directly imported into the MSC.Patran and be transformed into a finite element model. The finite element analysis process consists of the grid partition, adding boundary conditions and defining material properties in the MSC. Patran [6]. The CAD drawing of the tappet provided by the manufacturer is shown in Fig.1. The 3D model of the tappet built according to the Fig.1 is shown in Fig.2.

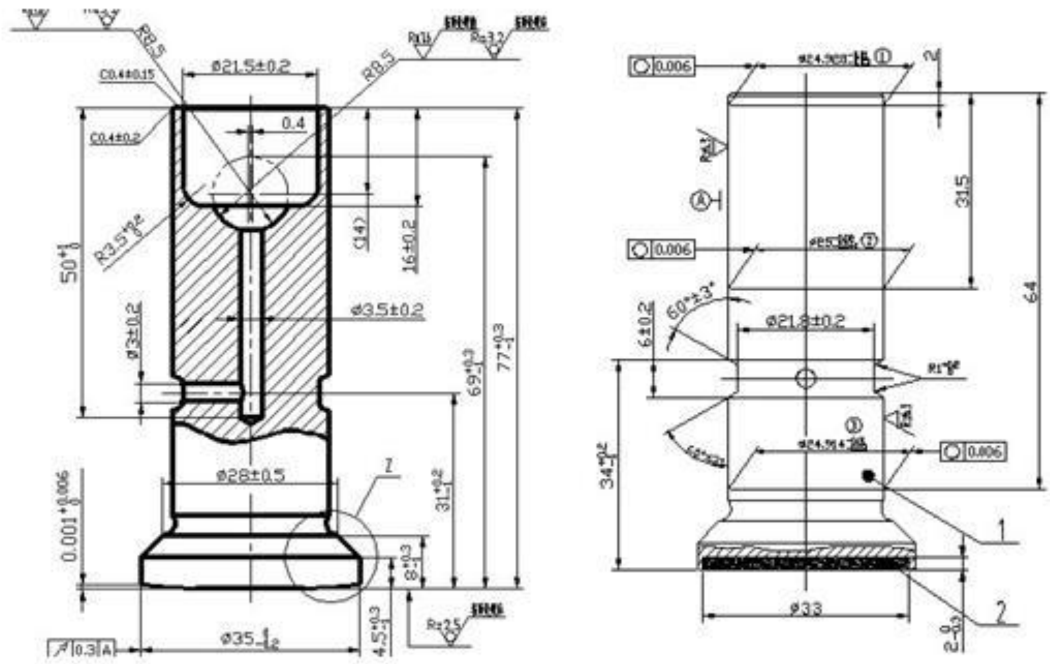

Fig.1 Tappet CAD drawings

Fig.2 Tappet three-dimensional model

There are several ways to mesh in Patran, including ISO meshing, free meshing, mixed meshing, sweeping and dragging meshing, and so on. In this paper, the lattice of the tappet is divided into 56131 second-order tetrahedral elements (Tet10) and 81975 nodes by using the free meshing method. The obtained model is shown in Fig.3. The tappet is divided into the intake tappet and exhaust tappet, for which their material properties are different. They must be established separately when defining their material properties. The material of the exhaust tappet is round steel 20Cr and the corresponding national standard is 26-2-GB/T702-2004. The lower end of the exhaust tappet is inlaid with cemented carbide whose national standard is GB / T11101-2209. The intake tappet material is made of alloy cast iron and the corresponding national standard is GH-345CuCr61. The specimens were heat treated by the test. The camshaft material tested is the 45# steel. Its surface hardness is about from 55HRC to 61HRC. The tip hardness of the camshaft is around from 606HV to 685HV. The base diameter of the cam is 52.16 mm. The height from the bottom of the exhaust cam to its tip is 61.0 mm, and the height from the bottom of the intake cam to its tip is 60.7 mm. The parameters of the relevant material properties which are based on the MTS bench test [7，8] and the national standard are shown in Table 1. The test bench used is the MTS 810. The specimen test is shown in Fig.4.

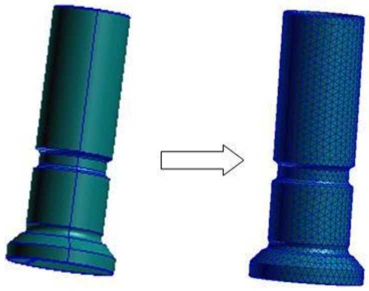

Fig.3 Tappet Grid model

Fig.4 Tensile testing

Table 1

The properties of the tappet material

Name	Material	Elastic Modulus [GPa]	σ_s [MPa]	σ_b [MPa]	Poisson's ratio
Intake tappet	Alloy cast iron	157	322	402	0.27
Exhaust tappet	Round steel	206	462	577	0.3
	Carbide	206	960	1200	0.3

In the above table, the yield strength 'σ_s' is the yield limit when the metal material yields, that is, the stress that resists the microplastic deformation. The tensile strength 'σ_b' is the maximum bearing capacity of metal under static tensile conditions.

When the tappet is working, the tappet will tilt due to the lateral thrust of the cam in a certain direction. The tilt will cause wear between the single side of the cylinder pipe and the tappet. The cam surface is constantly in contact with the fixed point of the tappet bottom，which results in uneven wear of the working contact surface. In order to reduce the amount of tappet surface wear，the exhaust tappet has a small taper on the top surface of the structure. The contact line between the cam and the tappet is offset from the axis of the tappet [9]. The tappet is raised by the cam when the engine system is in working state. The tappet rotates about its own axis under the friction force between the contact points to achieve uniform wear. The tappet moves up and down and rotates during its working period [10]. In order to make the movement of the tappets coincide with the actual movement, the boundary conditions in Patran are added as follows: some constraints need to be assumed in the simulation process. For example, translational freedom and rotational freedom in the X and Z directions are restricted, only translational freedom and rotational freedom in the Y directions are free; the boundary constraints of the tappet of the intake and exhaust column are the same. Then the model could be set up after constraints and load are added. The cam-tappet pair model with boundary constraints is shown as Fig.5.

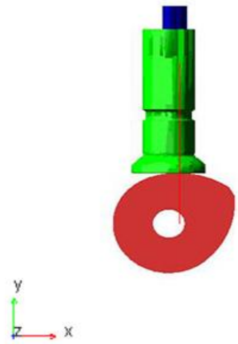

Fig.5 Boundary constraints

3. Analysis of Dynamic Characteristics

Structural Strength Analysis. Firstly, strength of intake valve tappet structure is analyzed. According to the finite element model established above, the stress nephogram and strain nephogram diagram of the intake train tappet with the maximum load are obtained, as shown in Fig. 6 and Fig. 7.

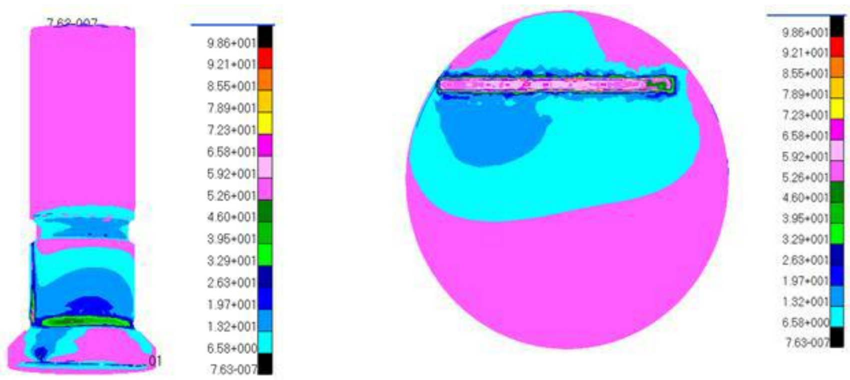

Fig. 6 Intake valve tappet stress nephogram

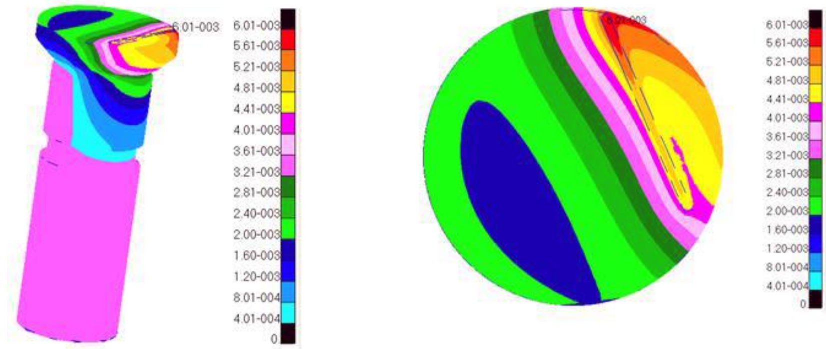

Fig. 7 Intake valve tappet strain nephogram

Secondly, strength of exhaust valve tappet structure is analyzed, too. Similarly, through the analysis as above, the stress nephogram and strain nephogram of the exhaust valve tappet under the maximum load are obtained, as shown in Fig.8 and Fig.9.

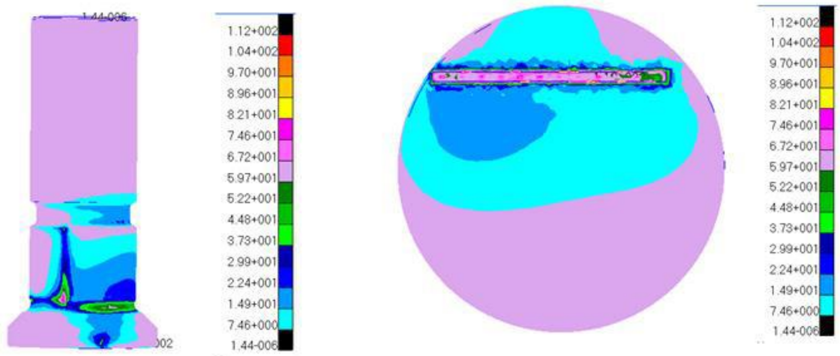

Fig. 8 Exhaust valve tappet stress nephogram

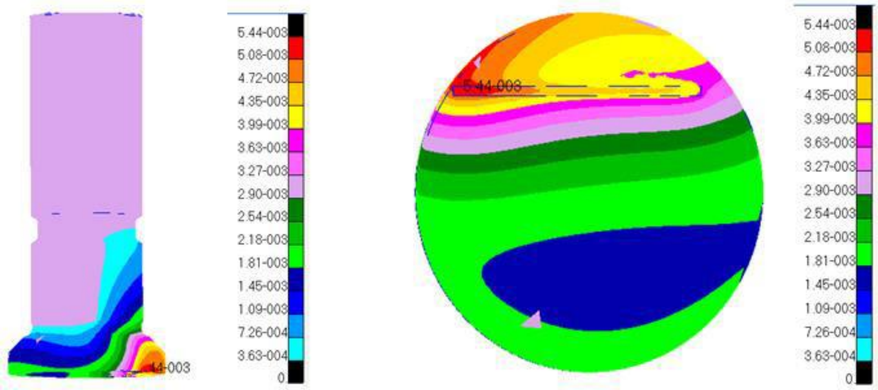

Fig. 9 Exhaust valve tappet strain nephogram

The material of the intake valve tappet is alloy cast iron GH-345CuCr61 and the material of the exhaust tappet matrix is round steel 20Cr. As can be seen from Table 1, the yield limit and the strength limit of the intake valve tappet material are $\sigma_s \geq 322$MPa and $\sigma_b \geq 402$ MPa respectively; the yield limit and the strength limit of the exhaust valve taper material are $\sigma_s \geq 462$ MPa and $\sigma_b \geq 577$ MPa respectively. As can be seen from Fig.6 and Fig.8, the maximum stress of the intake valve tappet is 98.6 MPa under the maximum load. The maximum stress of the exhaust valve tappet is 112 MPa. Compare the results with the given data in Table 1, both the maximum stress of the intake valve tappet and the exhaust valve tappet are far less than the yield limit and the strength limit. In addition, the cemented carbide layer embedded on the top of the tappet has a tensile strength of 1200 MPa. It can be seen from Fig.7 and Fig.9 that the maximum deformation of both the intake valve tappet and the exhaust valve tappet are very small, which is 6.01e-3mm and 5.44e-3mm respectively. Namely, both the intake and the exhaust valve tappet designs are in line with the requirements.

Modal Analysis. When the tappet's working frequency from external excitation is equal to its inherent vibration frequency or has an integral multiple of its natural frequency, the tappet resonates. The resonance of the valve will bring various degrees of damage which lead the valve to rebound and shock. In serious cases, it is possible to damage the entire valve system. The vibration characteristics of the tappet can be obtained through modal analysis. In this paper, the vibration characteristic of the tappet is analyzed under Free State and no load is applied by using modal analysis method. The material properties of the intake and exhaust valve tappet for the engine we adopted are different, therefore the material properties of the intake and exhaust valve tappet must be added into respectively before performing the modal analysis [11, 12]. After importing the geometrical model of x_t Parasolid, the finite element mesh of the entity was divided by using the automatic grid generator. Then the model was used to generate 10 nodes tetrahedral finite element grids, which consists of a total of 56131 units and 81975 nodes. When the '.mnf' files generated by Nastran were used by Adams, there is a problem about how to convert the unit in the two software. The unit 'T' in Nastran should be converted to the unit 'kg' in Adams so that the right conclusion will be obtained. The intake and exhaust valve tappets are of the same structure, so the modal analysis results are not much different [13]. The ten order vibration modes of the intake valve tappet are shown in Fig. 10.

The natural vibration frequencies of the first ten orders of the intake and exhaust valve tappet are shown in Table 2.

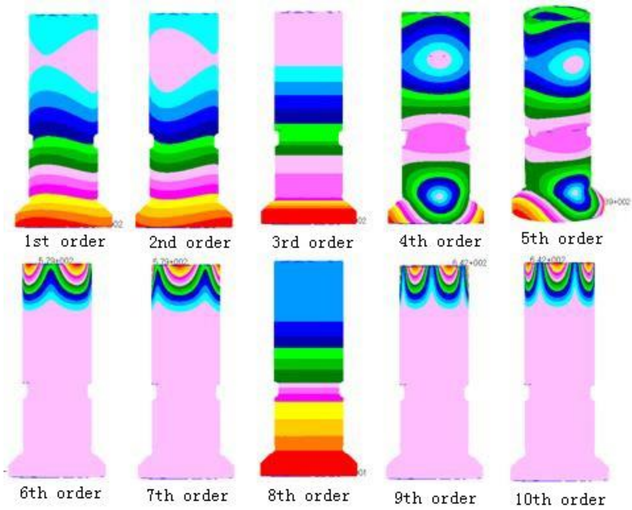

Fig. 10 Tappet's 10 vibration mode

Table 2

The natural frequency of each order

Order	Intake [Hz]	Exhaust [Hz]	Order	Intake [Hz]	Exhaust [Hz]
First order	3054.4	3449.1	Sixth Order	15330	17677
Second order	3054.8	3461.9	Seventh Order	15335	17683
Third order	8776.9	9787.6	Eighth Order	16282	18488
Fourth order	15044	16923	Ninth Order	22346	26346
Fifth order	15045	17084	Tenth Order	22352	26351

As can be seen from Table 2, among the first ten natural frequency of the intake and exhaust valve tappet, the first-order natural frequencies are the lowest, which are 3054.4 Hz and 3449.1 Hz respectively. The theoretical calculation of the resonant frequency of the tappet is 17.5 Hz. Therefore, the results show that the intake and exhaust valve tappet does not resonate during the working process.

Contact Stress Analysis. The engine valve mechanism has the characteristics of high speed and high flexibility. It is not reasonable enough to regard the engine valve mechanism as a complete rigid body when calculating the contact stress between the cam and the tappet. So, the impact of elastic deformation for the engine valve mechanism must be considered when calculating the contact stress. In this research, the valve mechanism is defined as a flexible body when performing dynamic simulation analysis. The research object is a kind of valve mechanism system which the cam is assembled under the valve mechanism. The 3D model is established in the Pro/E and a variety of constraints are added. Then the tappet model and some components with smaller rigidity are imported into the finite element software Patran. Next these model elements are discretized. At last， the transformation of the finite element model is done and the corresponding files are generated, which include modal information such as nodes, units, part materials, and stress and strain. The information included in the above files is used by the Adams software to generate a flexible body. Based on the flexible body, an object that is used to carry out contact stress analysis is established. Considering the topological relation of each moving component, the dynamic simulation model of valve mechanism with rigid-flexible coupling multi-body system of the valve mechanism is established, regardless of the valve clearance and the imposed corresponding constraints between the other components and the object established. The deformation of the flexible body can be regarded as the superposition on each other of the finite element modal modes through the calculation of simulation dynamics. The dynamic characteristics of the tappets in the valve mechanism can be expressed.

There are multiple failure forms in the valve mechanism system. The most important form of failure is the fatigue failure caused by the wear between the cam and the tappet. The contact stress between the cam and the tappet is an important indicator which is used to evaluate the friction and wear of the cam and the tappet [14]. In order to study the wear problem of the valve train, it is necessary to study the contact stress between the cam and the tappet. The contact stress between the cam and the tappet can be used to estimate the reliability of the contact surface [15]. The contact stress nephogram between the cam and the intake and the exhaust valve tappets can be obtained with the flexible body dynamics model through the simulation of the finite element software Patran, as shown in Fig.11 and Fig.12.

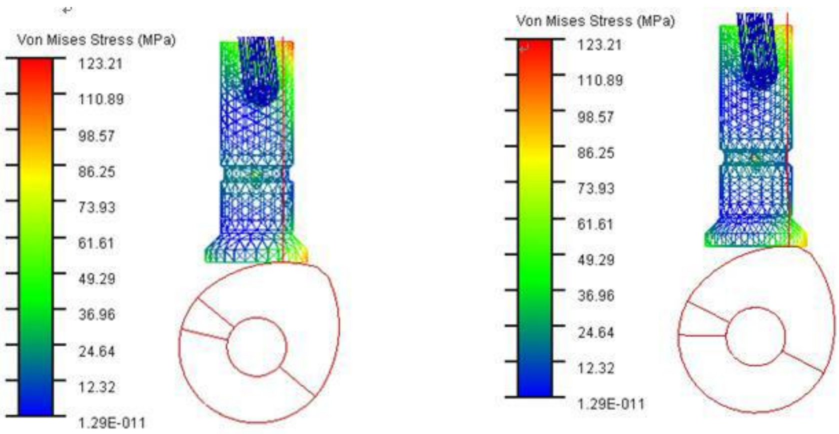

Fig.11 Intake valve tappet contact stress nephogram

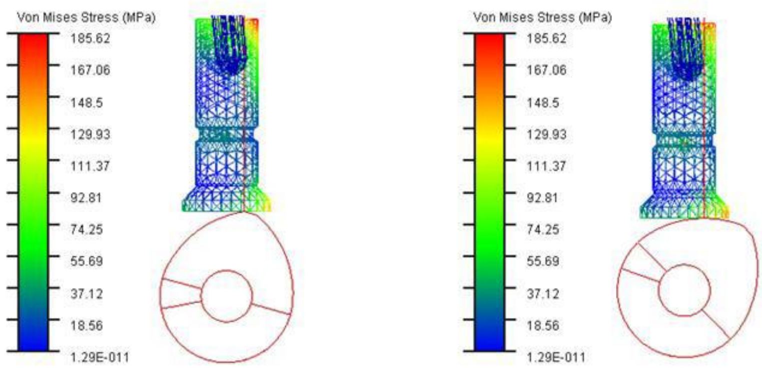

Fig.12 Exhaust valve tappet contact stress nephogram

It can be seen from Fig. 11 that the maximum contact stress of the cam-tappet contact surface is 123.21 MPa at the intake position, which is slightly larger than the static stress peak of the tappet under the maximum load. As it can be seen in Fig. 12, the maximum contact stress of the cam-tappet contact surface is 185.62 MPa at the exhaust position, which is also greater than the static stress peak of the tappet under maximum load. This shows that the cam-tappet produces a greater stress in the contact condition than the static load alone, which requires checking whether the contact stress is within the allowable stress range of the design. It can be seen from Table 1 that the yield limit of the intake valve tappet material is $\sigma_s \geq 322$MPa, the strength limit $\sigma_b \geq 402$ MPa, the exhaust column taper yield is $\sigma_s \geq 462$ MPa, the strength limit is $\sigma_b \geq 577$ MPa. It can be seen from Fig.11

and Fig.12 that the stress peak of the cam-intake valve tappet is 123.21 MPa and the peak value of the cam-exhaust valve tappet is 185.62 MPa, which is far less than the yield limit and the strength limit given by Table 1. The compared results show that the contact stress between the intake and exhaust valve tappet meets the requirements.

Simultaneously, it can be seen from Fig.11 and Fig.12 that the peak value of the contact stress of the cam-tappet pair is at the maximum position of the cam and the tappet. The contact stress and the static value are basically the same as the camshaft rotation. In addition, the contact stress between the cam and the tappet is not evenly distributed in the axial contact area of the camshaft because the tappet itself rotates during the actual operation of the engine.

4. Conclusion

In this paper, the intake valve tappet and the exhaust valve tappet material properties of a certain type of engine are determined by the tensile testing. The engine intake and exhaust valve tappets were analyzed with MSC.Patran and MSC.Nastran respectively. The results of structural strength analysis show that the strength of the intake valve tappet and the exhaust valve tappet in this model meet the strength requirements. During the work time, the intake valve tappet and exhaust valve tappets will not resonate according to the modal analysis. The critical areas of the cam-tap pair can be identified by the contact stress analysis. The peak of contact stress appeared where the cam and tappet had the highest static force. The research is significant for fatigue life prediction and wear analysis of tappets.

Acknowledgment

The authors would like to express their appreciation for the financial support provided by Hebei Province Natural Science Foundation (Grant No. E2017210166) and Hebei Province colleges and universities Science and Technology Research Project (Grant No. ZD2017068)

R E F E R E N C E S

[1]. *Xu Hongman*. Calculation of contact stress between cam and tappet. Internal combustion engine, 2001 (1): 25-26.
[2]. *Shen Jipeng*. Gasoline engine gas distribution mechanism dynamics analysis and cam profile optimization [D]. Jilin University, 2016.
[3]. *Yang Jianhua*. Variant hyperbolic function with gas cam profile characteristics. Journal of Shaoyang University, Natural Science Edition, 2010, 7 (3): 39-42.
[4]. *Liu Xiaolin*. Engine cam abnormal wear problem [D]. Tsinghua University, 2014.

[5]. *Guo Hongliang*. Engine dynamic mechanism of the experimental study. China Science and Technology, 2010 (24): 12-15.
[6]. *HE Shaohua*. A multi-body system impact response spectrum analysis method. Vibration and Shock, 2011, 30 (4): 127-130.
[7]. *Jin Yugang*. High-speed gasoline engine gas distribution mechanism analysis and optimization [D]. Hunan University, 2014
[8]. *With Bo*. Micro-car Drive Bridge test system key technology research [D]. Zhejiang University, 2015.
[9]. *Gao Mingxiang*. Virtual environment and its behavior modeling technology research[D].Hubei University of Technology, 2008, 23 (2): 13-15.
[10]. *Tang Zhanfei*. Medium-sized CNG engine cam / tappet contact stress and fatigue wear failure analysis [D]. Dalian University of Technology, 2008.
[11]. *Du Shanyi*. Based on modeling and simulation and vibration analysis of the engine fault diagnosis method [D]. South China University of Technology, 2013
[12]. *Tian Yunfeng*. Fatigue life and experimental study of engine tappet [D]. Shijiazhuang Railway University, 2013
[13]. *Liu Peng*. Internal combustion engine gas distribution system dynamics and structural strength[D]. Huazhong University of Science and Technology, 2014
[14]. *Per Lind Helm*, Stefan Borland, Miguel Salvo Cortes. Characterization of Wear on a Cam Follower System in a Diesel Engine. Wear, 2003, 254 (11): 1199-1207.
[15]. *Tang Shao-Xiong, Ma Li, Yang Dai-Hua, et al*. Dynamic simulation of engine cam wear. Journal of Wuhan University of Technology, 2002, 24 (5): 117-120.