International Journal of Research in Engineering and Science (IJRES)
ISSN (Online): 2320-9364, ISSN (Print): 2320-9356
www.ijres.org Volume 4 Issue 8 ǁ August. 2016 ǁ PP. 01-07

Static Analysis of Valve Spring Based on ANSYS Workbench Software

Wenqiang Qi[1], XunchengWu[2], ShaXu[2]

[1](*College of Automotive Engineering, Shanghai University of Engineering Science, China*)
[2](*College of Automotive Engineering, Shanghai University of Engineering Science, China*)
[2](*College of Automotive Engineering, Shanghai University of Engineering Science, China*)

ABSTRACT*:* *Valve springis the important part of the valve assemblyin Internal combustion engine gas distribution mechanism. It can ensure the normal operation of the valve assembly. In order to study the stress distribution of valve spring in the working process, firstly, we make use of CATIA softwareto establish more accurate 3D model of valve springaccording to the different position of valve spring festival distance, then using ANSYS softwareto carry out finite element analysis of the valve spring mode. The results of finite element analysis show that the maximum stress value is distributed on the inner side of the valve springand the valve spring strength meet the requirements.*

Keywords*:* *Valve spring, stress distribution, CATIA, finite element analysis, maximum stress value*

I. INTRODUCTION

When the valve is opened, the valve can move accurately with the cam. At the same time, the valve spring is used to absorb the inertia force generated by the transmission parts in the process of opening and closing the valve to prevent the transmission parts from being separated from each other and the normal operation of the air distribution mechanism is destroyed. Valve springs is subjected to frequent alternating loads [1]. In addition, miniaturization, high speed is the development trend of automotive engine, which requires that valve spring not only has a strong design stress, but also the installation space should be reduced [2-3]. This requires that the design of valve spring should fully weigh the various performance indicators of valve spring to ensure that the various properties of the spring can achieve the best, which can improve the comprehensive performance of valve spring.

The actual workload type of valve spring is alternating loads, so valve spring is usually fractured because of the fatigue failure. In order to ensure reliable operation, the valve spring should have proper elastic, sufficient strength and fatigue strength. When the working frequency of valve spring is equal to or a multiple of its vibration frequency, resonance will occur. Strong resonance will break down the normal operation of the valve: valve bounce, seating impact and the spring break [4]. In the actual production process of valve spring, the rejection rate of valve spring is high because of failing to timely and effective online judge rejects. Therefore, it is particularly important that how to judge rejects online and the basis of judging rejects as well as adjusting the process parameters. The valve spring is shown in Figure 1-1.

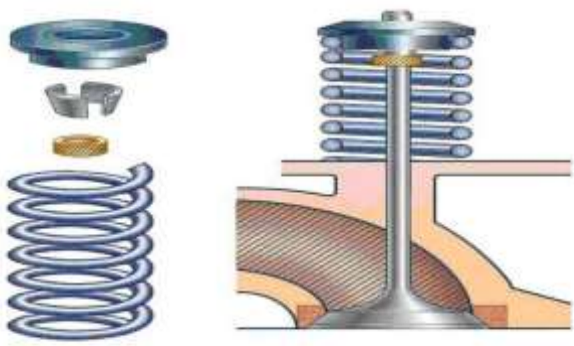


Fig.1-1 Diagram of valve spring

II. INTRODUCTION TO FINITE ELEMENT ANALYSIS

2.1 Introduction of finite element theory

The finite element method is based on the theory of elastic theory and computational mathematics, and then combining with the computer to solve the equation, which is a good numerical analysis technique for solving engineering problems. At present, the finite element method of analysis has been widely applied to various fields of analysis of engineering structures [5].

Modeling is the first step to solve the finite element method, so the finite element model is established in the finite element analysis. The standard of judgingthe model quality is good or bad is based on whether it can correctly simulate the physical characteristics of the analysis structure. Therefore, in order to better solve the problem, in the preprocess stage of the model need to pay attention to the following points:1, the connection quality ofelements and nodes;2, the nodespace position;3, the set of material parameters of the structure; 4 the combination of different elements; 5,the definition of load, constraints and boundary conditions. The above data is the foundation of the structure of finite element analysis. In addition, it also needs to consider theconvergence and rationalityofthe mesh shape and density distribution.The solving problems are more complex, the datais more and the requirements of the accuracy of data are more stringent.Becausein the preparation process of data, appearing a small mistake will lead to that the computational analysis cannot be carried out smoothly. It is an important and complicated process to carry out the finite element analysis of the structure to realize the reasonable structure design.

2.2 Finite element softwareintroduction

ANSYS Workbenchis themainstream analysis software in the CAE fields. Since its birth, with the development of the world information technology and the theory of finite element, it has been highly evaluated and widely used in various fields. ANSYS Workbench is a new generation of multi physics collaborative simulation environment developed by ANSYS Company. It contains three main modules [6]: (1) Parametric modeling (Design Modeler,DM) it has a CAD Software style and can facilitate the establishment of parametric model of structure ; (2) Analysis tool modules (Design Space,DS) it can implement static analysis, modal analysis, harmonic analysis, and other types of analysis ; (3) Optimum design of modules (Design Xplore,DX) it canshare the parameterization data from the DM , theDS and other CAD Systems, so that in the DX to change a variety of design options, so as to study the response of various scenariosin order to improve the reliability of the product.ANSYS Workbench set design, simulation and optimization, which can achieve the transfers of parameters between thedifferent function modules. So that different designers, departments and data can be able to work together in a unified environment [7].

III. Valve spring CAD Modeling

According to the given valve spring pitches in different locations, using three dimensional software CATIA to segment a spring model, and fairing curves of different segments of the spring , and we will save the CAD model for the.stp format which can identified by the ANSYS Workbench software. To build the valve spring model as shown in Figure 3-1.

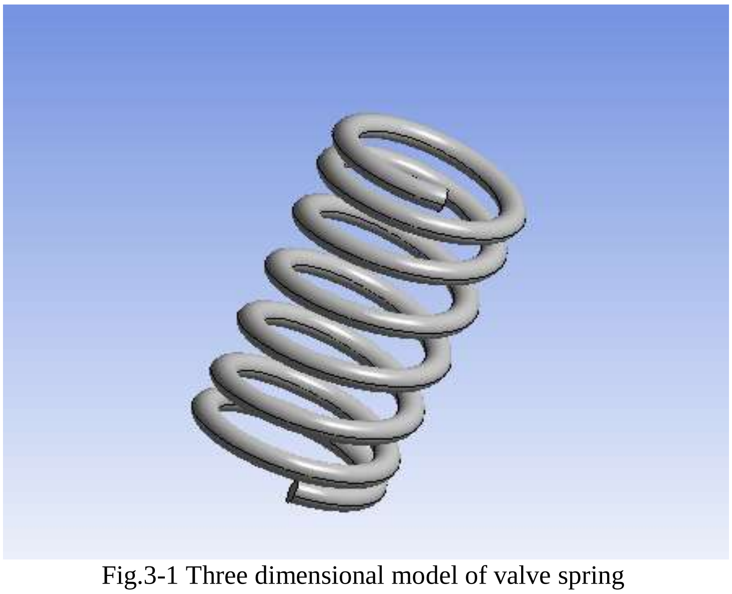


Fig.3-1 Three dimensional model of valve spring

The basic parameters of the valve spring are shown in table3-1 :

Table 3-1 Parameters of the valve springs

Symbol	**The name**	**Numerical**
$\boldsymbol{n_t}$	Total coils	7
$\boldsymbol{n}$	Effective number of rings	5
	Rotation direction	D
$\boldsymbol{H_0}$**/mm**	Free height	46.6
$\boldsymbol{D}$**/mm**	Pitch diameter	22.1
$\boldsymbol{d}$**/mm**	Wire diameter	3.1

IV. STATICS ANALYSIS OF VALVE SPRING

4.1Base of static analysis

Static analysis is one of the analysis methods thatengineering structure design engineers often use.And static load in general, includes the centralized / distributed static, temperature load, forced displacement, inertia force and so on. The static analysis can be easily obtained some variables, such as node force, constraint (reverse) force, element internal force, element stress, the strain energy and node displacement.The static analysis does not consider the damping, which is mainly used for the static strength calculation, the calculation of the deformation and stress of the component. What's more,it is usually used for the strength and stiffness of the product design stage [8-9].

In the static analysis of valve spring, the inertia force is neglected, and the influence of acceleration is not considered in the balance equation, so the whole system control equation is:

$$\left[K\right]\left\{u\right\} = \left\{F\right\} \quad (4\text{-}1)$$

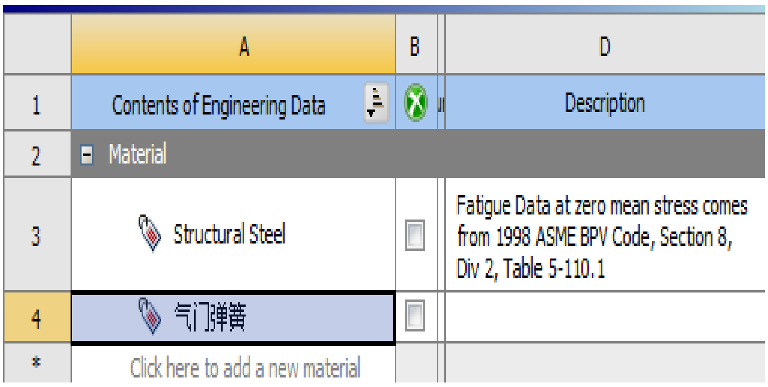
4.2 Material added and discretization of the model

The Materialof valve springs is QTEVA 70 SC .And it has the excellent high fatigue resistance and good anti relaxation propertiesunder the working environment of high temperature. The tensile strength of QTEVA 70 SC is 1910 to 2010MPa.The specific material parameters of valve spring are listed in the following table 4-1.

Table 4-1 Performance parameters for the valve spring

Parts	Spring modulus (Pa)	Poisson's ratio	Shear modulus (Pa)

In the finite element analysis software Workbench ANSYS, the material library contains a large number of original material models, so the material modelsare very rich, and we also can build new materials.As shown in Figure 4-1, the new material properties of the valve spring can be given. This simulation uses a custom form to add material, and we can create new material, custom elastic modulus, Poisson's ratio and so on.



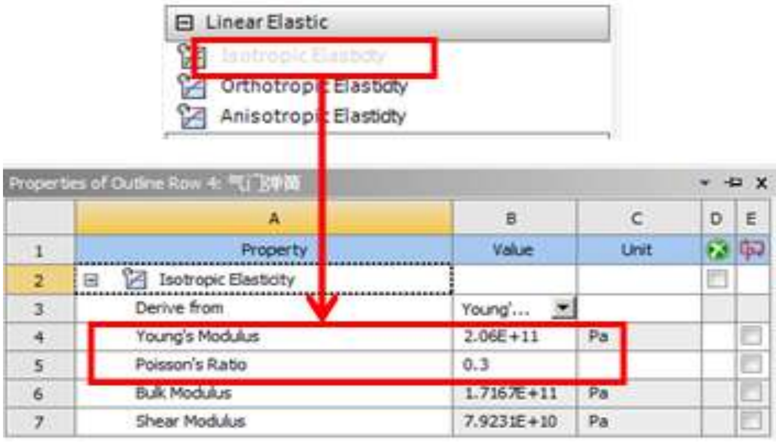


Fig.4 -1 Create new material model

The quality of the meshes in finite element software directly affects the precision and speed of the calculation of the ANSYS Workbenchsoftware, and ANSYS Workbench provides several methods of mesh division. According to the shape of the elements, themesh typescommonly usedcan be divided into: (1) Automatic mesh method;(2) Tetrahedral mesh method, which contains two kinds of methods, namely: Patch Conforming method, which is the Workbench own function; PatchIndependent method, which is mainly rely on the CFD ICENI software package; (3) The Swept Meshing. (4) The Multizonemethod, this is one of the highlights of the Workbench ANSYS mesh division; (5) The Hex Dominant. Due to the size and number of the mesh elements directlyaffect the efficiency and accuracy of the final results. Therefore, the mesh element division is very essential, and the mesh element division of all models will depend onthe same specifications in order to compare the stress ofdifferent models. Thedivision results of the model are shown in Table 4-2 and figure 4-2.

Table4-2The summary information of the modeldivision

Project	Related degrees	Related centers
Number	0	Medium

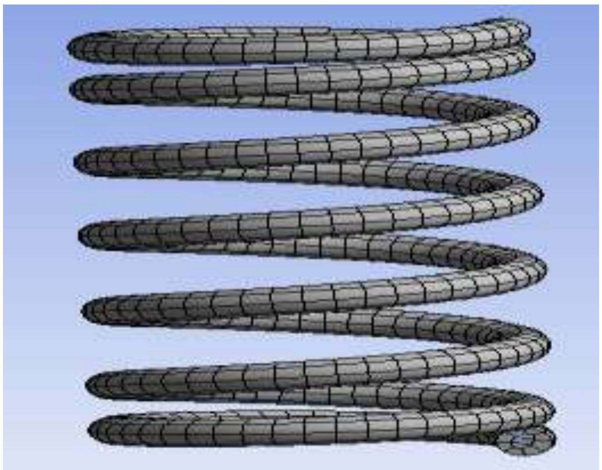


Fig.4-2The mesh division model

Mesh quality checks: Element Quality is a kind of common mesh quality inspection criteria. 1 represents a perfect cube or square; 0 indicates the volume is 0 or negative volume. The valve spring model has 18660 nodes, 3531 elements. The mesh quality check results are shown in figure 4-3.

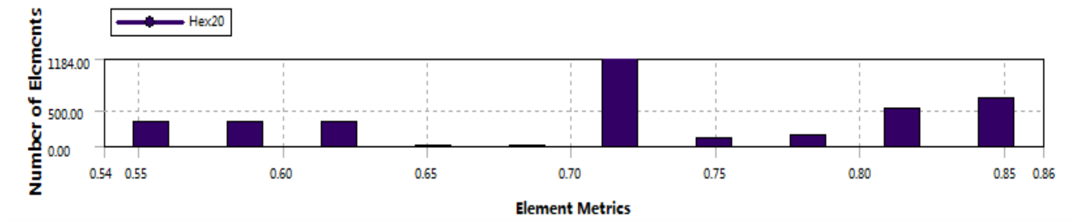


Fig.4-3 Results statistics

4.3 Contact Definitions of conditions and boundary conditions

Contacts in ANSYS Workbench are set as follows: finding connections, selecting the contactpropertiesand then selecting the contact surface. We need to create valve spring itself self-contact.ANSYS Workbench has five contact methods, respectively for the binding, no friction, non-separation contact, frictional contact, rough contact. The spring contact is frictional contact and it's more in line with the actual condition. The coefficient of friction is set to general 0.1 and the normal stiffness is set to 0.01.The smaller normal stiffness can reduce the difficulty of solving, and improve the convergence efficiency of the solution.

Selecting the nodes of the top circle of the spring model and using the remote force to control the selected nodes, applying concentrated force F=550N andas the B shown in figure 4-4. Due to the selected nodesare not in the same horizontal plane, we will apply the remote displacementat the same nodesto control he remote force in order to prevent the load tilted. Then the nodes will go along the Z axis displacement and the degrees of freedom of X axis and Y axis are restricted. As the A shown in figure 4-4.Then we will Select the appropriate location of the lower end of the spring and apply remote displacement to controlthe downward displacement of the distal spring ring ,as the C、D、Eshown in figure 4-4. Finally, we will fixall the degrees of freedom of the F position, as shown in figure 4-4.

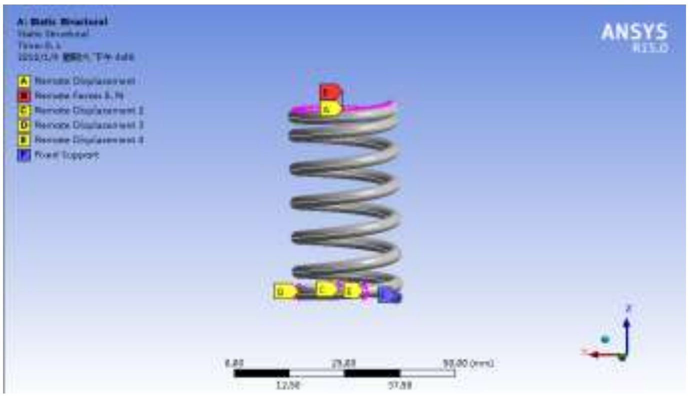


Fig.4-4Boundary conditions

V. STATIC ANALYSIS OF VALVE SPRING

5.1 Set solution and output requirements

After assigning materials, mesh and setting boundary conditions, we also need to set the solution option. Due to the spring load, the spring will have a large deformation. Therefore, it is very necessary to open the large deformation option in the process of solving the problem. Opening the large deformation in the solving process will lead to constantly updating the stiffness matrix of the spring, which will make convergence more difficult. In order to reduce the difficulty and increase the convergence efficiency, the analysis steps will be added to solve the problem and the load will be loaded step by step. At the same time, the sub analysis step can be set to prevent the penetration of the solution process. ANSYS Workbench does not need to set the output before calculating the results. After the end of the solution, we only need to click "Solution" to choose the output options, such as displacement, stress, strain, etc. As shown in Figure 5-1

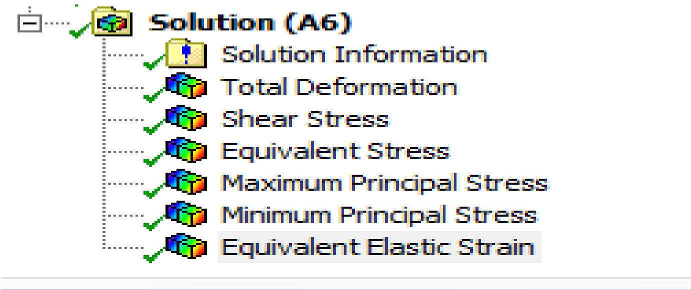


Fig.5-1Output settings

5.2 Stress and deformation

Figure 5-2 is the total deformationcloud of the valve spring under the force, we can see that the maximum deformation is 22.976mm; the minimum deformation is 0mm.

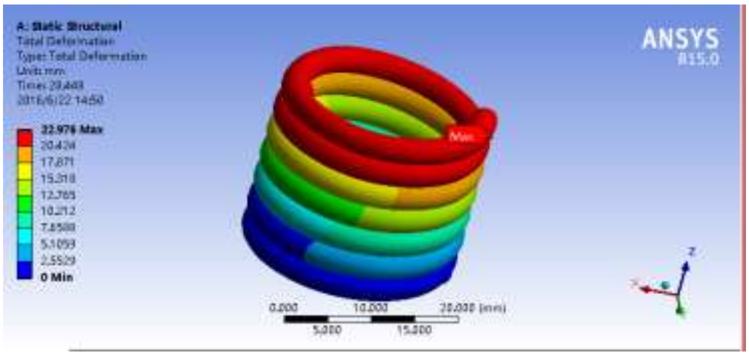


Fig.5-2 Total deformation

Figure 5-3 is the shear stress of valve spring, we can see that the maximum shear stress is 815.7Mpa, the minimum shear stress is -827.14Mpa

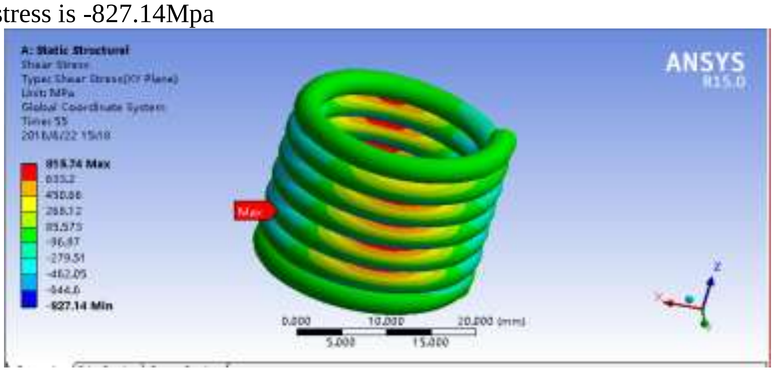


Fig.5-3 Shear stress

Figure 5-4 is the Mises stress cloud of valve spring, we can see that the maximum equivalent stress is 1690.6Mpa; the minimum equivalent stress is 0.594Mpa.

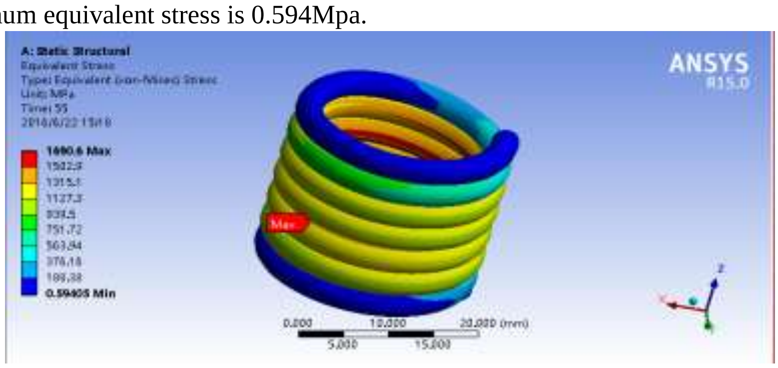


Fig.5-4 Equivalent stress

Figure 5-5 is the maximum principal stress of valve spring, we can see that the maximum value is 1017.7Mpa, the minimum value is -51.277Mpa.

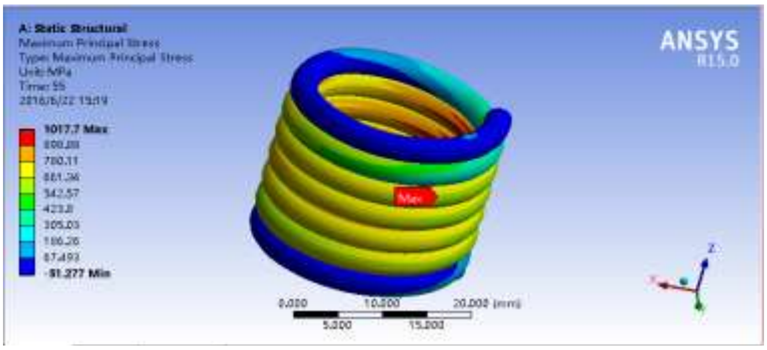


Fig.5-5 Maximum principal stress

The material of valve spring is the OTEVA 70 SC, which is the super pure steel. The size range of the material is 3.00-3.20mm and the tensile strength is 1960-2010MPa.In the F=550N condition, from the above stress contour of the valve spring we can know that the maximum value of Mises stress is 1690.6Mpa. Meanwhile, because of the contact problem, the local area of spring will occur stress concentration. The maximum value of stress 1690.6MPa appears on the inside of the spring and the minimum stress 0.594MPa appears on the outside of the spring. What's more, its stress value is less than the spring material maximum strength limit 1960MPa and meeting its strength and stiffness requirements. The deformation of valve spring along the Z direction is shown in figure5-2. The deformation of Z direction under the working load is 22.976mmand according to the theoretical value calculation formula (5-1):

$$\delta_{\max} = \frac{F_{\max}}{k} = \frac{1690.6}{77.9869} = 21.678mm \qquad (5\text{-}1)$$

Due to the spring support ring of the actual deformation is not considered in the derivation of the formula, and we also don't take into account the deformation between the spring seat and the spring. What's more, the spring radial deformation has a certain influence on the stiffness. Therefore, a certain error between the actual calculation result and the theoretical value is allowed.

VI. CONCLUSION

(1)The axial deformation of valve springunder the working load is22.976mm andthe result of theoretical value calculation is 21.678mm.Due to formula calculation does not take into account the deformation between the seat and spring, so a certain error is allowed. This shows that the design of the valve spring can meet the requirements of the work.
(2)After work load is imposed on the valve spring,the stress cloud chart indicates thatthe stressmaximumvalve of valve springunder the maximum working load is 1690.6MPa. It is far less than the spring material strength limit 1960Mpa.This shows that the designed valve spring meets the strength requirements.
(3) Under the working load, the maximum stress occurs on the inside of the valve spring and the minimum stress occurs in the outsideof the spring.The analysis results show that the stress of the outer ring of thevalve spring is obviously smaller than that of the inner ring andthe stress of the bearing ring is also relatively small, which is consistent with the fact that the inner ring of the spring is most likely to be damaged in the actual work of valve spring.

REFERENCES

[1]. FengguoTian, GuiyueKou..The optimization design and calculation of diesel engine valve spring[J]. ManufacturingAutomation, 2011,33(1): 43-51.
[2]. XunGong. The high strengthdevelopment direction of theautomotive suspension spring [J]. Mechanical and electrical product guide ,2003,18(3):67-70
[3]. BoLiu. Automotive suspension spring and its development trend [J]. Modern parts,2007,(12):56-59.
[4]. Ying Han. Three dimensional finite element calculation and analysis of stiffnessvalve spring [J]. Internal combustion engine and power device ,2007,1:22-25
[5]. Renxian Li,.The foundation of finite element method [M].Beijing :National Defence Industry Press, 2010.
[6]. Bing Li, He Zhengjia, Chen Xuefeng .ANSYS Workbench simulation and optimized design[M]. Beijing: Tsinghua University Press .2011.
[7]. GuangyiPu.Thebasic tutorial and example explanation of ANSYS Workbench 12[M]. Beijing: China Water Conservancy and Hydropower Publishing House,2010.10.
[8]. ZhianSong .ANSYS and ANSYS WORKBENCH engineering application.Beijing: National Defense University Press ,2010.
[9]. Wei Fu.The finite element analysis of wind turbine generator hub [D].Chongqing : Chongqing University,2007.